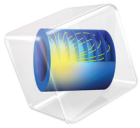


Acoustics Module

Application Library Manual

5.3a

COMSOL

Acoustics Module Application Library Manual

© 1998–2017 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/trademarks.

Version: COMSOL 5.3a

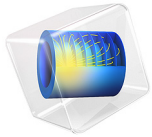
Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM020202



Absorptive Muffler

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example describes the pressure-wave propagation in a muffler for an internal combustion engine. The approach used here is generally applicable to analyzing the damping of propagating pressure waves as well as determining the transmission properties of a given system. The model shows how to analyze both inductive and resistive damping in pressure acoustics. The main output is the transmission loss for the frequency range 50 Hz–1500 Hz. It is represented both as a continuous curve and given in 1/3 octave bands.

See also the application Eigenmodes in a Muffler, which computes the propagating modes in the main chamber of the muffler.

Model Definition

The muffler—schematically shown in Figure 1—consists of a 24-liter resonator chamber with a section of the centered exhaust pipe included at each end. The model is first set up assuming that the chamber is empty. Secondly, it is lined with 15 mm of absorbing glass wool.

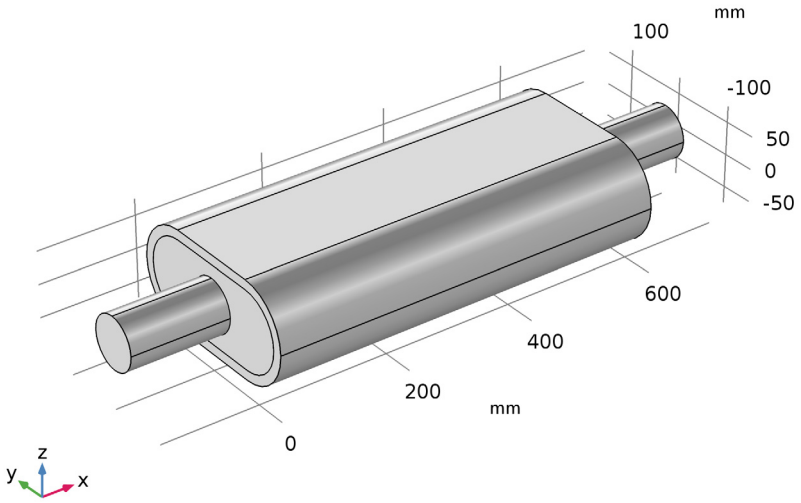


Figure 1: Geometry of the lined muffler, the liner is the outer layer in the main muffler volume. The exhaust fumes enter through the left pipe and exit through the right pipe.

DOMAIN EQUATIONS

This model solves the problem in the frequency domain using the Pressure Acoustics, Frequency Domain interface. The model equation is a slightly modified version of the Helmholtz equation for the acoustic pressure p:

$$\nabla \cdot \left(-\frac{\nabla p}{\rho}\right) - \frac{\omega^2 p}{c^2 \rho} = 0$$

where ρ is the density, c is the speed of sound, and ω is the angular frequency.

In the absorbing glass wool, modeled as a Poroacoustics domain, the damping enters the equation as a complex speed of sound, $c_c = \omega/k_c$, and a complex density, $\rho_c = k_c Z_c/\omega$, where k_c is the complex wave number and Z_c equals the complex impedance. This is a so-called equivalent fluid model for the porous domain where the losses are modeled in a homogenized way.

For a highly porous material with a rigid skeleton, the well-known model of Delany and Bazley estimates these parameters as functions of frequency and flow resistivity. Using the original coefficients of Delany and Bazley (Ref. 1), the expressions are

$$k_c = k_a \cdot \left(1 + 0.098 \cdot \left(\frac{\rho_a f}{R_f}\right)^{-0.7} - i \cdot 0.189 \cdot \left(\frac{\rho_a f}{R_f}\right)^{-0.595}\right)$$

$$Z_c = Z_a \cdot \left(1 + 0.057 \cdot \left(\frac{\rho_a f}{R_f}\right)^{0.734} - i \cdot 0.087 \cdot \left(\frac{\rho_a f}{R_f}\right)^{-0.732}\right)$$

where R_f is the flow resistivity, and where $k_a = \omega/c_a$ and $Z_a = \rho_a c_a$ are the free-space wave number and characteristic impedance of air, respectively. This is the default selected porous model in the Poroacoustics domain feature. Several porous models can be selected here depending on the situation at hand. You can find flow resistivities in tables (see, for example, Ref. 4) or by measuring it. For glass-wool-like materials, Bies and Hansen (Ref. 2) give an empirical correlation

$$R_f = \frac{3.18 \cdot 10^{-9} \cdot \rho_{ap}^{1.53}}{d_{av}^2}$$

where ρ_{ap} is the material's apparent density and d_{av} is the mean fiber diameter. This model uses a rather lightweight glass wool with $\rho_{ap} = 12$ kg/m^3 and $d_{av} = 10$ µm.

Note: The Delany-Bazley model is valid for values of $\frac{\rho_f f}{R_f}$ up to $\frac{\rho_f f}{R_f} \approx 1$, so the upper frequency limit of 1500 Hz ensures that the poroacoustics model applies.

BOUNDARY CONDITIONS

The boundary conditions are of three types.

- At the solid boundaries, which are the outer walls of the resonator chamber and the pipes, the model uses sound hard (wall) boundary conditions. The condition imposes that the normal velocity at the boundary is zero.
- The boundary condition at the inlet involves a combination of incoming and outgoing plane waves (see Ref. 3 and the documentation for further details). This boundary condition is always valid as long as the frequency is kept below the cutoff frequency for the second propagating mode in the tube. Above this frequency only the propagating plane wave component is modeled.
- At the outlet boundary, the model specifies a radiation condition for an outgoing plane wave.

Results and Discussion

The pressure distribution in the absorptive muffler without the lining material is shown in Figure 2 for the frequency $f = 1500$ Hz. From the figure it is seen that at this frequency not only longitudinal standing waves exist but also transverse modes are present.

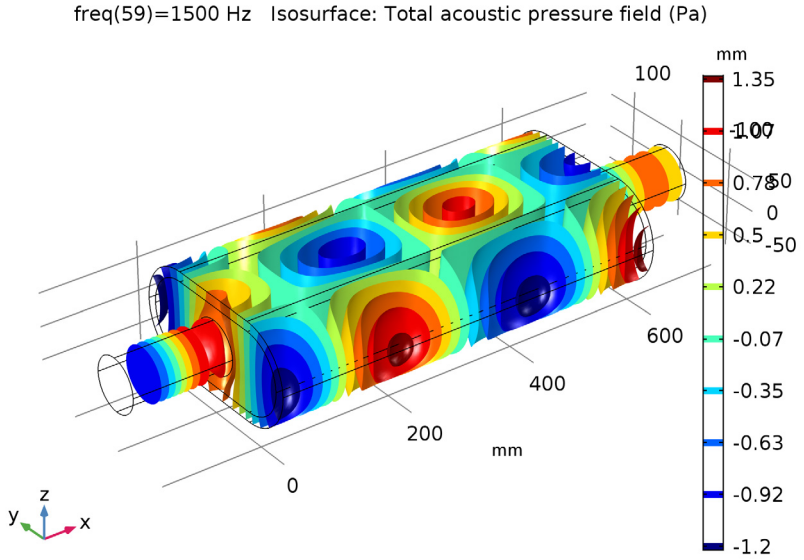


Figure 2: Pressure represented as an iso-barometric surface plot in the muffler.

An important parameter for a muffler is the transmission loss or attenuation. It is defined as the ratio between the incoming and outgoing acoustic energy. The attenuation or transmission loss d_w (in dB) of the acoustic energy is defined by

$$d_w = 10\log\left(\frac{w_{\text{in}}}{w_{\text{out}}}\right)$$

Here w_{in} and w_{out} denote the incoming power at the inlet and the outgoing power at the outlet, respectively. You can calculate each of these quantities as an integral over the corresponding inlet and outlet surface. Assuming plane wave propagation, the expressions are

$$w_{\text{out}} = \int_{\partial\Omega} \frac{|p|^2}{2\rho c} dA \qquad w_{\text{in}} = \int_{\partial\Omega} \frac{p_0^{\,2}}{2\rho c} dA$$

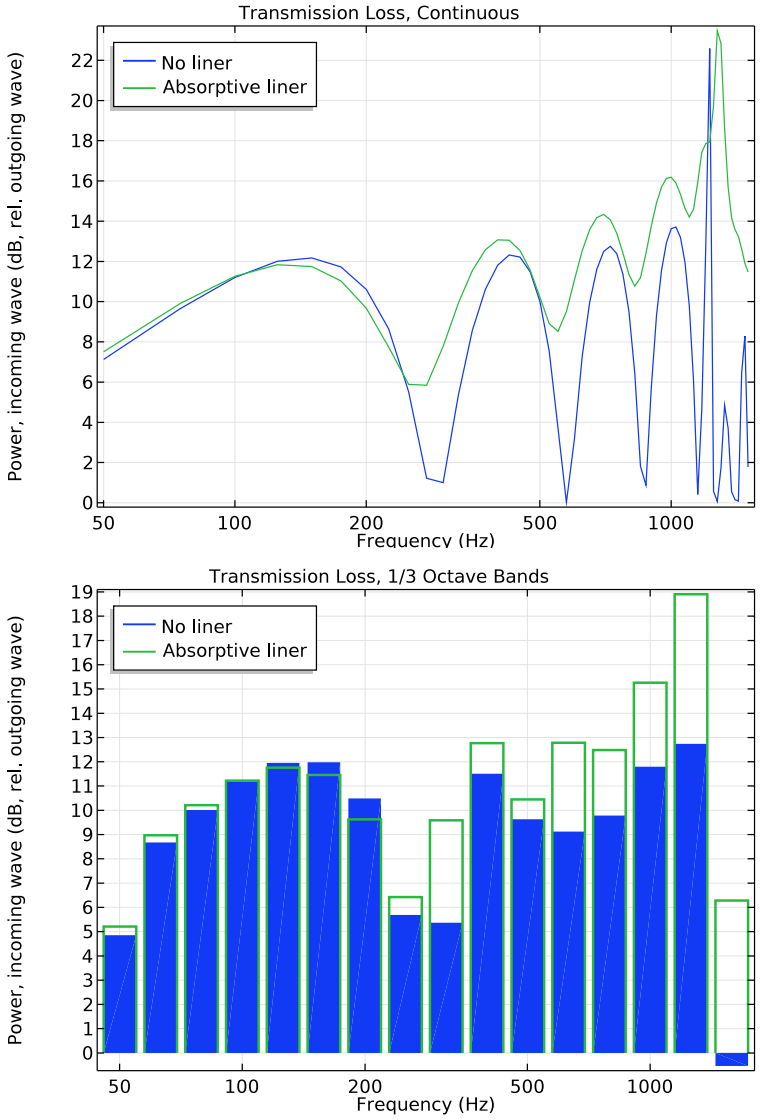


Figure 3: Comparison of the transmission loss as function of frequency for the empty muffler and the muffler with absorptive lining. (top) The transmission loss depicted as a continuous curve. The first four dips are due to longitudinal resonances. In the muffler with absorbing lining the dips are still present, but the general trend is that the higher the frequency, the better the damping. (bottom) The same data but depicted in 1/3 octave bands.

Figure 3 (top and bottom) shows the result of a parametric frequency study. The two graphs represent the case of an empty muffler without any absorbing lining material (blue lines) and the case with a layer of glass wool lining on the chamber's walls (green lines). In the top figure the transmission loss is depicted as a continuous curve (pure tone sweep) while it is depicted in 1/3 octave bands in the bottom figure. Both graphs are created using the Octave Band plot of the Acoustics Module.

The graph for the undamped muffler shows that damping works rather well for most low frequencies. At frequencies higher than approximately 1250 Hz, the behavior is more complicated and there is generally less damping. This is because, for such frequencies, the tube supports not only longitudinal resonances but also cross-sectional propagation modes. Not very far above this frequency a whole range of modes that are combinations of this propagation mode and the longitudinal modes participate, making the damping properties increasingly unpredictable. For an analysis of these modes, see the related model Eigenmodes in a Muffler. The glass-wool lining improves attenuation at the resonance frequencies as well as at higher frequencies.

The flow of energy in the muffler is shown in Figure 4 at 1500 Hz without the liner. The plot represents the intensity field depicted as streamlines. The intensity is per definition the time average of the intensity and thus represents the average energy flow in the system. Here from the inlet to the outlet. Change between solutions and frequencies to study and visualize the sound-absorbing properties of the muffler.

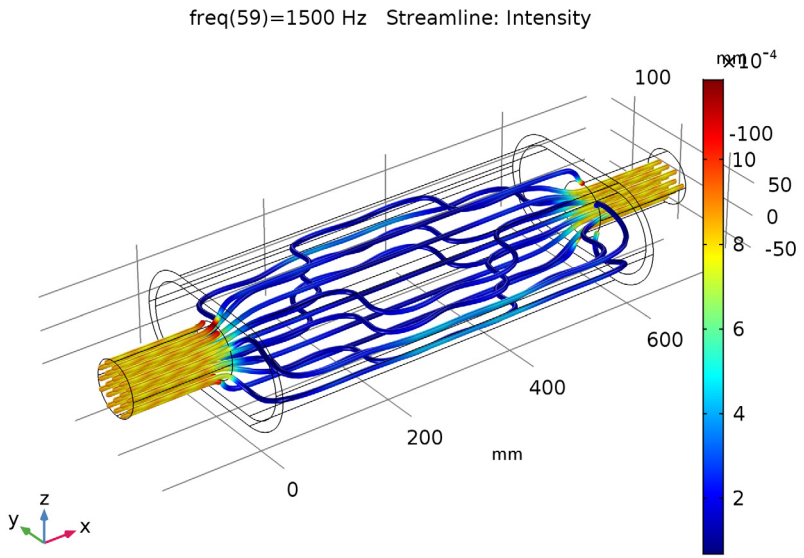


Figure 4: Intensity streamlines at 1500 Hz without the liner.

Notes About the COMSOL Implementation

This model uses the Pressure Acoustics interface available with the Acoustics Module. This interface has the Delany-Bazley coefficients built-in to the Poroacoustics feature. The Delany-Bazley is a so-called one parameter empirical model and the only damping parameter you need to supply is the flow resistivity.

The parametric solver provides results for a range of frequencies. The software computes integrals in the power expressions using boundary integration coupling operators, and it plots the resulting transmission loss versus frequency.

References

1. M.A. Delany and E.N. Bazley, "Acoustic Properties of Fibrous Absorbent Materials," *Appl. Acoust.*, vol. 3, pp. 105–116, 1970.

2. D.A. Bies and C.H. Hansen, "Flow Resistance Information for Acoustical Design," *Appl. Acoust.*, vol. 13, issue 5, Sept./Oct., pp. 357–391, 1980.

3. D. Givoli and B. Neta, “High-order Non-reflecting Boundary Scheme for Time-dependent Waves,” *J. Comp. Phys.*, vol. 186, pp. 24–46, 2003.

4. T. J. Cox and P. D’Antonio, *Acoustic Absorbers and Diffusers*, 2nd ed., Taylor and Francis, 2009.

Application Library path: `Acoustics_Module/Automotive/absorptive_muffler`

Modeling Instructions

The instructions take you through two versions of the model, first one with a completely hollow chamber with rigid walls, then one where the chamber is lined with glass wool.

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `absorptive_muffler_parameters.txt`.

The parameters define the physical values and the geometrical dimensions of the system. The geometry is now parameterized and simply changing the value of a dimension in the parameters list will update the geometry automatically.

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)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 Click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `W`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Fillet 1 (fil1)

1 On the **Work Plane** toolbar, click **Fillet**.

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

3 On the object **r1**, select Points 1–4 only.

4 In the **Settings** window for **Fillet**, locate the **Radius** section.

5 In the **Radius** text field, type `H/2`.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `W-2*D`.

4 In the **Height** text field, type `H-2*D`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Fillet 2 (fil2)

1 On the **Work Plane** toolbar, click **Fillet**.

2 On the object **r2**, select Points 1–4 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `(H-2*D)/2`.

5 On the **Work Plane** toolbar, click **Build All**.

Work Plane 1 (wp1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

2 On the **Workplane** toolbar, click **Close**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (mm)
L

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

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

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

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

5 Locate the **Position** section. In the **x** text field, type `-L_io`.

6 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

7 In the **x** text field, type `1`.

8 In the **z** text field, type `0`.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

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

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

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

5 Locate the **Position** section. In the **x** text field, type `L`.

6 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

7 In the **x** text field, type `1`.

8 In the **z** text field, type `0`.

9 Click **Build All Objects**.

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

The geometry should look like the one depicted in Figure 1.

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Inlet` in the **Label** text field.

3 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 1 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Outlet` in the **Label** text field.

3 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 28 only.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 From the **Selection** list, choose **Inlet**.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 From the **Selection** list, choose **Outlet**.

You have now defined the integration coupling operators `intop1` and `intop2`, for integration over the inlet and outlet respectively. Use these operators in defining the in- and outgoing power, as defined in the Results and Discussion section.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`w_in`	`intop1(p_in^2/(2*acpr.rho*acpr.c))`	W	`Power of the incoming wave`
`w_out`	`intop2(abs(p)^2/(2*acpr.rho*acpr.c))`	W	`Power of the outgoing wave`

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

By default the first material you add applies on all domains so you do not need to alter the geometric scope settings.

In the second version of this model, you will use a lining material in Domain 2. Add such a material with an empty selection.

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

MATERIALS

Material 2 (mat2)

In the **Settings** window for **Material**, type `Absorptive Liner` in the **Label** text field.

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 **Model Inputs** section.

3 In the T text field, type `T0`.

4 In the p_A text field, type `p0`.

Plane Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundaries 1 and 28 only.

Incident Pressure Field 1

1 On the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 In the **Settings** window for **Incident Pressure Field**, locate the **Boundary Selection** section.

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

4 Locate the **Incident Pressure Field** section. In the p_0 text field, type `p_in`.

5 From the c list, choose **From material**.

6 From the **Material** list, choose **Air (mat1)**.

You have now specified the plane wave radiation condition to be active on both the inlet and outlet boundaries, but with an incident wave on the inlet. The remaining boundaries by default use the sound hard condition.

Now, add a poroacoustics model for the absorptive liner domain. You will deactivate this domain when configuring the first study step.

Poroacoustics 1

1 On the **Physics** toolbar, click **Domains** and choose **Poroacoustics**.

2 Select Domain 2 only.

3 In the **Settings** window for **Poroacoustics**, locate the **Model Input** section.

4 In the T text field, type `T0`.

5 Locate the **Porous Matrix Properties** section. From the **Porous elastic material** list, choose **Absorptive Liner (mat2)**.

The material data for the flow resistivity will now be picked up from the Absorptive Liner material. Enter the data in the material.

MATERIALS

Absorptive Liner (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Absorptive Liner (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Flow resistivity	Rf	R_f	Pa·s/m²	Poroacoustics model

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

To more easily locate and select a boundary use the wireframe rendering option for the graphics.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

3 Select Boundaries 6, 9, and 16 only.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

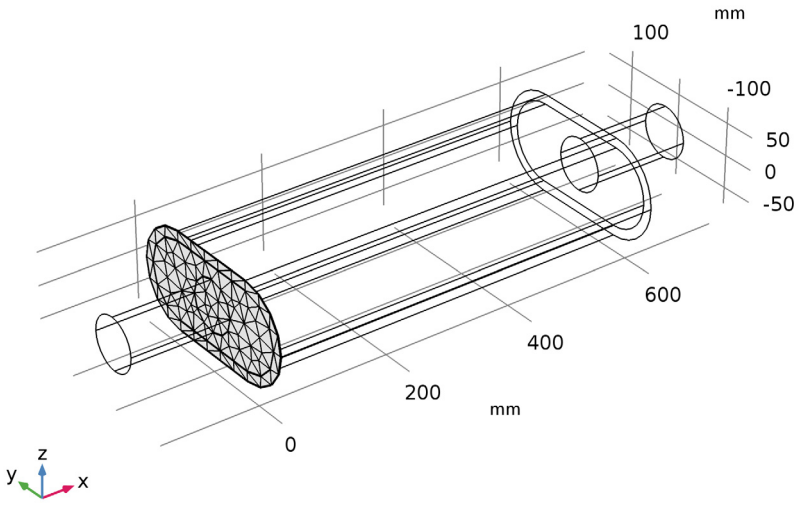
2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `343[m/s]/1500[Hz]/5`.

The global maximum element size is set equal to the minimal wavelength divided by 5, that is, $\lambda_{min}/5 = c/f_{max}/5$, where c is the speed of sound.

5 Click **Build All**.



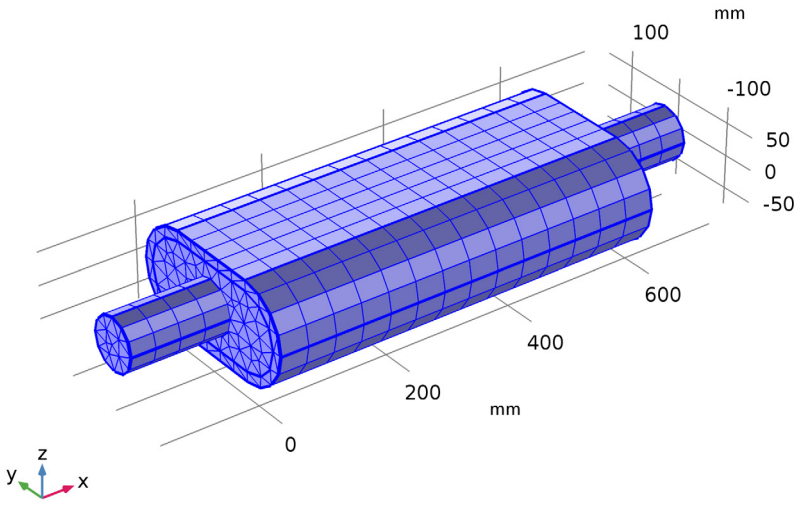
Swept 1

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

Now, simply push the **Build All** button and the mesher will automatically detect source and destination boundaries for the swept mesh.

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

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



STUDY 1

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

2 In the **Settings** window for **Study**, type `Study 1 - No Liner` in the **Label** text field.

STUDY 1 - NO LINER

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - No Liner** 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 `range(50,25,1500)`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify physics tree and variables for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)>Poroacoustics 1**.

6 Click **Disable**.

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

RESULTS

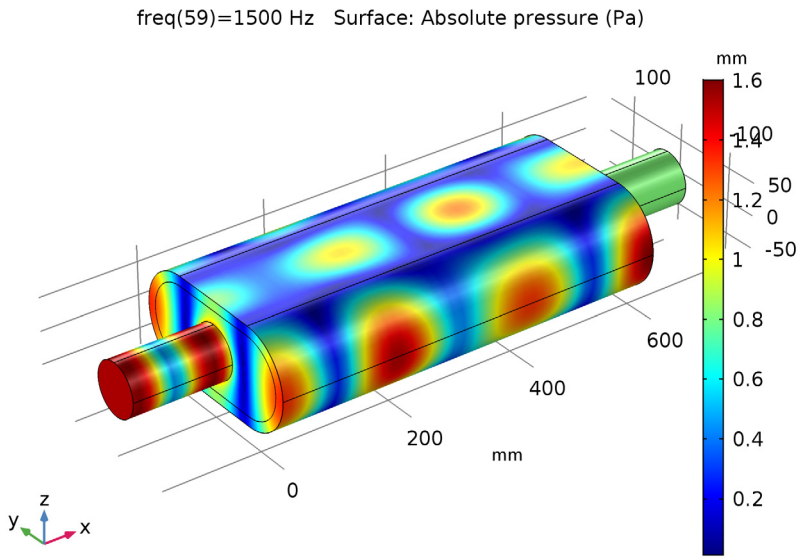
Acoustic Pressure (acpr)

The first one of the default plots shows the pressure distribution on the walls of the muffler at the highest frequency, 1500 Hz. To get a better view of the standing wave pattern, you can plot the norm of the pressure instead of the real part of the pressure.

Surface 1

1. In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** 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>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp - Absolute pressure**.
3. On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

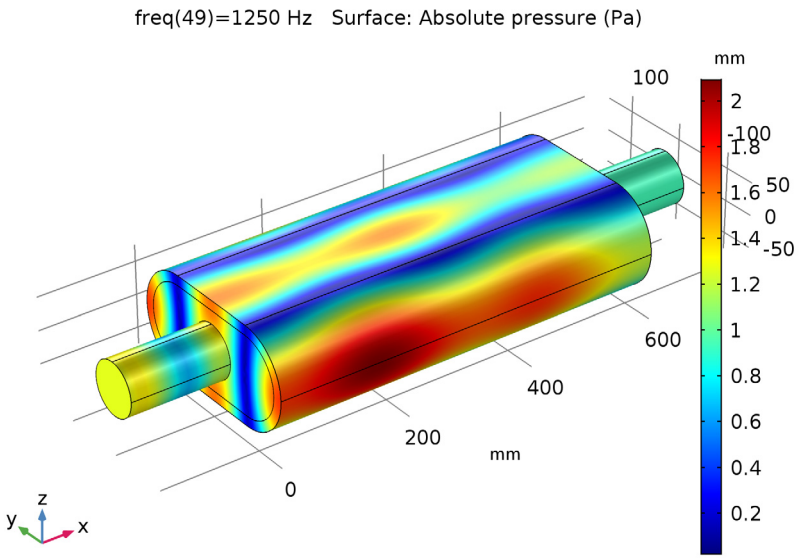
Acoustic Pressure (acpr)



The pattern is very different at different frequencies. See for example what happens at 1250 Hz.

1. In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Parameter value (freq (Hz))** list, choose **1250**.

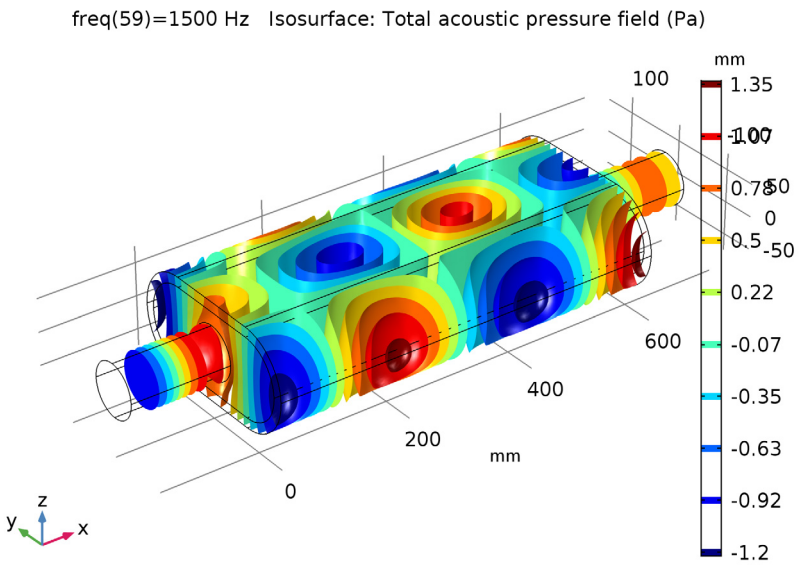
4 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.



At 1250 Hz, the absolute value of the pressure does not vary much with the x-coordinate. The reason is that this is just higher than the cutoff frequency for the first symmetric propagating mode, which is excited by the incoming wave. For a separate analysis of the propagating modes in the chamber, see the Eigenmodes in a Muffler model.

The two other default plot groups show the sound pressure level on the wall surface and the pressure inside the muffler as isosurfaces.

Acoustic Pressure, Isosurfaces (acpr)



Proceed to plot the transmission loss of the muffler system. Use the Octave Band plot, which makes it possible to plot any transfer function both as band plots and as continuous curves (sweeps).

1D Plot Group 4

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

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

Octave Band 1

1 On the **Transmission Loss, Continuous** toolbar, click **More Plots** and choose **Octave Band**.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `w_in/w_out`.

5 From the **Expression type** list, choose **Transfer function**.

6 Locate the **Plot** section. From the **Style** list, choose **Continuous**.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
No liner

10 On the **Transmission Loss, Continuous** toolbar, click **Plot**.

The plot should be a reproduction of the blue curve in Figure 3.

Proceed to solve the model including a layer of absorptive glass wool on the muffler line. Continue working from where you left off with the model developed thus far and add a second study to keep your existing results intact.

ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

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

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

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - Absorptive Liner` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 2 - ABSORPTIVE LINER

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - Absorptive Liner** 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 `range(50,25,1500)`.

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

RESULTS

Acoustic Pressure, Isosurfaces (acpr)

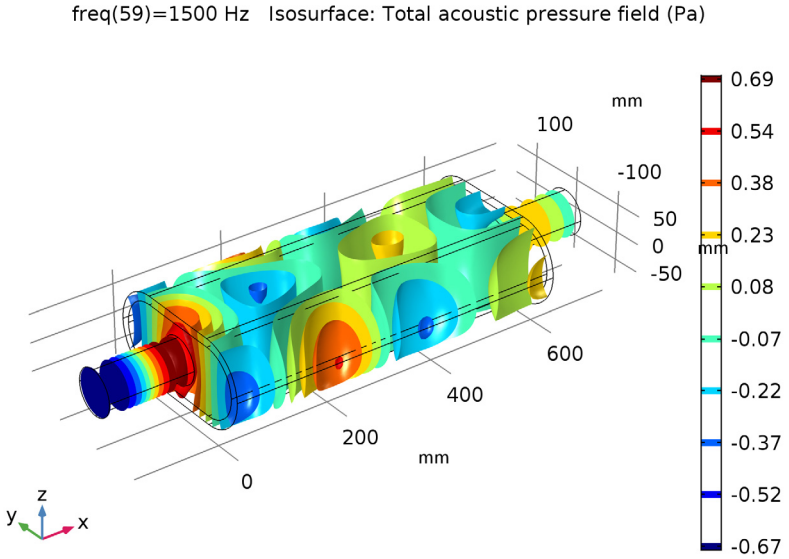
You chose not to have new default plots generated. Once the solution process is finished you can use the existing plot groups and just switch the data set to see how the damping material affects the solution.

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Isosurfaces (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - Absorptive Liner/Solution 2 (sol2)**.

4 On the **Acoustic Pressure, Isosurfaces (acpr)** toolbar, click **Plot**.



At 1500 Hz, the pressure in the chamber is much lower than before.

Proceed to study how the transmission loss has changed with the addition of the lining. Duplicate the first plot and select the new data set, first do a bit of formatting.

Transmission Loss, Continuous

1 In the **Model Builder** window, under **Results** click **Transmission Loss, Continuous**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

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

4 In the **Title** text area, type `Transmission Loss, Continuous`.

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

6 In the associated text field, type `Power, incoming wave (dB, rel. outgoing wave)`.

7 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Octave Band 2

1 In the **Model Builder** window, under **Results>Transmission Loss, Continuous** right-click **Octave Band 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Study 2 - Absorptive Liner/Solution 2 (sol2)**.

4 Click to expand the **Legends** section. In the table, enter the following settings:

Legends
`Absorptive liner`

5 On the **Transmission Loss, Continuous** toolbar, click **Plot**.

The plot should look like that in Figure 3 top.

Duplicate the Transmission Loss plot and change the format to 1/3 octave bands.

Transmission Loss, Continuous 1

1 In the **Model Builder** window, under **Results** right-click **Transmission Loss, Continuous** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Transmission Loss, 1/3 Octave Bands` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Transmission Loss, 1/3 Octave Bands`.

Octave Band 1

1 In the **Model Builder** window, expand the **Results>Transmission Loss, 1/3 Octave Bands** node, then click **Octave Band 1**.

2 In the **Settings** window for **Octave Band**, locate the **Plot** section.

3 From the **Style** list, choose **1/3 octave bands**.

Octave Band 2

1 In the **Model Builder** window, under **Results>Transmission Loss, 1/3 Octave Bands** click **Octave Band 2**.

2 In the **Settings** window for **Octave Band**, locate the **Plot** section.

3 From the **Style** list, choose **1/3 octave bands**.

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Type** list, choose **Curve**.

5 In the **Width** text field, type 2.

6 On the **Transmission Loss, 1/3 Octave Bands** toolbar, click **Plot**.

The plot should look like that in Figure 3 bottom.

Now, create a plot that represents the energy flux through the muffler system. Use streamlines that follow the intensity vector. You can change between solutions and frequencies to study and visualize the muffler's sound-absorbing properties.

3D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

Streamline 1

1 Right-click **Intensity** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Intensity>acpr.Ix,acpr.Iy,acpr.Iz - Intensity**.

3 Select Boundary 1 only.

4 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

5 In the **Tube radius expression** text field, type 2.

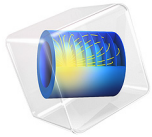
Color Expression 1

1 Right-click **Results>Intensity>Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Pressure Acoustics, Frequency Domain>Intensity>acpr.I_mag - Intensity magnitude**.

3 On the **Intensity** toolbar, click **Plot**.

This should reproduce Figure 4.



Acoustic Cloaking

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Recent studies (see Ref. 1) have shown the feasibility of manufacturing "invisibility cloaks" using layered metamaterials. Draping an object in a cloak makes it transparent, or nearly transparent, to electromagnetic waves.

The same principle can be used in acoustics to hide an object from acoustic radiation. This example looks at sound scattering from a plane wave incident on a hard-walled cylinder. Results with the cylinder clad in a cloak (described in Ref. 2) are compared with a solution without the cloak.

Model Definition

A cylinder with a 1 m radius is surrounded by a metamaterial cloak consisting of 50 layers of two alternating fluid-like materials similar to those in Ref. 2. Each layer has a thickness of 2 cm and its material properties are as follows:

$$\rho_1 = \frac{r + \sqrt{2rR_1 - R_1^2}}{r - R_1}\rho_b$$

$$c_1 = \frac{R_2 - R_1}{R_2}\frac{r}{r - R_1}c_b$$

$$\rho_2 = \rho_b^2/\rho_1$$

$$c_2 = c_1$$

In this formula:

- ρ_1, ρ_2, c_1, and c_2 are the density and speed of sound of materials 1 and 2;
- $\rho_b = 1.25$ kg/m^3 and $c_b = 343$ m/s, are the density and speed of sound in the outside medium, which is air;
- R_1 and R_2 are the inner and outer radius of the cloak; and
- r is the distance to the cylinder axis.

The model considers a frequency of $f = 200$ Hz and solves the Helmholtz equation for the total acoustic pressure:

$$\nabla \cdot \left(-\frac{\nabla p_t}{\rho}\right) - \frac{\omega^2 p_t}{\rho c^2} = 0$$

Here, p_t is the (total) acoustic pressure. To describe an incident plane wave traveling in the x-direction, a background field p_b is defined as $e^{-ik_b x}$, where $k_b = 2\pi f/c_b$ is the propagation constant in the background medium. The equation is solved for the scattered field p_s, using the definition

$$p_t \equiv p_b + p_s$$

The geometric mirror symmetry of the problem is used to reduce the modeling domain half of the full geometry (see Figure 1). The background medium is truncated with a cylindrical radiation condition (see the *Acoustics Module User's Guide* for details about the theory) on its outer boundaries.

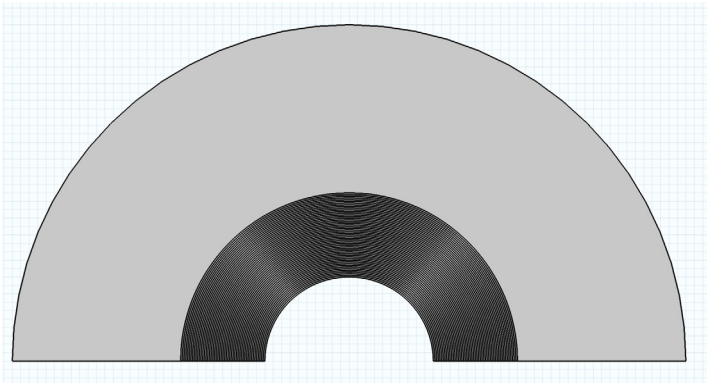


Figure 1: The center cylinder cut out from the geometry is covered by the cloak—the 50 layers are too thin to see clearly in this image. Outside the cloak, the background material is truncated at a 4 m distance from the center.

Results and Discussion

The first version of this model considers the case of an uncloaked cylinder, surrounded only by air. The incident pressure wave is scattered in all directions and creates a standing wave pattern as shown by the absolute pressure in Figure 2. Figure 3 shows the same plot with the cloak present. Note the different color ranges. Without the cloak, the absolute pressure peaks at 1.93 Pa, which is almost double the amplitude of the incident wave. With the same range, the pressure variations with the cloak present would hardly be visible.

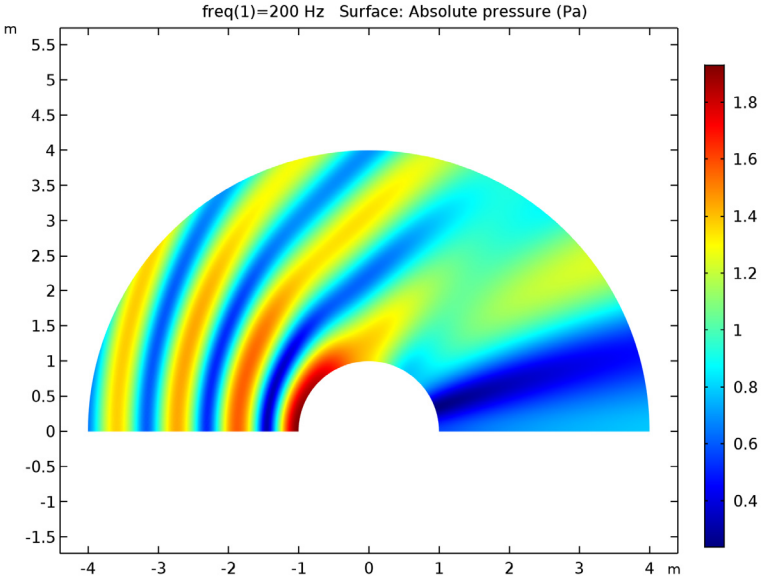


Figure 2: Absolute value of the local pressure without the cloak.

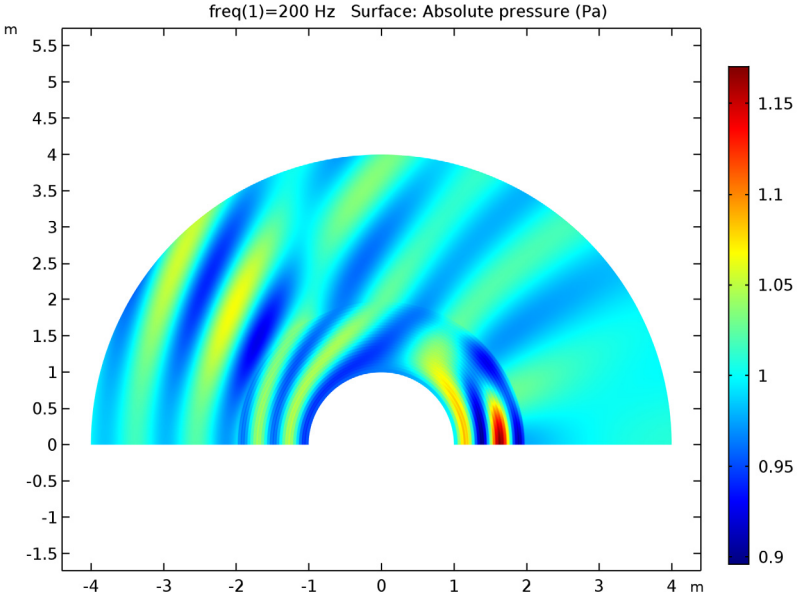


Figure 3: Absolute pressure with the cloak present.

Another way to visualize the effect of the cloak is to look at the instantaneous pressure. Figure 4 shows that outside the cloak the pressure deviates only slightly from that of a perfectly plane wave.

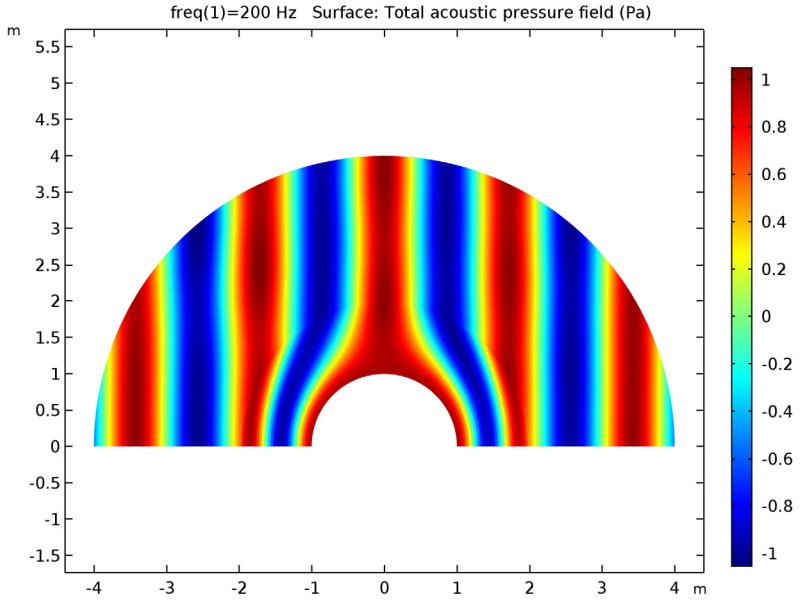


Figure 4: Instantaneous acoustic pressure distribution.

The instructions and results that follow, deal only with a frequency of 200 Hz, but you can modify the model and use a different frequency, or sweep over a range of frequencies, to see how this affects the cloaking.

References

1. Several COMSOL Conference papers:
http://www.comsol.com/papers/5478/
http://www.comsol.com/papers/9831/
http://www.comsol.com/papers/5463/

2. D. Torrent and J. Sánchez-Dehesa1, "Acoustic Cloaking in Two Dimensions: a Feasible Approach," *New Journal of Physics*, vol. 10, p. 063015, 2008.

Application Library path: `Acoustics_Module/Tutorials/acoustic_cloaking`

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**.
2. In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6. Click **Done**.

GEOMETRY 1

The geometry of this model is simple but a bit repetitive to create, due to the large number of circles involved. You will therefore import it inserting a sequence from the geometry file. The instructions to the geometry can be found in the appendix at the end of this document.

1. On the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `acoustic_cloaking_geom_sequence.mph`.
3. On the **Geometry** toolbar, click **Build All**.
4. Click the **Zoom Extents** button on the **Graphics** toolbar.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
rhob	1.25[kg/m^3]	1.25 kg/m³	Density, background material
cb	343[m/s]	343 m/s	Speed of sound, background material
R1	1[m]	1 m	Inner cloak radius
R2	2[m]	2 m	Outer cloak radius

DEFINITIONS

The material properties in the cloaking involve dependencies on the inner and outer radius, the local distance from the center, as well as the properties of the surrounding background fluid. It is convenient to define these expressions as variables.

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
r	sqrt(x^2+y^2)	m	Radial coordinate
rho1	rhob*(r+sqrt(2*r*R1-R1^2))/(r-R1)	kg/m³	Density, Material 1
c1	cb*(R2-R1)/R2*r/(r-R1)	m/s	Speed of sound, Material 1
rho2	rhob^2/rho1	kg/m³	Density, Material 2
c2	c1	m/s	Speed of sound, Material 2

MATERIALS

To fully appreciate the effect of the cloaking, first run a study without it. This means all domains will be made of air.

Material 1 (mat1)

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 Air in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Density	rho	rhob	kg/m³	Basic
Speed of sound	c	cb	m/s	Basic

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

First, specify the background field. It is defined by selecting the speed of sound `c` which defines the propagation constant (the wavenumber) as `omega/c`. Here, `c` is taken from the material air such that it is constant.

Background Pressure Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Background Pressure Field**.

2 In the **Settings** window for **Background Pressure Field**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Background Pressure Field** section. In the p_0 text field, type `1`.

5 From the *c* list, choose **From material**.

6 From the **Material** list, choose **Air (mat1)**.

The default condition on the exterior boundaries is Sound hard. Staying with this on the (not yet) hidden object means you are treating it as having a perfectly reflective surface. The Sound Hard condition also applies to the symmetry boundaries. The only boundaries where you need a different condition are the ones where you want the scattered wave to radiate out from the model.

Cylindrical Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Cylindrical Wave Radiation**.

2 Select Boundaries 103 and 206 only.

MESH 1

With the default free mesh everywhere, you would create more elements than necessary because of the many thin layers in the cloak. It is more efficient to combine a free mesh in the air with a mapped mesh in the cloak.

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **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 **All domains**Remove Domain 1 from the selection.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Boundaries 154 and 155 only.

Zooming in a little will make it easier to select these boundaries.

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

4 In the **Number of elements** text field, type 30.

Size 1

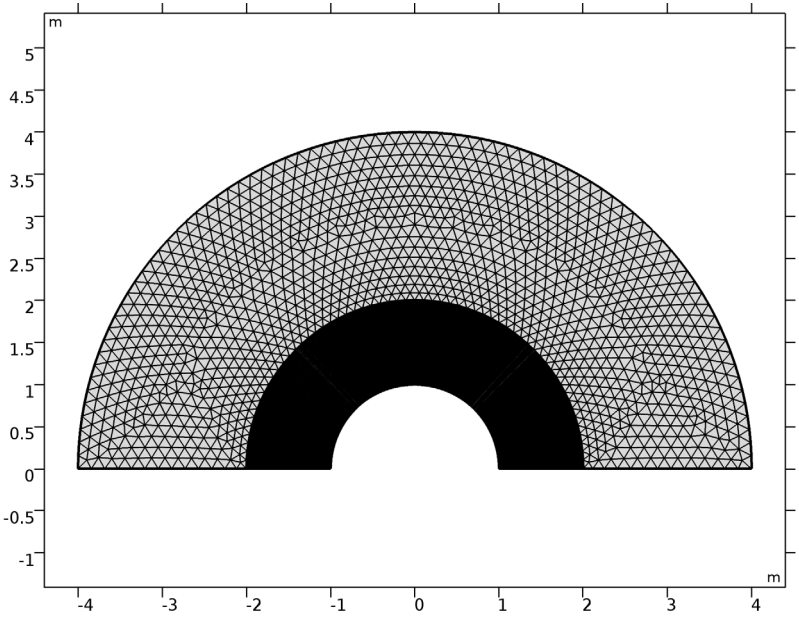
1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.

2 Right-click **Free Triangular 1** and choose **Size**.

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

4 From the **Predefined** list, choose **Extra fine**.

5 Click **Build All**.



STUDY 1

Step 1: Frequency Domain

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

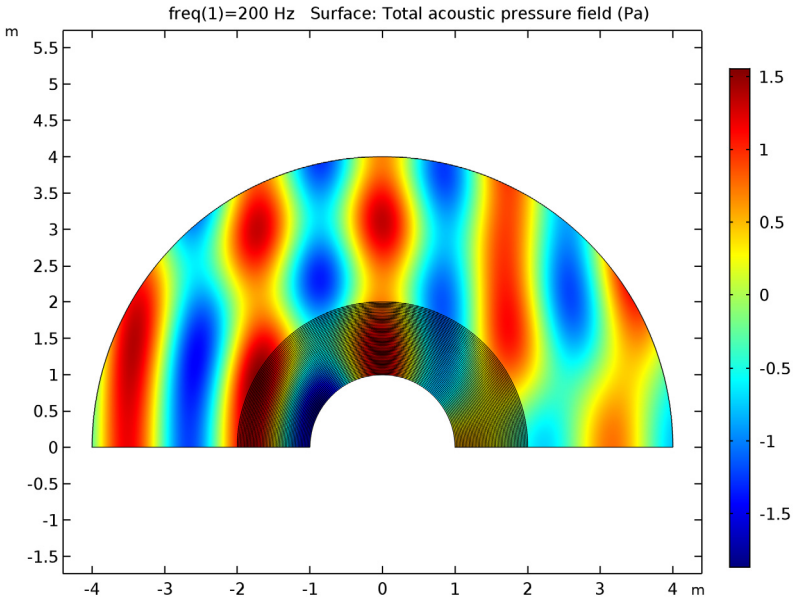
2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type 200.

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

RESULTS

Acoustic Pressure (acpr)



The default plot shows the total acoustic pressure field. The wavy pattern and the fact that the field is smaller behind the object shows that much of the wave gets reflected. To avoid showing the contours of the cloak, you can turn off the edge visualization.

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

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

3 Clear the **Plot data set edges** check box.

The second default plot group contains the plot settings to display the sound pressure level.

You can get a better view of the reflections by plotting the absolute value of the pressure.

Surface 1

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

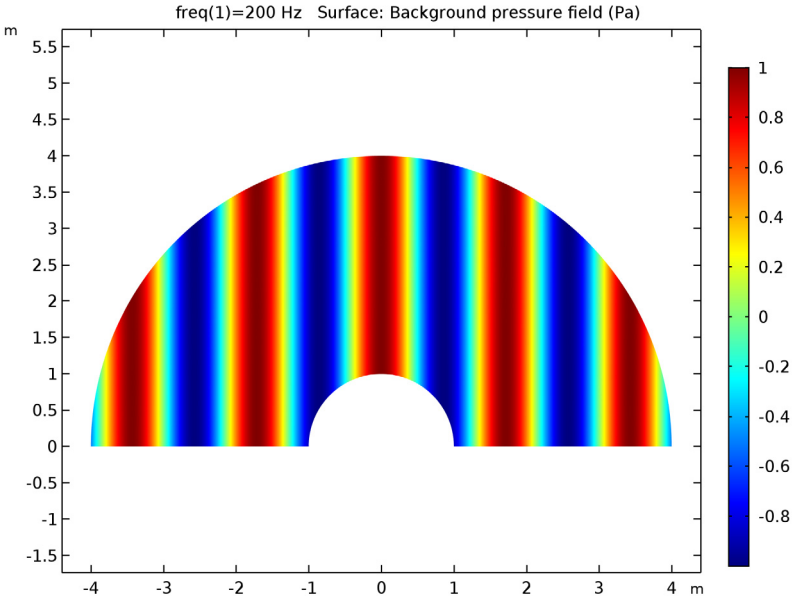
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp - Absolute pressure**.

3 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

Your plot should now look like Figure 2. You can verify that your background field looks correct by plotting it.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.p_b - Background pressure field**.

5 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.



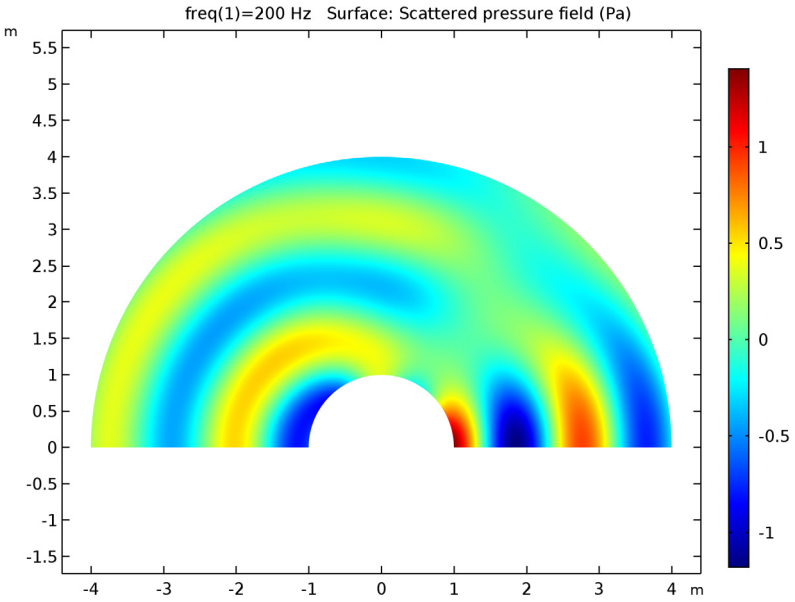
The plot now indeed shows your x-directed plane wave with its 1 Pa amplitude.

Finally, try visualizing the scattered pressure field, which is the perturbation caused by the cylinder - had it not been present, the scattered field would have been zero. When

you include a background field in a model, the scattered field is what the software actually computes.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Pressure Acoustics, Frequency Domain> Pressure and sound pressure level>acpr.p_s - Scattered pressure field**.

7 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.



MATERIALS

Now add the cloak.

Material 2 (mat2)

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 `Mat1` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Explicit Selection 1**.

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

Property	Variable	Value	Unit	Property group
Density	rho	rho1	kg/m³	Basic
Speed of sound	c	c1	m/s	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type Mat2 in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Explicit Selection 2**.

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

Property	Variable	Value	Unit	Property group
Density	rho	rho2	kg/m³	Basic
Speed of sound	c	c2	m/s	Basic

ROOT

Next, add a new study to the model. Using this study when computing the solution with the cloak present lets you keep your current results for a comparison.

ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

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

5 On 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 In the **Frequencies** text field, type 200.

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

RESULTS

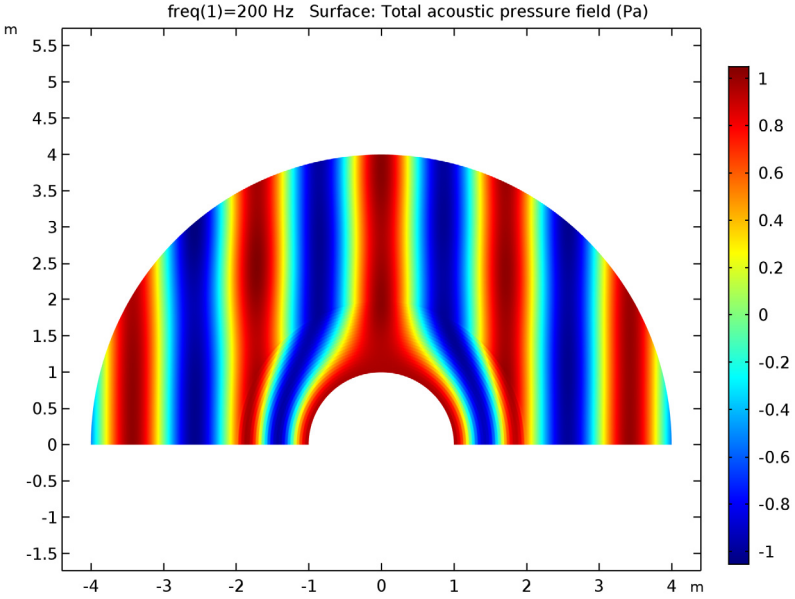
Acoustic Pressure (acpr) 1

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr) 1**.

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

3 Clear the **Plot data set edges** check box.

4 On the **Acoustic Pressure (acpr) 1** toolbar, click **Plot**.



The total pressure field is now an only slightly distorted version of the background field. This indicates that the reflections are minor and the object well hidden.

For a better quantitative measure and to reproduce Figure 3, plot the absolute pressure.

Surface 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (acpr) 1** 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 **Model>Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp - Absolute pressure**.

3 On the **Acoustic Pressure (acpr) 1** toolbar, click **Plot**.

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>2D**.

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **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 180.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	2
Layer 2	0.02
Layer 3	0.02
Layer 4	0.02
Layer 5	0.02
Layer 6	0.02
Layer 7	0.02
Layer 8	0.02
Layer 9	0.02
Layer 10	0.02
Layer 11	0.02
Layer 12	0.02
Layer 13	0.02
Layer 14	0.02
Layer 15	0.02
Layer 16	0.02
Layer 17	0.02
Layer 18	0.02
Layer 19	0.02
Layer 20	0.02
Layer 21	0.02

Layer name	Thickness (m)
Layer 22	0.02
Layer 23	0.02
Layer 24	0.02
Layer 25	0.02
Layer 26	0.02
Layer 27	0.02
Layer 28	0.02
Layer 29	0.02
Layer 30	0.02
Layer 31	0.02
Layer 32	0.02
Layer 33	0.02
Layer 34	0.02
Layer 35	0.02
Layer 36	0.02
Layer 37	0.02
Layer 38	0.02
Layer 39	0.02
Layer 40	0.02
Layer 41	0.02
Layer 42	0.02
Layer 43	0.02
Layer 44	0.02
Layer 45	0.02
Layer 46	0.02
Layer 47	0.02
Layer 48	0.02
Layer 49	0.02
Layer 50	0.02
Layer 51	0.02

Delete Entities 1 (del1)

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

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

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

4 On the object **c1**, select Domain 52 only.

Ignore Edges 1 (ige1)

1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 In the **Settings** window for **Ignore Edges**, locate the **Input** section.

3 Clear the **Ignore adjacent vertices** check box.

4 Click the **Select Box** button on the **Graphics** toolbar.

5 On the object **fin**, select Boundaries 52–102 only.

6 Click **Build Selected**.

DEFINITIONS

Explicit Selection 1 (sel1)

On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

GEOMETRY 1

Explicit Selection 1 (sel1)

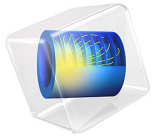
On the object **ige1**, select Domains 3, 5, 7, 9, 11, 13, 15, 17, 19, 21, 23, 25, 27, 29, 31, 33, 35, 37, 39, 41, 43, 45, 47, 49, and 51 only.

Explicit Selection 2 (sel2)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 On the object **ige1**, select Domains 2, 4, 6, 8, 10, 12, 14, 16, 18, 20, 22, 24, 26, 28, 30, 32, 34, 36, 38, 40, 42, 44, 46, 48, and 50 only.

3 On the **Geometry** toolbar, click **Build All**.



Acoustic Levitator

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

An ultrasonic standing wave levitator, also called acoustic levitator, is a device used for levitating fluid and solid particles in an acoustic field. The standing acoustic waves exert an acoustic radiation force on the particles. The force is a second order effect and stems from a combination of the time averaged pressure and inertial interaction between the particles and the acoustic field.

By levitating a particle it is possible to study, for example, its drying kinetics under different external conditions as temperature and humidity (see Ref. 1). The levitator has also been used to study combustion processes, the formation of ice particles and snow flakes, and is also used as an acoustic tweezer in micro-gravity for example in space missions.

The model is that of a simplified 2D acoustic levitator geometry driven at a constant frequency. Small elastic particles are released uniformly in the standing acoustic field and their path is determined when influenced by the acoustic radiation force, viscous drag, and gravity.

Note: This application requires the Particle Tracing module.

Model Definition

The levitator consists of a transducer of width D_t and a concave reflector of width D_r and curvature $1/R$. The distance between the reflector and the transducer is H. The driving frequency of the system is f_0 = 58 kHz and the system is filled with air at 20 ^{o}C. The geometry is shown in Figure 1. Standard values for the dimensions of the levitator are given in Ref. 2 and are as follows in the model:

TABLE 1: STANDARD DIMENSIONS AND PHYSICAL VALUES

SYMBOL	VALUE	DESCRIPTION
c_0	343 m/s	Speed of sound
f_0	58 kHz	Driving frequency
λ_0	5.9 mm	Wavelength
$D_t = 2\lambda_0$	11.8 mm	Transducer width
$D_r = 3\lambda_0$	17.7 mm	Reflector width
$H = 5\lambda_0/2$	14.8 mm	Reflector-transducer distance
$R = H$	14.8 mm	Radius of curvature of reflector

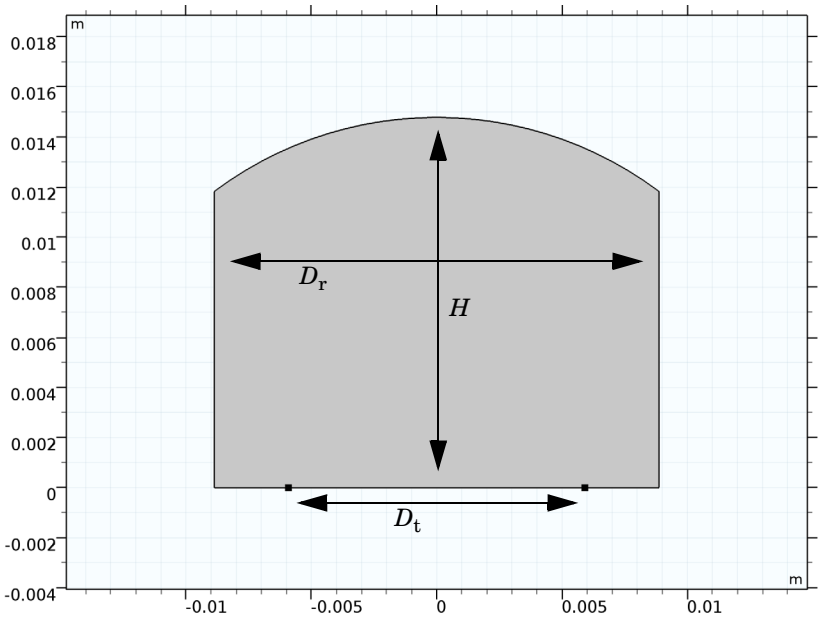


Figure 1: Levitator geometry.

In this model the particle diameter d_p is assumed to be small compared to the wavelength λ_0 as they are modeled as point particles. The condition $d_p << \lambda_0$ is also necessary for the acoustic radiation force term to be physically correct. The second condition is that the particle diameter d_p should be larger than the acoustic viscous boundary layer thickness δ_v. In the present system δ_v is of the order 10 µm. Table 2 lists the particle properties uses in this model.

TABLE 2: PARTICLE PROPERTIES

SYMBOL	VALUE	DESCRIPTION
d_p	0.6 mm	Particle diameter
ρ_p	500 kg/m^3	Particle density
K_p	2.2 GPa	Particle bulk modulus

Results and Discussion

The particles are released in the standing acoustic wave field in the levitator. The pressure field is shown in Figure 2, with plane wave radiation conditions at the open boundaries.

The corresponding sound pressure level in the system is shown in Figure 3.

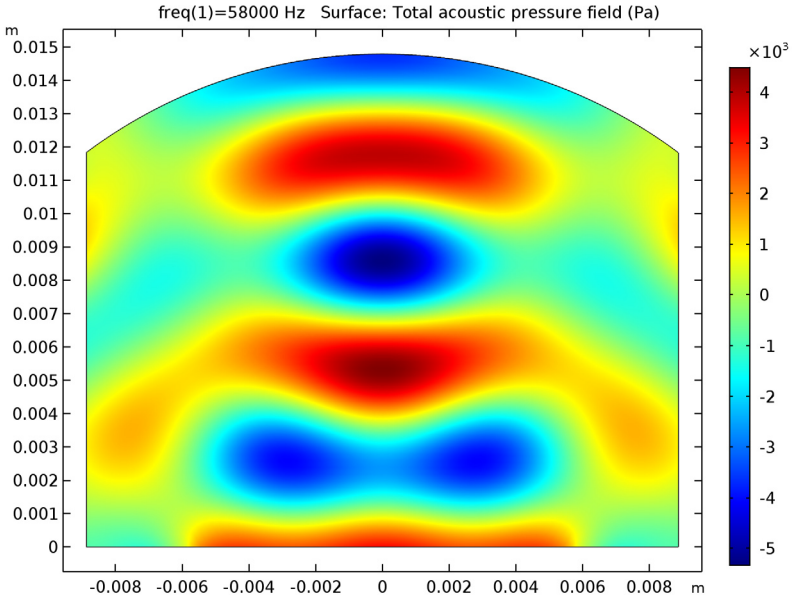


Figure 2: Real part of the pressure field, Re(p).

Note that the sound pressure level reaches as much as 170 dB SPL. Fortunately for the practical application of this device, this is outside the human auditory range. By tuning the amplitude of the normal acceleration of the transducer `a0` (under parameters in the model tree), it is possible to determine the value at which the radiation force is no longer large enough to levitate the particles.

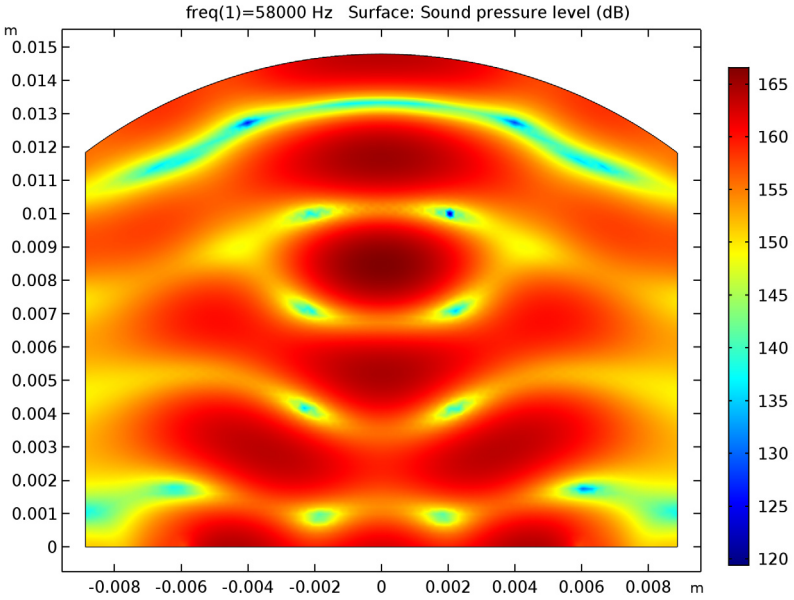


Figure 3: Sound pressure levels in the levitator.

The positions of the particles at $t = 0.01$ s and $t = 0.3$ s are shown in Figure 4

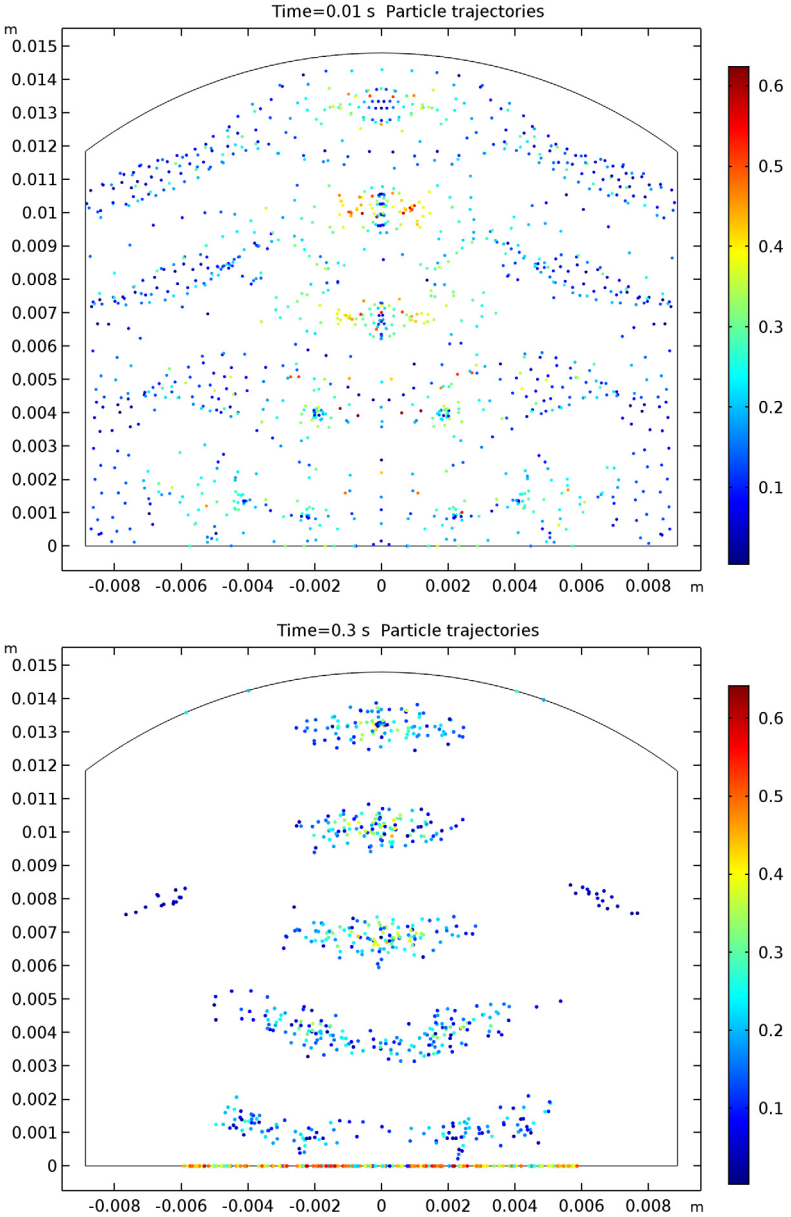


Figure 4: Positions of the particles at t = 0.01 s (top) and t = 0.3 s (bottom). The colors refer to the instantaneous particle velocities.

Finally, the particle position and their trajectories is shown in Figure 5 using the magnitude of the radiation potential as color legend. Since the radiation force is defined as the gradient of the radiation potential, it is clear that the particles tend toward the lowest potential value.

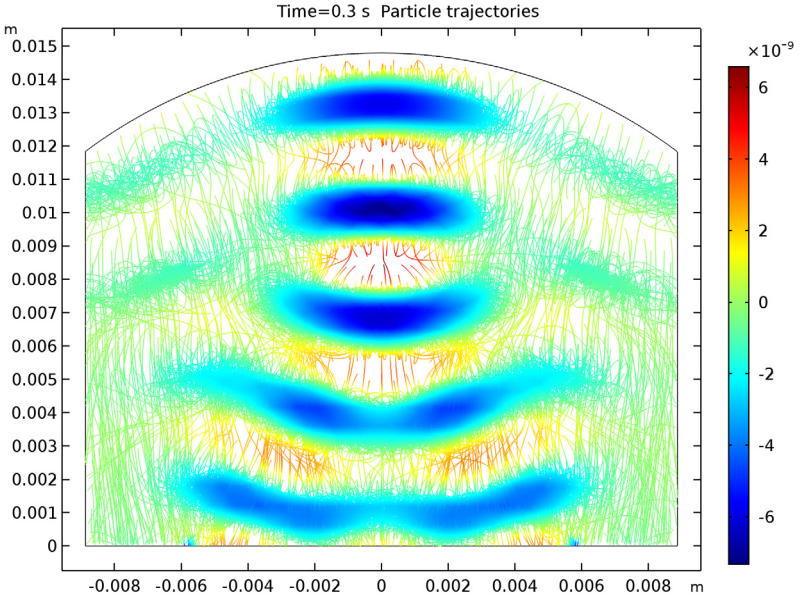


Figure 5: Particle position and trajectories plotted using the radiation potential as coloring.

References

1. A. Brask, T. Ullum, P. Thybo, and M. Wahlberg, "High-Temperature Ultrasonic Lavitator for Investigating Drying Kinetics of Single Droplets," *6th Int. Conf. on Multiphase Flow, ICMF 2007, (Leipzig, July 9-13)*, paper 789, 2007.

2. E.G. Lierke and L. Holitzner, "Perspectives of an Acoustic-Electrostatic/ Electrodynamic Hybrid Levitator for Small Fluid and Solid Samples," *Meas. Sci. Technol.*, vol. 19, p. 115803, 2008.

Application Library path: `Acoustics_Module/Nonlinear_Acoustics/acoustic_levitator`

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Fluid Flow>Particle Tracing> Particle Tracing for Fluid Flow (fpt)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Frequency Domain**.

8 Click **Done**.

STUDY 1

In this model, you first solve for the acoustic field in the frequency domain. Therefore, deselect the Particle Tracing for Fluid Flow physics.

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 **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Particle Tracing for Fluid Flow** interface.

As a second step, you solve for the particle movement in the time domain.

ADD STUDY

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

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Pressure Acoustics, Frequency Domain (acpr)** interface.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Time Dependent**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `acoustic_levitator_parameters.txt`.

The parameters loaded here define the geometrical dimensions, the driving frequency, typical wavelength, and particle properties. Because the geometry is now parametrized, changing the dimensions in the parameters list will update the geometry automatically.

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

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

4 In the **Sector angle** text field, type `180`.

5 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `Dr`.

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

5 Locate the **Position** section. In the **x** text field, type `-Dr/2`.

6 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Intersection 1 (int1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 Right-click **Intersection 1 (int1)** and choose **Build Selected**.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **x** text field, type `-Dt/2`.

4 Right-click **Point 1 (pt1)** and choose **Build Selected**.

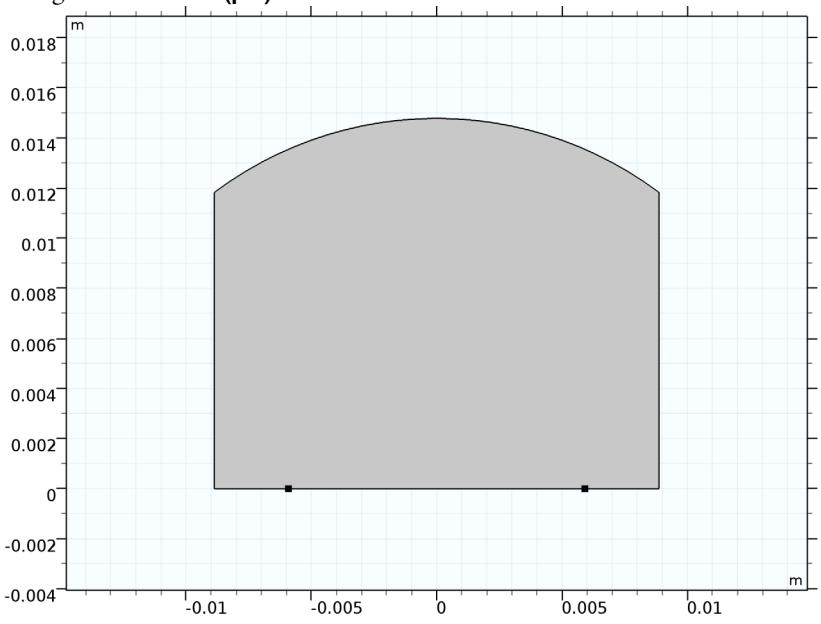
Point 2 (pt2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

3 In the **x** text field, type `Dt/2`.

4 Right-click **Point 2 (pt2)** and choose **Build Selected**.



ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Normal Acceleration 1

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

2 Select Boundaries 3 and 4 only.

3 In the **Settings** window for **Normal Acceleration**, locate the **Normal Acceleration** section.

4 In the a_n text field, type `a0`.

Plane Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundaries 1, 2, 5, and 6 only.

PARTICLE TRACING FOR FLUID FLOW (FPT)

Particle Properties 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)> Particle Tracing for Fluid Flow (fpt)** node, then click **Particle Properties 1**.

2 In the **Settings** window for **Particle Properties**, locate the **Particle Properties** section.

3 In the ρ_p text field, type `rho_p`.

4 In the d_p text field, type `d_p`.

5 In the **Model Builder** window, click **Particle Tracing for Fluid Flow (fpt)**.

Wall 2

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

2 Select Boundaries 1, 2, 5, and 6 only.

3 In the **Settings** window for **Wall**, locate the **Wall Condition** section.

4 From the **Wall condition** list, choose **Disappear**.

Particles are set to freeze on the solid walls but they disappear where the system is open (at the radiation boundaries).

Release 1

1 On the **Physics** toolbar, click **Domains** and choose **Release**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

Acoustophoretic Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Acoustophoretic Force**.

2 In the **Settings** window for **Acoustophoretic Force**, locate the **Acoustic Properties** section.

3 From the *p* list, choose **Pressure (acpr)**.

4 Locate the **Advanced Settings** section. Select the **Use piecewise polynomial recovery on field** check box.

5 Click in the **Graphics** window and then press Ctrl+A to select all domains.

Gravity Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Gravity Force**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

Drag Force 1

1 On the **Physics** toolbar, click **Domains** and choose **Drag Force**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

MESH 1

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

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

3 From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `lambda0/8`.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.

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

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

3 In the **Frequencies** text field, type `f0`.

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

RESULTS

Acoustic Pressure (acpr)

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

This first figure should look like the one in Figure 2.

Sound Pressure Level (acpr)

The sound pressure level plot should look like Figure 3.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,0.0005,0.3)`.

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1, Frequency Domain**.

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

RESULTS

Particle Trajectories (fpt)

1 In the **Model Builder** window, under **Results** click **Particle Trajectories (fpt)**.

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

3 From the **Time (s)** list, choose **0.01**.

4 On the **Particle Trajectories (fpt)** toolbar, click **Plot**.

The particle position at t = 0.01s is reproduced in Figure 4 (top).

5 From the **Time (s)** list, choose **0.3**.

6 On the **Particle Trajectories (fpt)** toolbar, click **Plot**.

The particle position at t = 0.3s is reproduced in Figure 4 (bottom).

2D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 From the **Data set** list, choose **Particle 1**.

4 From the **Time (s)** list, choose **0.3**.

Color Expression 1

1 Right-click **2D Plot Group 4** and choose **Particle Trajectories**.

2 In the **Model Builder** window, under **Results>2D Plot Group 4** right-click **Particle Trajectories 1** and choose **Color Expression**.

3 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Particle Tracing for Fluid Flow>Particle properties>fpt.acof1.Urad - Radiation potential**.

4 On the **2D Plot Group 4** toolbar, click **Plot**.

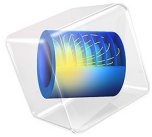
This figure is depicted in Figure 5. It plots the radiation potential as a color expression on the particle trajectories. Note that the particles move towards the low potential regions, as expected.

2D Plot Group 4

1 In the **Model Builder** window, under **Results** click **2D Plot Group 4**.

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

As an optional extension of the model, you can go on to the **Export** node and animate the particle plot.



Acoustic Scattering off an Ellipsoid

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example studies the scattering of an incident plane acoustic wave off a rigid ellipsoid geometry. The model utilizes the scattered field formulation, which enables the separation of the incident (background) pressure field and the scattered field. Using the far-field calculation feature, the scattered field is determined at a given distance outside the computational domain. The results are presented as 3D cross-section plots and as polar plots of the scattered far-field pressure and sound pressure level.

Model Definition

Figure 1 shows a sketch of the modeled system. A rigid ellipsoid is hit by an incident plane wave (here named the background pressure field p_b). The scattered field off the ellipsoid is denoted p.

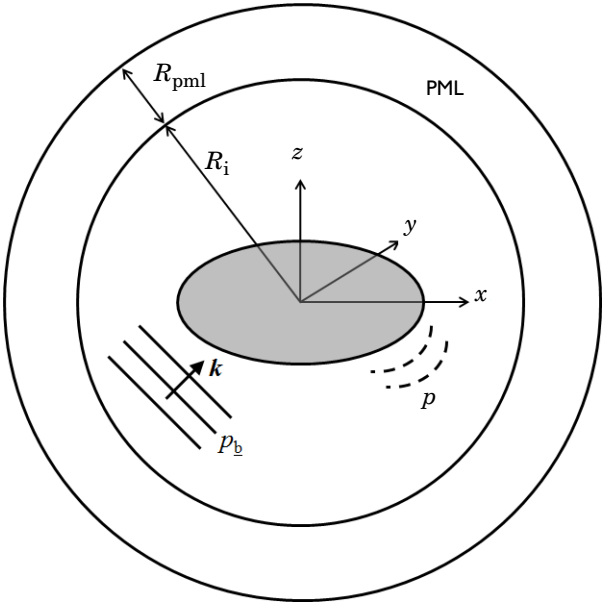


Figure 1: Sketch of the modeled system showing the incident background pressure field, p_b, the scattered field, p, geometric scales, and the PML layer.

The total acoustic field p_t is given by the sum of the scattered and the background pressure field such that

$$p_t = p_b + p$$
$$p_b = p_0 e^{-i(\mathbf{k}\cdot\mathbf{x})} \quad (1)$$

The background pressure field is a plane wave of amplitude p_0 moving in the direction $\mathbf{k}$ with wave number $|\mathbf{k}| = 2\pi f_0/c_0$, where f_0 is the frequency and c_0 is the speed of sound. The governing equations are implemented as a scattered field formulation such that only the scattered field p is solved for. See the *Acoustics Module User's Guide* for information about the background pressure field feature.

The ellipsoid is located inside a computational domain of total radius $R_i + R_{pml}$, where the layer of thickness R_{pml} is the absorbing *perfectly matched layer* (PML); see Figure 1. The PML is used as a non-reflecting and absorbing boundary that mimics a domain stretching to infinity. For more information about PMLs, see the *COMSOL Multiphysics Reference Manual* for information about infinite elements.

The surrounding fluid in this model is water. Approximate physical quantities (for water at 20°C) and dimensions, used in the model, are given in the table below.

TABLE 1: PHYSICAL QUANTITIES AND DIMENSIONS

SYMBOL	VALUE	DESCRIPTION
R_i	1 m	Radius of inner modeled water region
R_{pml}	0.5 m	PML layer thickness
R_{far}	10 m	Distance at which the far-field is evaluated
A	0.5 m	x-semiaxis of ellipsoid
B	0.25 m	y-semiaxis of ellipsoid
C	0.25 m	z-semiaxis of ellipsoid
f_0	1000 Hz	Driving frequency
c_0	1500 m/s	Speed of sound in water
λ_0	1.5 m	Wavelength in water at f0

MESH

When modeling a wave problem the computational mesh has to provide sufficient resolution of the waves. In 3D acoustic models it is necessary to have a minimum of 6 elements per wavelength when using second-order elements (this is the default element for pressure acoustics). This sets a limit to the maximum element size for the mesh of

$$L_{max} = \frac{\lambda_0}{6} = \frac{1}{6}\frac{c_0}{f_0}$$

which for this model corresponds to 0.25 m.

Inside the perfectly matched layer (PML) it is also important to take care of the mesh. In order for the PML to dampen the outgoing waves optimally, it is advised to use a swept mesh containing 6 elements. Figure 2 shows the mesh used in this model.

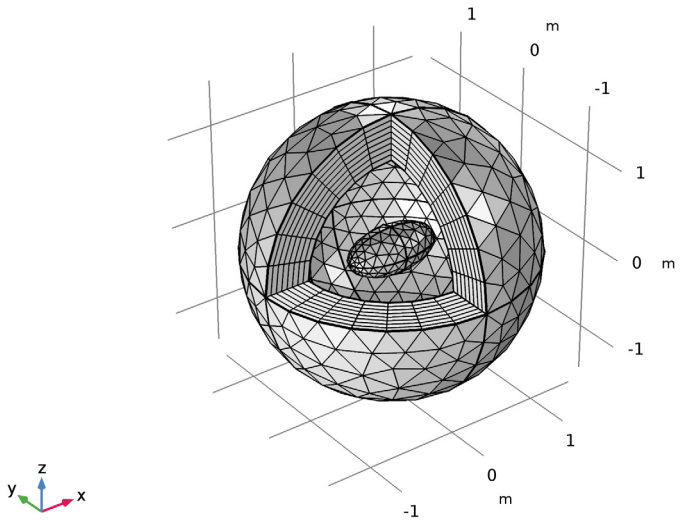


Figure 2: Mesh plot depicting the extruded mesh in the PML region and the mesh on the ellipsoid surface.

FAR FIELD

After solving a pressure acoustics model, it is possible to determine the pressure outside the computational domain using the far-field calculation feature. The far-field feature solves the Helmholtz-Kirchhoff (H-K) integral on the selected boundaries. The selected boundaries need to form a closed surface around all sources and scatterers. If the model has symmetries these can be included. Note that two versions of the H-K integral exist, one that only determines the pressure at the infinity limit (an approximation to the H-K integral is then used) and one version that solves the full H-K integral. In this model we

use the full integral and can thus determine the exact far-field pressure (including phase) at any point and distance outside the computational domain.

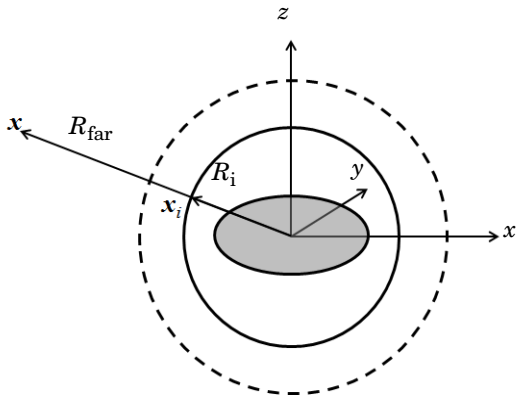


Figure 3: Relation between a coordinate defined in the far field boundary $\mathbf{x}_i$ *and the coordinate* $\mathbf{x}$ *at a distance* R_{far}.

For plotting purposes, the far-field pressure variable `pfar` is defined by COMSOL. This variable defines the pressure at any coordinates x, y, and z that are outside the boundary in which the far-field calculation is defined (for $|x| > R_i$ in Figure 3). The far-field pressure (variable name `acpr.ffc1.pfar`) and far-field sound pressure level (variable name `acpr.ffc1.Lp_pfar`) are easily plotted and visualized using the far-field plot types. They exist in 1D plot groups for plotting on, for example, a polar plot group, in 2D plot groups, and in 3D plot groups for creating 3D polar plots.

Finally, note that in order to get a precise evaluation of the far-field variable the evaluation of the H-K integral must be accurate. This requires having a good numerical estimate of the normal derivative of the pressure on the far-field calculation surface (adjacent to the PML layer). The easiest way to achieve this is by adding a single boundary layer mesh element on the inside of the acoustic domain. The thickness of this layer should be one tenth of the element size in the domain such that

$$L_{\text{boundary layer}} = \frac{L_{\max}}{10} = \frac{\lambda_0}{60} = \frac{1}{60}\frac{c_0}{f_0}$$

Figure 4 shows the total acoustic field p_t. It is the sum of the scattered field p and the incident background pressure field p_b, shown in Figure 5.

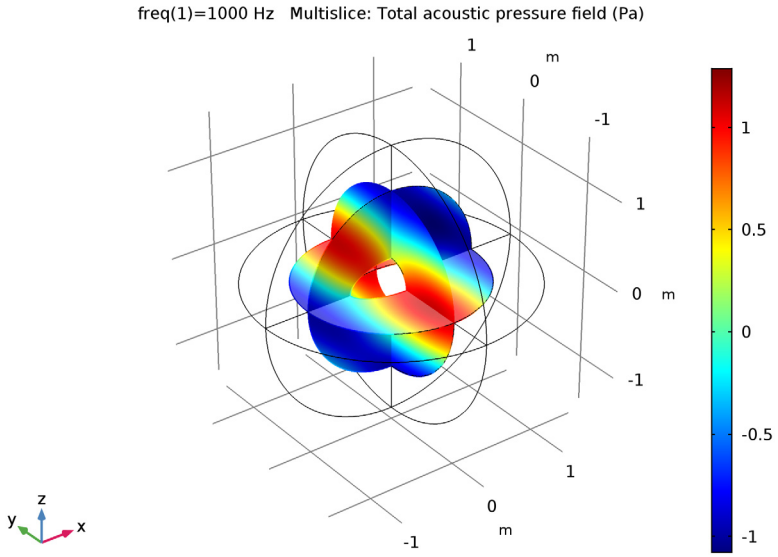


Figure 4: Total acoustic field at f = 1000 Hz.

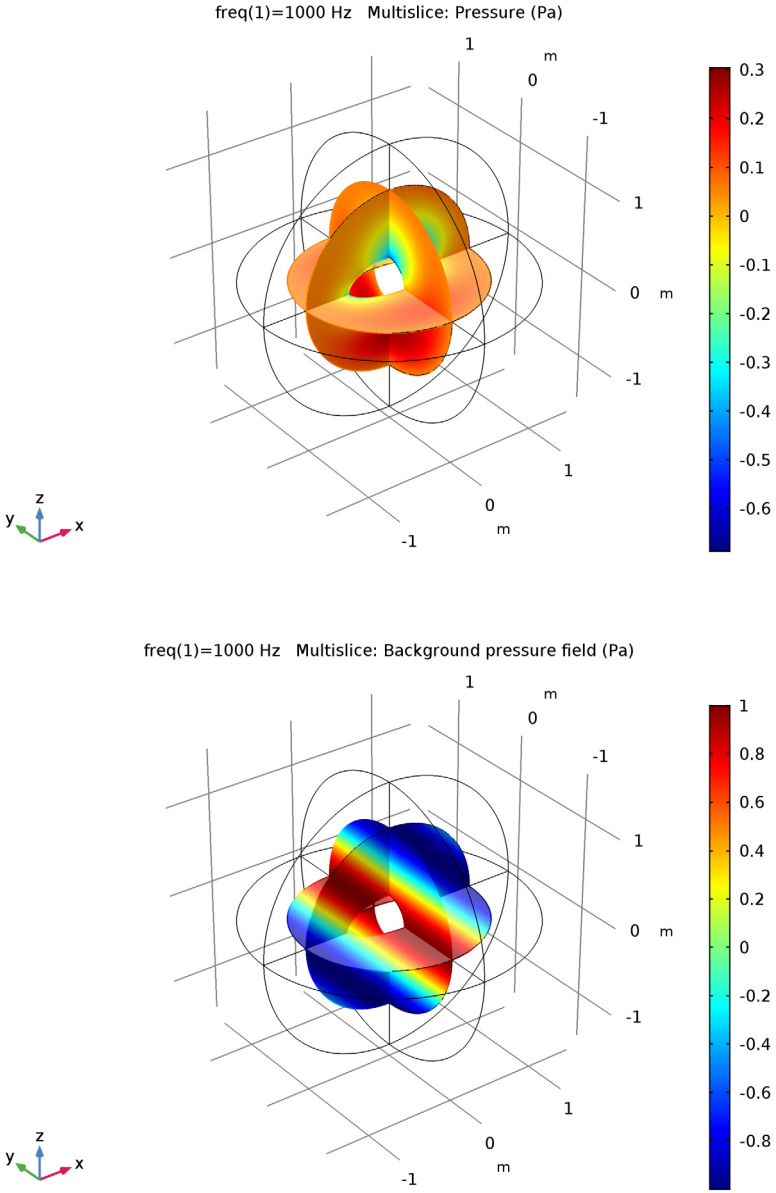


Figure 5: Scattered acoustic field (top) and incident plane-wave acoustic field (bottom).

Figure 6 plots the pressure in the far field at the distance $R_{\text{far}} = 10$ m. The data is retrieved in the *xy*-plane and presented as a polar plot, with 0° corresponding to the positive *x* direction. The sound pressure level in the far field is likewise represented in a polar plot in Figure 7. It is easy to determine the pressure and sound pressure level at another distance; simply change the parameter value for `Rfar` under the parameters and update the solution (press F5). The plots are then updated accordingly.

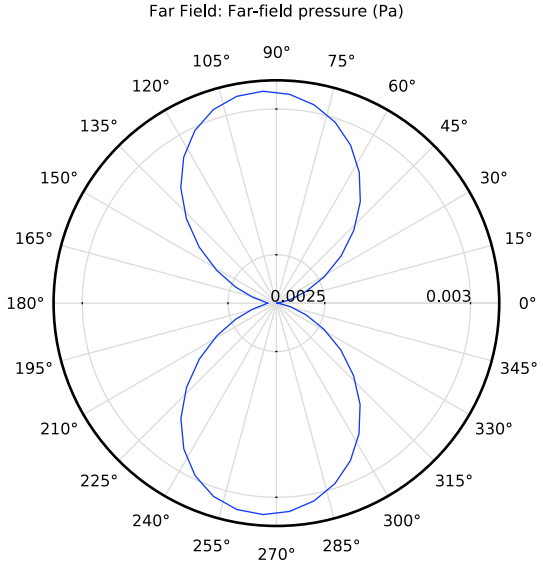


Figure 6: Polar plot of the pressure p at distance $R_{\text{far}} = 10$ m from the origin. The plot represents data in the xy-plane.

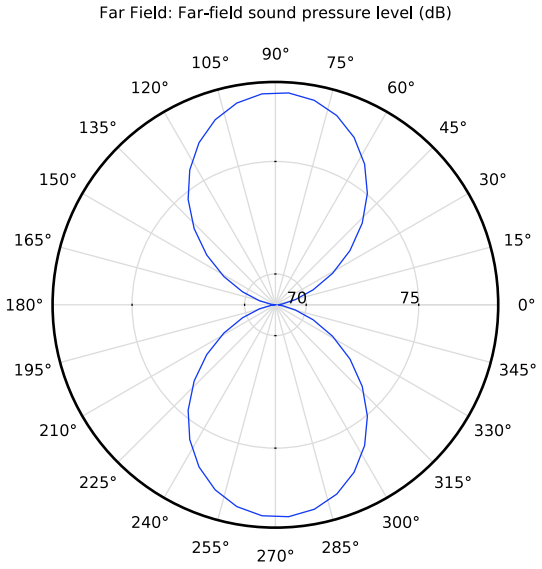


Figure 7: Polar plot of the far-field sound pressure level in the xy-plane.

The spatial response is visualized as a 3D far-field plot in Figure 8. The plot represents the sound pressure level. The radial dB scale zero-point has been moved to 66 dB in order to enhance the visualization of the notches in the far-field pattern. The surface color scale is the actual sound pressure level.

Finally, in Figure 9 the pressure is plotted outside of the computational mesh using the Grid 3D data set. Here the scattered pressure by plotting `pfar(x,y,z)`, alternatively plot the sound pressure level by plotting `acpr.ffc1.Lp_pfar`.

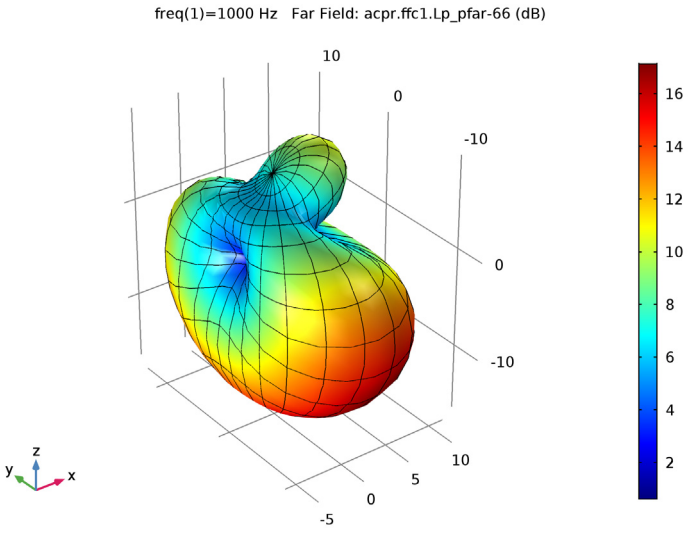


Figure 8: 3D far-field plot of the sound pressure level, the surface color scale is the actual sound pressure level.

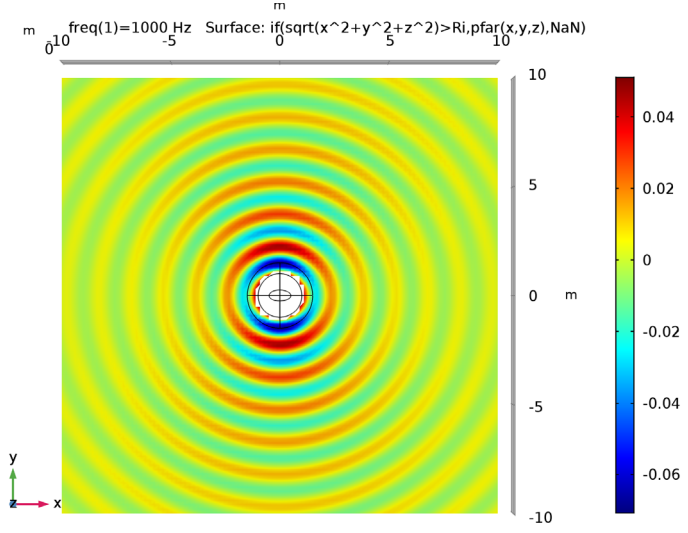


Figure 9: The scattered field depicted outside the computational mesh using the grid data set and plotting the far-field variable pfar.

Application Library path: `Acoustics_Module/Tutorials/acoustic_scattering`

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6. Click **Done**.

GLOBAL DEFINITIONS

Load the parameters defining the physical values and the geometric dimensions of the system from file (see Table 1).

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `acoustic_scattering_parameters.txt`.

GEOMETRY 1

Ellipsoid 1 (elp1)

1. On the **Geometry** toolbar, click **More Primitives** and choose **Ellipsoid**.
2. In the **Settings** window for **Ellipsoid**, locate the **Size and Shape** section.
3. In the **a-semiaxis** text field, type A.

4 In the **b-semiaxis** text field, type `B`.

5 In the **c-semiaxis** text field, type `C`.

6 Right-click **Ellipsoid 1 (elp1)** and choose **Build Selected**.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

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

3 In the **Radius** text field, type `Ri+Rpml`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Rpml

5 Right-click **Sphere 1 (sph1)** and choose **Build Selected**.

6 Click the **Zoom Extents** button on the **Graphics** toolbarto see the full geometry and then select wireframe rendering for easier visualization of the internal geometry. This makes selecting internal domains and boundaries much easier.

7 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **sph1** only.

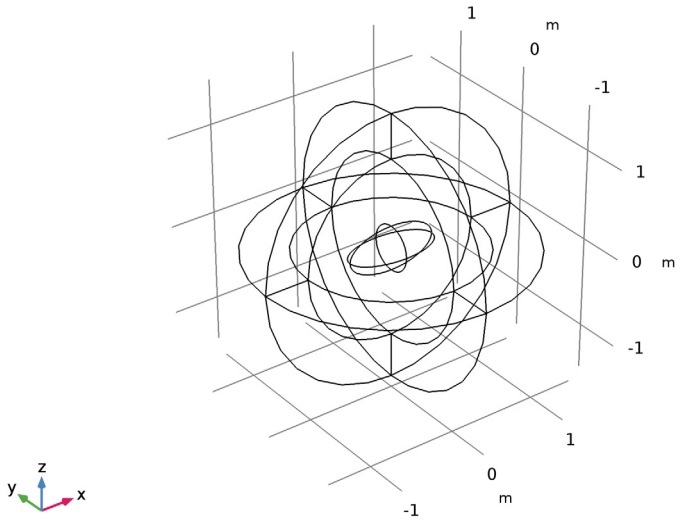
3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **elp1** only.

6 Click **Build All Objects**.

The geometry should look like that in the figure below.



DEFINITIONS

Now create selections for use when setting up the model. First create a selection for the perfectly matched layer. This is the outermost layer of the sphere, which consist of 8 domains.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 Select Domains 1–4 and 6–9 only.

3 In the **Settings** window for **Explicit**, type PML in the **Label** text field.

Add a selection for the boundaries on which the far-field is calculated. These boundaries must surround all scatterers, in this case the ellipsoid.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundaries 9–12, 22, 23, 29, and 34 only.

5 In the **Label** text field, type `Far-Field`.

ADD MATERIAL

1 On 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>Water, liquid**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Set the model reference pressure to 1 Pa, the default for water, and set the reference speed of sound to that in water. The latter option is used to determine the scaling used in the perfectly matched layer (PML).

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Sound Pressure Level Settings** section.

2 From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

3 Locate the **Typical Wave Speed for Perfectly Matched Layers** section. In the c_{ref} text field, type `1480[m/s]`.

The incident pressure field is defined as a Background Pressure Field domain contribution. In this model the incident wave has an amplitude of $p_0 = 1$ Pa and is traveling in the direction $e_k = (1,0,1)$.

Background Pressure Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Background Pressure Field**.

2 Select Domain 5 only.

3 In the **Settings** window for **Background Pressure Field**, locate the **Background Pressure Field** section.

4 In the p_0 text field, type `1`.

5 From the c list, choose **From material**.

6 From the **Material** list, choose **Water, liquid (mat1)**.

7 Specify the $\mathbf{e}_k$ vector as

1	x

0	y
1	z

Now set up the far-field calculation.

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 In the **Settings** window for **Far-Field Calculation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Far-Field**.

4 Locate the **Far-Field Calculation** section. From the **Type of integral** list, choose **Full integral**.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

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

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

MESH 1

Set the global maximum element size as six elements per wavelength. Add a thin boundary layer, of thickness `lambda0/6/10`, adjacent to the PML domain. Turn off the smooth transition option - the single layer is used to get a good normal gradient evaluation.

Boundary Layers 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Boundary Layers**.

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

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

4 Select Domain 5 only.

5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.

3 In the **Number of boundary layers** text field, type `1`.

4 Locate the **Boundary Selection** section. From the **Selection** list, choose **Far-Field**.

5 Locate the **Boundary Layer Properties** section. From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `lambda0/6/10`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `lambda0/6`.

5 Click **Build All**.

Finally, use swept mesh with eight elements for the PML.

Distribution 1

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

2 Right-click **Swept 1** and choose **Distribution**.

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

4 In the **Number of elements** text field, type `8`.

5 In the **Model Builder** window, click **Mesh 1**.

6 In the **Settings** window for **Mesh**, click **Build All**.

Hide some domains and boundaries to get a better view of the interior parts when reviewing the mesh.

7 Click the **View Unhidden** button on the **Graphics** toolbar.

8 Click the **Click and Hide** button on the **Graphics** toolbar.

9 Click the **Select Boundaries** button on the **Graphics** toolbar.

10 Select Boundary 6 only.

11 Select Boundary 10 only.

The mesh should look like the one depicted in Figure 2.

Remember to reset the hiding in order to see the full model when processing the results.

12 Click the **Reset Hiding** button on the **Graphics** toolbar.

STUDY 1

1. In the **Settings** window for **Study**, locate the **Study Settings** section.
2. Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1. In the **Model Builder** window, expand the **Study 1** node, then 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 `f0`.
4. On the **Home** toolbar, click **Compute**.

RESULTS

Plot the total acoustic field (Equation 1).

3D Plot Group 1

1. On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Total Field` in the **Label** text field.

Selection 1

1. On the **Total Field** toolbar, click **More Plots** and choose **Multislice**.
2. In the **Model Builder** window, right-click **Multislice 1** and choose **Selection**.
3. In the **Settings** window for **Selection**, locate the **Selection** section.
4. Click **Paste Selection**.
5. In the **Paste Selection** dialog box, type `5` in the **Selection** text field.
6. Click **OK**.
7. On the **Total Field** toolbar, click **Plot**.

Total Field

The resulting plot should look like Figure 4.

Total Field 1

1. In the **Model Builder** window, under **Results** right-click **Total Field** and choose **Duplicate**.
2. In the **Settings** window for **3D Plot Group**, type `Scattered Field` in the **Label** text field.

Scattered Field

To reproduce the plot of the scattered acoustic field shown in Figure 5 (top), proceed as follows:

Multislice 1

1 In the **Model Builder** window, expand the **Results>Scattered Field** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `p`.

4 On the **Scattered Field** toolbar, click **Plot**.

Scattered Field 1

1 In the **Model Builder** window, under **Results** right-click **Scattered Field** and choose **Duplicate**.

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

Multislice 1

1 In the **Model Builder** window, expand the **Results>Background Field** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_b`.

4 On the **Background Field** toolbar, click **Plot**.

Background Field

This plot represents the background or incident acoustic field, and should look like the plot in Figure 5 (bottom).

Background Field 1

1 In the **Model Builder** window, under **Results** right-click **Background Field** and choose **Duplicate**.

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

Sound Pressure Level

Use the full data set to visualize the sound pressure level (SPL) of the scattered acoustic field in the PML.

Multislice 1

In the **Model Builder** window, expand the **Results>Sound Pressure Level** node.

Selection 1

1 In the **Model Builder** window, expand the **Multislice 1** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **All domains**.

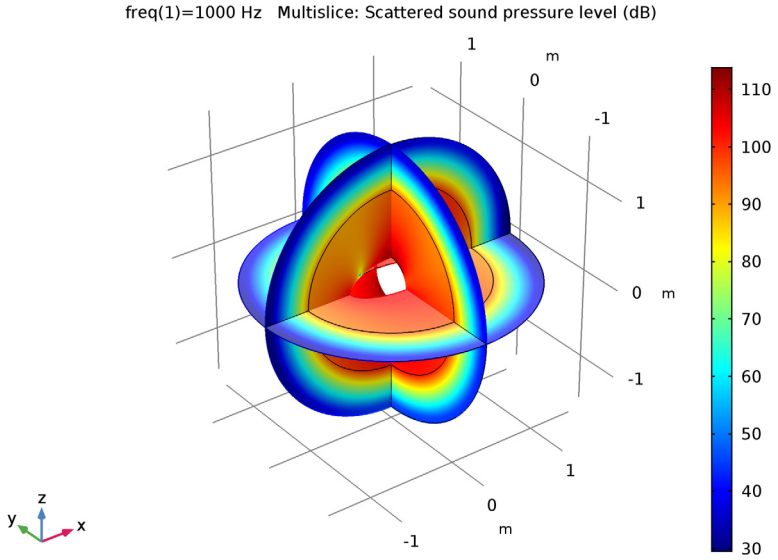
Multislice 1

1 In the **Model Builder** window, under **Results>Sound Pressure Level** click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.Lp_s`.

4 On the **Sound Pressure Level** toolbar, click **Plot**.



Observe that the scattered field is highly damped in the PML.

Continue with visualization of the scattered far-field pressure and sound pressure level at the distance R_{far} = 10 m. Use the dedicated far-field plot types.

Reproduce the scattered pressure field (Figure 6) and the sound pressure level of the scattered pressure field (Figure 7) at a distance of 10 m from the center of the ellipsoid in the xy-plane as follows:

Polar Plot Group 5

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

2 In the **Settings** window for **Polar Plot Group**, type `Far-Field Pressure xy-Plane` in the **Label** text field.

Far Field 1

1 On the **Far-Field Pressure xy-Plane** toolbar, click **More Plots** and choose **Far Field**.

2 In the **Settings** window for **Far Field**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.ffc1.pfar`.

Per default the real part of a variable is plotted. If you need the imaginary part write `imag(acpr.ffc1.pfar)` or the absolute value write `abs(acpr.ffc1.pfar)`.

4 Locate the **Evaluation** section. Find the **Evaluation distance** subsection. In the **Radius** text field, type `Rfar`.

5 On the **Far-Field Pressure xy-Plane** toolbar, click **Plot**.

Far-Field Pressure xy-Plane 1

1 In the **Model Builder** window, under **Results** right-click **Far-Field Pressure xy-Plane** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Far-Field SPL xy-Plane` in the **Label** text field.

Far Field 1

1 In the **Model Builder** window, expand the **Results>Far-Field SPL xy-Plane** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.ffc1.Lp_pfar`.

4 On the **Far-Field SPL xy-Plane** toolbar, click **Plot**.

Now, plot the scattered pressure field and the sound pressure level of the scattered pressure field at a distance of 10 m from the ellipsoid in the yz-plane. Note that the yz-plane has the normal in the x-direction.

Far-Field Pressure xy-Plane 1

1 In the **Model Builder** window, under **Results** right-click **Far-Field Pressure xy-Plane** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Far-Field Pressure yz-Plane` in the **Label** text field.

Far Field 1

Set the reference direction. It defines the direction in space that corresponds to 0 deg. in the polar plot.

1 In the **Model Builder** window, expand the **Results>Far-Field Pressure yz-Plane** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Reference direction** subsection. In the **x** text field, type `0`.

4 In the **y** text field, type `1`.

5 In the **z** text field, type `0`.

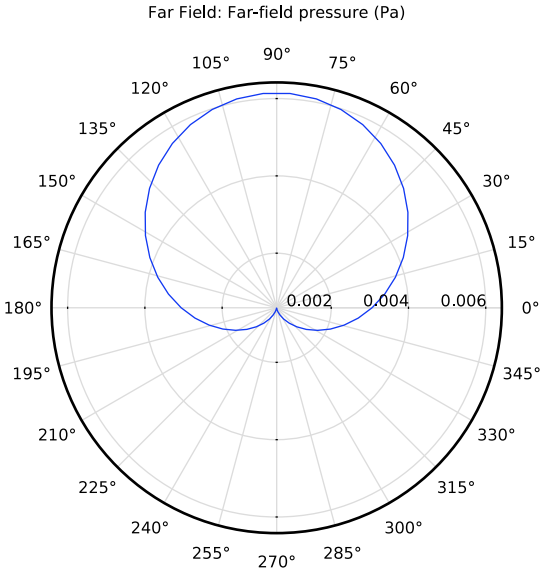
Set the normal to get the yz-plane.

6 Find the **Normal** subsection. In the **x** text field, type `1`.

7 In the **y** text field, type `0`.

8 In the **z** text field, type `0`.

9 On the **Far-Field Pressure yz-Plane** toolbar, click **Plot**.



Far-Field SPL xy-Plane 1

1 In the **Model Builder** window, under **Results** right-click **Far-Field SPL xy-Plane** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Far-Field SPL yz-Plane` in the **Label** text field.

Far Field 1

1 In the **Model Builder** window, expand the **Results>Far-Field SPL yz-Plane** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Reference direction** subsection. In the **x** text field, type `0`.

4 In the **y** text field, type `1`.

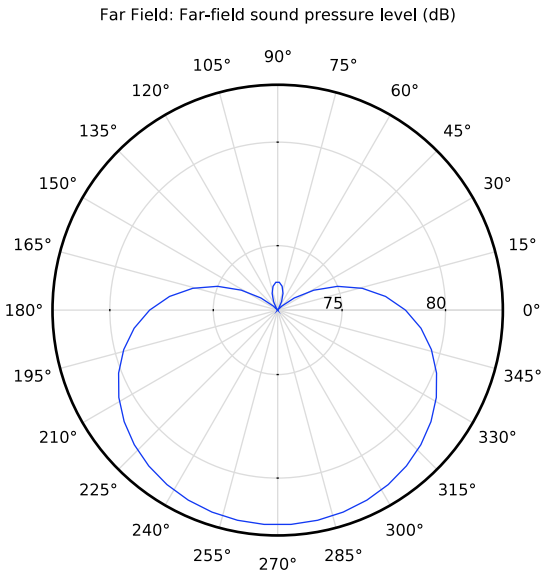
5 In the **z** text field, type `0`.

6 Find the **Normal** subsection. In the **x** text field, type `1`.

7 In the **y** text field, type `0`.

8 In the **z** text field, type `0`.

9 On the **Far-Field SPL yz-Plane** toolbar, click **Plot**.



Finally, plot the far-field sound pressure level as a 3D polar plot.

3D Plot Group 9

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `3D Polar Far-Field SPL` in the **Label** text field.

Create a new view for this figure. This will keep the view and zoom settings on all the previous figures.

3 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

Far Field 1

1 On the **3D Polar Far-Field SPL** toolbar, click **More Plots** and choose **Far Field**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of elevation angles** text field, type `20`.

4 In the **Number of azimuth angles** text field, type `30`.

5 Find the **Sphere** subsection. From the **Sphere** list, choose **Manual**.

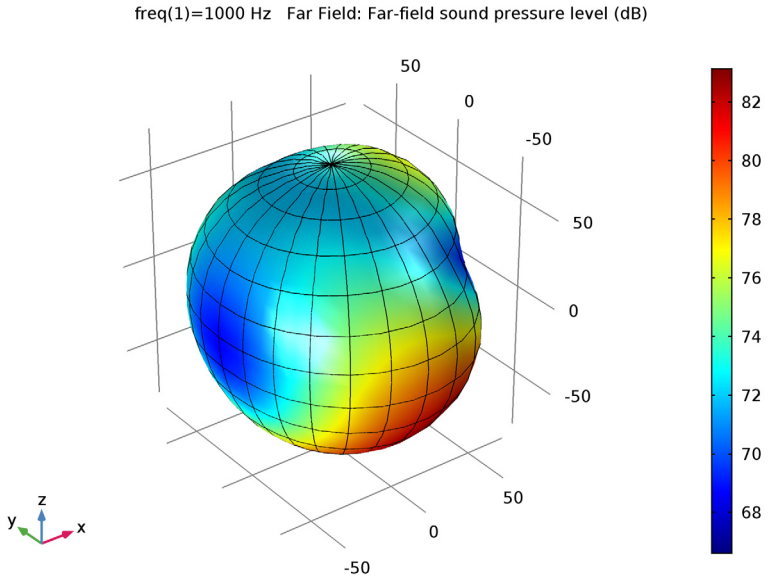
6 In the **Radius** text field, type `Rfar`.

7 Locate the **Coloring and Style** section. From the **Grid** list, choose **Fine**.

8 On the **3D Polar Far-Field SPL** toolbar, click **Plot**.

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

The figure should look like the one below. Click on the figure and rotate it to get a sense of the 3D spatial response of the sound pressure level of the scattered field.



In order to better visualize the spatial response, change the plot expression by subtracting 66 dB. This will move the dB scale zero point. The color scale on the surface still represents the sound pressure level. The plot should look like the one in Figure 8.

10 Locate the **Expression** section. In the **Expression** text field, type `acpr.ffc1.Lp_pfar-66`.

11 On the **3D Polar Far-Field SPL** toolbar, click **Plot**.

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

Data Sets

Finally, create and use a grid 3D data set to plot the pressure field in the xy-plane outside of the computational mesh. Reproducing Figure 9.

The settings for the grid data set can be modified to show other cross sections. The plot can also be modified to, for example, plot the sound pressure level `acpr.ffc1.Lp_pfar`.

Grid 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Grid>Grid 3D**.

2 In the **Settings** window for **Grid 3D**, locate the **Parameter Bounds** section.

3 Find the **First parameter** subsection. In the **Minimum** text field, type `-10`.

4 In the **Maximum** text field, type `10`.

5 Find the **Second parameter** subsection. In the **Minimum** text field, type `-10`.

6 In the **Maximum** text field, type `10`.

7 Find the **Third parameter** subsection. In the **Maximum** text field, type `0`.

8 Click to expand the **Resolution** section. In the **x resolution** text field, type `100`.

9 In the **y resolution** text field, type `100`.

10 In the **z resolution** text field, type `2`.

3D Plot Group 10

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

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

3 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 2**.

Surface 1

1 Right-click **Exterior Field** and choose **Surface**.

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

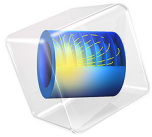
3 From the **Data set** list, choose **Grid 3D 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `if(sqrt(x^2+y^2+z^2)>Ri,pfar(x,y,z),NaN)`.

5 On the **Exterior Field** toolbar, click **Plot**.

6 Click the **Go to XY View** button on the **Graphics** toolbar.

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



Acoustic-Structure Interaction

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This model provides a tutorial on modeling the interaction of sound with elastic, solid structures by using a specific example of an elastic cylinder in water. The sound will cause movement of the solid cylinder, which in turn will induce new sound waves in the air; thus, full two-way coupling between the acoustic medium (water) and the cylinder is required to realistically simulate this situation.

This model is similar to the interaction of acoustic signals (sound) with most everyday objects: Liquid or gas acoustics coupled to structural objects have application in many engineering fields, for example:

- Loudspeakers
- Acoustic sensors
- Nondestructive impedance testing
- Medical ultrasound diagnostics of the human body

The intent with this tutorial model is to illustrate the modeling process rather than to provide an exhaustive illustration of the acoustic-solid interaction capabilities of COMSOL.

This model requires both the Acoustics Module and the Solid Mechanics Module.

Model Definition

This model simulates the behavior of a solid cylinder in an water domain with an incident acoustic wave in the water. The object's walls are impacted by the acoustic pressure. The model calculates the frequency response from the solid and then feeds this information back to the acoustics domain so that it can analyze the resulting wave pattern. As such, the model is a good example of a scattering acoustic-solid interaction problem.

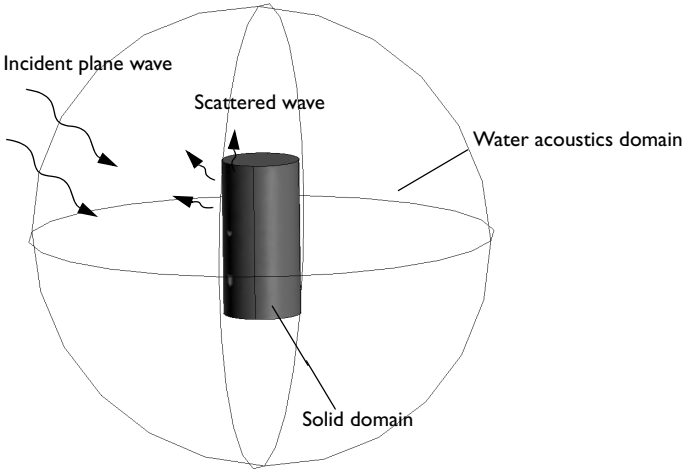


Figure 1: Geometric setup of an aluminum cylinder immersed in water.

To set up the model, use the Acoustic-Solid Interaction, Frequency Domain multiphysics coupling. This coupling involves two physics interfaces: Solid Mechanics and Pressure Acoustics, Frequency Domain. It also defines an Acoustic-Structure Boundary coupling feature.

Figure 1 illustrates the aluminum cylinder immersed in water. The incident wave is 60 kHz, in the ultrasound region. The cylinder is 2 cm in height and has a diameter of 1 cm. The water acoustic domain is truncated as a sphere with a reasonably large diameter. What drives the system is an incident plane wave from the surroundings into the spherical boundary. The harmonic acoustic pressure in the water on the surface of the cylinder acts as a boundary load in the 3D solid to ensure continuity in pressure. The model calculates harmonic displacements and stresses in the solid cylinder, and it then uses the normal acceleration of the solid surface in the acoustics domain boundary to ensure continuity in acceleration.

DOMAIN EQUATIONS

The default Pressure Acoustics feature models harmonic sound waves in the water domain by means of the Helmholtz equation for sound pressure:

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

Here the acoustic pressure is a harmonic quantity, $p = p_0 e^{i\omega t}$, and p is the pressure (SI unit: N/m^2), ρ_c is the density (kg/m^3), ω is the angular frequency (SI unit: rad/s), and c is the speed of sound (SI unit: m/s).

TABLE 1: ACOUSTICS DOMAIN DATA

QUANTITY	VALUE	DESCRIPTION
ρ_c	997 kg/m^3	Density
c	1500 m/s	Speed of sound
$f=\omega/2\pi$	60 kHz	Frequency

To calculate the harmonic stresses and strains in the solid cylinder for a frequency-response analysis, use the default Linear Elastic Material Model feature under the Solid Mechanics interface. The material data comes from the built-in database for Aluminum 3003-H18.

BOUNDARY CONDITIONS

Outer Perimeter

On the outer spherical perimeter of the water domain (Figure 1) specify an incident plane wave to represent an incoming sound wave. A superimposed spherical wave is allowed to travel out of the system as a response from the cylinder. In the Pressure Acoustics, Frequency Domain interface you implement this scenario by using the prepared Spherical Wave Radiation boundary condition. Such boundary condition is useful when the surroundings are only a continuation of the domain.

TABLE 2: RADIATION BOUNDARY CONDITION SETTINGS

QUANTITY	VALUE	DESCRIPTION
$\hat{k}$	$(\sin\theta\cos\varphi,\ \sin\theta\sin\varphi,\ \cos\theta)$	Incident wave direction vector
p_0	1 Pa	Pressure amplitude

The incident wave direction is controlled by the two angles $0 < \theta < \pi$ and $0 < \varphi < 2\pi$.

Interface Cylinder-Water

The coupling between the fluid domain (pressure waves) and the solid is automatically done via the Acoustic-Structure Boundary coupling feature. The automatic boundary condition sets the boundary load $\mathbf{F}$ (force/unit area) on the solid cylinder to

$$\mathbf{F} = -\mathbf{n}_s p$$

where $\mathbf{n}_s$ is the outward-pointing unit normal vector seen from inside the solid domain. While on the fluid side the normal acceleration experienced by the fluid is set equal to the normal acceleration of the solid. Mathematically this means that

$$-\mathbf{n}_a \cdot \left(-\frac{1}{\rho_0}\nabla p + \mathbf{q}\right) = a_n$$

where $\mathbf{n}_a$ is the outward-pointing unit normal vector seen from inside the acoustics domain, and the normal acceleration a_n is equal to $(\mathbf{n}_a \cdot \mathbf{u})\omega^2$, where $\mathbf{u}$ is the calculated harmonic-displacement vector of the solid structure.

HARD-WALL COMPARISON

As a reference you can also study a simpler model where the solid interface is regarded as a hard wall. This implies that the cylinder will not be affected by sound, but its presence will nonetheless affect how the sound is distributed. In the model this is achieved by setting a fixed constraint on all the solid boundaries, that is, $\mathbf{u} = \mathbf{0}$. This reduces the above condition ($a_n = 0$) to the sound hard boundary condition

$$\mathbf{n}_a \cdot \left(-\frac{1}{\rho_0}\nabla p + \mathbf{q}\right) = 0$$

Results and Discussion

Figure 2 displays the sound pressure as a slice plot. It is clear from which direction the sound wave propagates into the domain. The values of the deformation are very small, but the acceleration is large enough to have an impact on the sound waves.

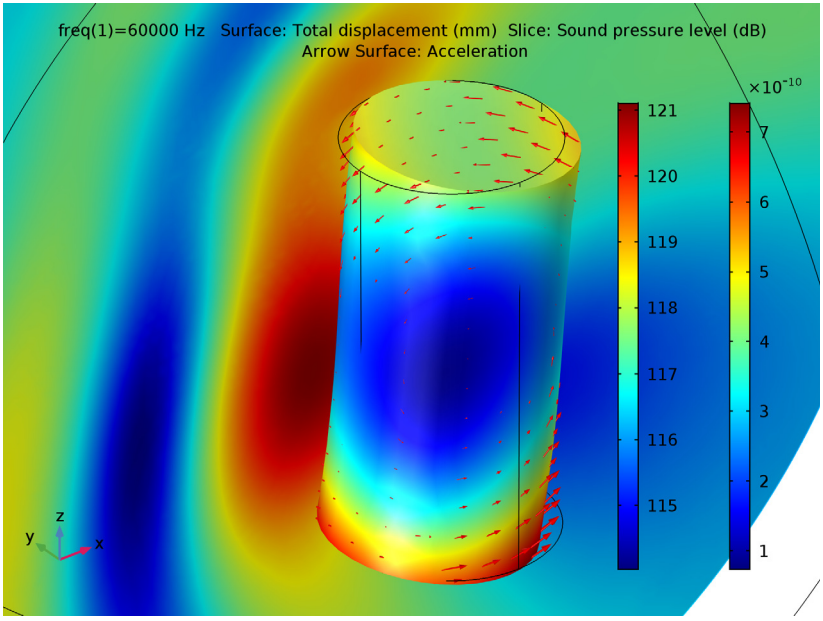


Figure 2: Displacement of the cylinder and the sound-pressure plot (dB) of the acoustic waves in the coupled problem. The arrow lengths are proportional to the surface acceleration, which is a direct measure of the sound-pressure interaction between the water and the cylinder.

Figure 3 shows a comparison between the hard-wall example and the full aluminum solid model. Near the cylinder wall the plot shows that the sound pressure level is higher on the upstream side for the hard-wall case than for the aluminum model. Conversely, the amplitude is lower for the hard-wall model than for the aluminum model on the downstream side. This shows that the hard wall reflects more and transmits less energy than the aluminum cylinder. The conclusion is that the mechanical properties of the metal object have an impact on the acoustic signature.

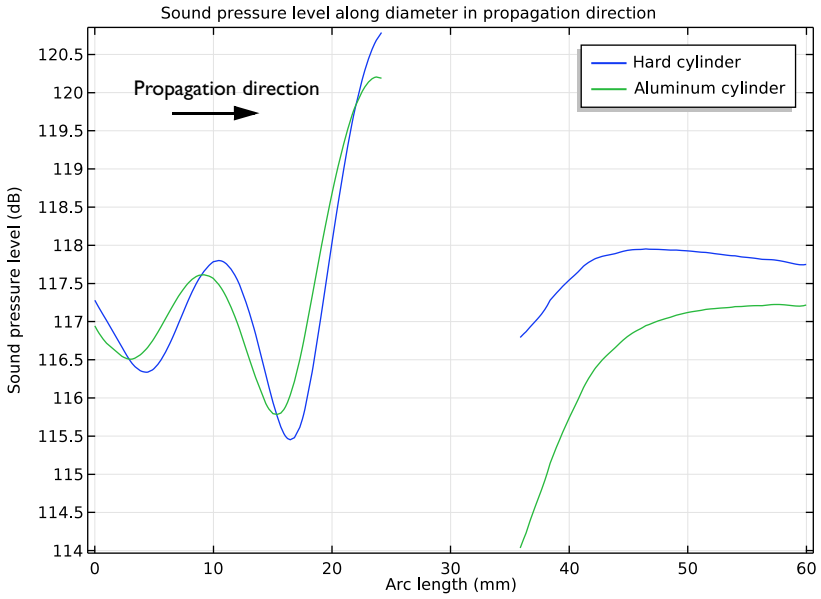


Figure 3: Sound pressure level on impact and on the shadow side of the cylinder.

Application Library path: `Acoustics_Module/Tutorials/acoustic_structure`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Solid Interaction, Frequency Domain**.
3. Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

You may either add the parameters manually or load them from a text file.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `acoustic_structure_parameters.txt`.

To add parameters manually, you can do as follows.

5 In the table, enter the following settings:

Name	Expression	Value	Description
f	60[kHz]	60000 Hz	Frequency
phi	(-pi/6)[rad]	-0.5236 rad	Wave direction angle, phi
theta	(4*pi/6)[rad]	2.0944 rad	Wave direction angle, theta
k1	sin(theta)*cos(phi)	0.75	Incident wave direction vector, X component
k2	sin(theta)*sin(phi)	-0.43301	Incident wave direction vector, Y component
k3	cos(theta)	-0.5	Incident wave direction vector, Z component
R	30[mm]	0.03 m	Model domain radius

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**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `5`.

4 In the **Height** text field, type `20`.

5 Locate the **Position** section. In the **z** text field, type `-10`.

6 Click **Build Selected**.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

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

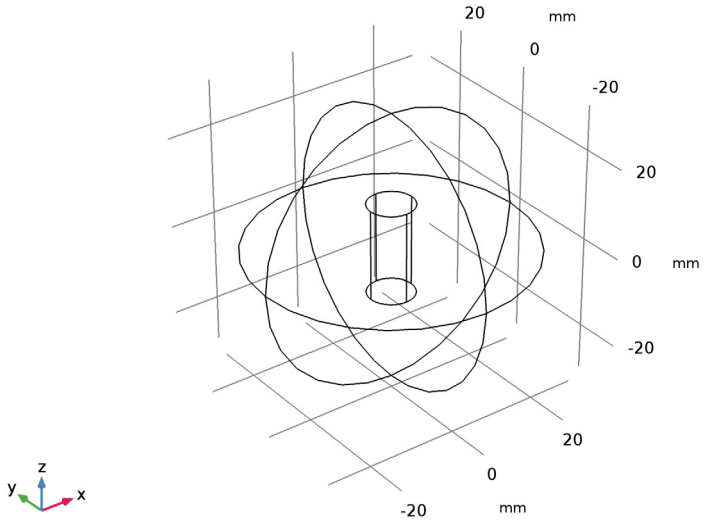
3 In the **Radius** text field, type `R`.

4 Click **Build Selected**.

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

To see the interior:

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



DEFINITIONS

Next, define a number of selections as sets of geometric entities to use when setting up the model.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Fluid Domain` in the **Label** text field.

3 Select Domain 1 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Solid Domain` in the **Label** text field.

3 Select Domain 2 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Radiation Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Select the **All domains** check box.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Solid Boundaries` in the **Label** text field.

3 Select Domain 2 only.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

ADD MATERIAL

1 On 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>Water, liquid**.

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

MATERIALS

Water, liquid (mat1)

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

2 From the **Selection** list, choose **Fluid Domain**.

ADD MATERIAL

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

2 In the tree, select **Built-In>Aluminum 3003-H18**.

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

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

MATERIALS

Aluminum 3003-H18 (mat2)

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

2 From the **Selection** list, choose **Solid Domain**.

Now, set up the physics of the problem by defining the domain physics conditions and the boundary conditions.

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 **Fluid Domain**.

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

Spherical Wave Radiation 1

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

2 In the **Settings** window for **Spherical Wave Radiation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Radiation Boundaries**.

Incident Pressure Field 1

1 On the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 In the **Settings** window for **Incident Pressure Field**, locate the **Incident Pressure Field** section.

3 In the p_0 text field, type 1.

4 From the c list, choose **From material**.

5 From the **Material** list, choose **Water, liquid (mat1)**.

6 Specify the $\mathbf{e}_k$ vector as

k1	x
k2	y
k3	z

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 **Solid Domain**.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

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

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `1500[m/s]/f/6`.

This setting is appropriate for the fluid domain and corresponds to 6 elements per wave length.

Add another Size node to specify a different maximum element size for the solid domain.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 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 From the **Selection** list, choose **Solid Domain**.

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

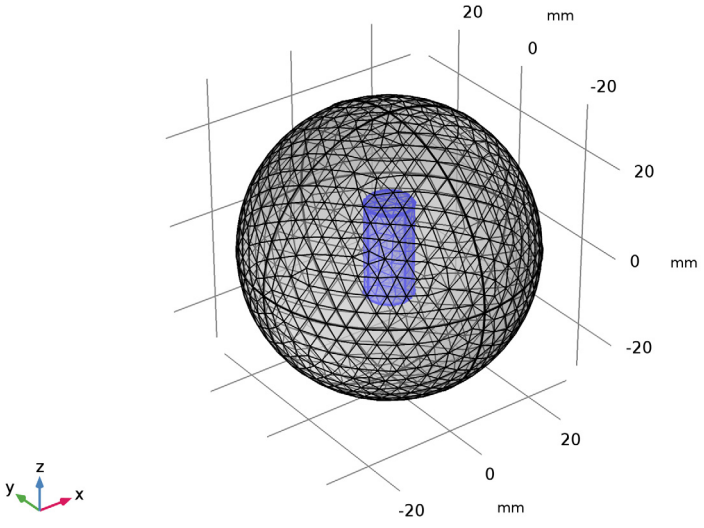
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `3`.

8 Click **Build All**.

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

10 Click the **Transparency** button on the **Graphics** toolbar.



To return to the default state:

11 Click the **Transparency** button on the **Graphics** toolbar.

STUDY 1

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

2 Clear the **Generate default plots** check box.

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 `f`.

Disable the Solid Mechanics interface, which corresponds to the hard cylinder case.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Solid Mechanics (solid)** interface.

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

Before visualizing this solution, include the structural analysis of the cylinder and compute the corresponding solution. You can do this by adding one more study.

ADD STUDY

1 On 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 **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, click **Study 2**.

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

3 Clear the **Generate default plots** check box.

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

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

6 In the **Frequencies** text field, type `f`.

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

RESULTS

To reproduce the plot in Figure 3, comparing the sound pressure levels along a diameter in the propagation direction for the two cases, begin by defining data sets as follows.

Cut Line 3D 1

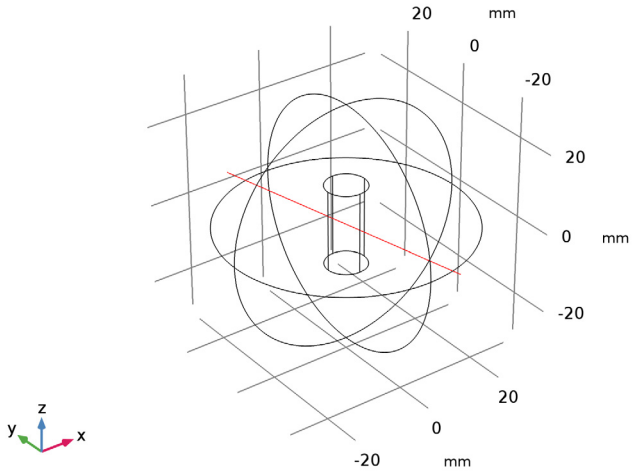
1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **X** to `-R*k1`, **Y** to `-R*k2`, and **Z** to `-R*k3`.

4 In row **Point 2**, set **X** to `R*k1`, **Y** to `R*k2`, and **Z** to `R*k3`.

5 Click **Plot**.



Cut Line 3D 2

1 Right-click **Cut Line 3D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Data** section.

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

1D Plot Group 1

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

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

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

4 In the **Title** text area, type `Sound pressure level along diameter in propagation direction`.

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

6 In the associated text field, type `Arc length (mm)`.

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

8 In the associated text field, type `Sound pressure level (dB)`.

Line Graph 1

1 Right-click **SPL Along Propagation** and choose **Line Graph**.

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

3 From the **Data set** list, choose **Cut Line 3D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain> Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Hard cylinder

Line Graph 2

1 Right-click **Results>SPL Along Propagation>Line Graph 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Cut Line 3D 2**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Aluminum cylinder

5 On the **SPL Along Propagation** toolbar, click **Plot**.

Finally, follow the instructions below to create the plot shown in Figure 2:

3D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

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

Surface 1

1 Right-click **Sound Pressure and 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>Solid Mechanics> Displacement>solid.disp - Total displacement**.

Slice 1

1 Right-click **Results>Sound Pressure and Displacement>Surface 1** and choose **Deformation**.

2 In the **Model Builder** window, under **Results** right-click **Sound Pressure and Displacement** and choose **Slice**.

3 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.

5 From the **Entry method** list, choose **Coordinates**.

6 In the **Y-coordinates** text field, type `5`.

Arrow Surface 1

1 Right-click **Sound Pressure and Displacement** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics> Acceleration and velocity>solid.u_ttX,...,solid.u_ttZ - Acceleration**.

3 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

4 Select the **Scale factor** check box.

5 In the associated text field, type `15`.

6 In the **Number of arrows** text field, type `5000`.

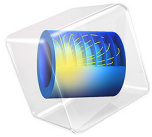
7 On the **Sound Pressure and Displacement** toolbar, click **Plot**.

Sound Pressure and Displacement

1 In the **Model Builder** window, under **Results** click **Sound Pressure and Displacement**.

2 Click **Plot**.

Click the **Zoom Box** button on the **Graphics** toolbar and then use the mouse to zoom in.



Acoustics of a Particulate-Filter-Like System

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of the acoustics in a particulate-filter-like system. Real systems, like diesel particulate filters (DPFs), are designed to remove/filter soot (diesel particles) from the exhaust of diesel engine vehicles. The porous medium in such systems are typically structured with long air-filled ducts. To simplify this model, the filter geometry is assumed to be axisymmetric and the ducts are represented by long cylindrical groves inside a porous material plug. Although the main function of a particulate filter is filtering of the exhaust flow, the filter also has acoustic damping properties that relate to the muffler system.

The model analyzes the acoustic properties of the simplified 2D axisymmetric particulate-filter like geometry using the Acoustic-Poroelastic Waves Interaction multiphysics interface. This interface describes the small-deformation elastic waves propagating in a porous material coupled to waves in a fluid by an Acoustic-Porous Boundary multiphysics coupling. The model accounts for the coupled displacement and is thus a fluid-structure interaction problem.

Model Definition

Three aligned cylinders make up the particulate filter system under study: an inlet, an outlet, and a main filter cylinder. The particulate filter is located inside the filter cylinder. Figure 1 shows a sketch of a cross section in the rz-plane of the 2D axisymmetric geometry. The filter in the central region is of length $L_{\text{filter}} = 200$ mm with a filter radius of $R_{\text{filter}} = 150$ mm. The inlet and outlet pipe radii are $R_{\text{tube}} = 50$ mm. The filter consists of a structured air-filled porous material (the brown region), which could be a silicon carbide matrix. The air-filled groves (light blue) have a width of $d_{\text{h}} = 5$ mm and the porous walls are of thickness $h_{\text{t}} = 3.2$ mm. At the end of each grove there is an impermeable steel plug (black). The rest of the system is filled with air.

The wide groves are used to simplify the model. In real DPF systems the groves are replaced by long slender ducts with a typical width of 1–2 mm and the porous walls have a typical width of 0.3–0.5 mm.

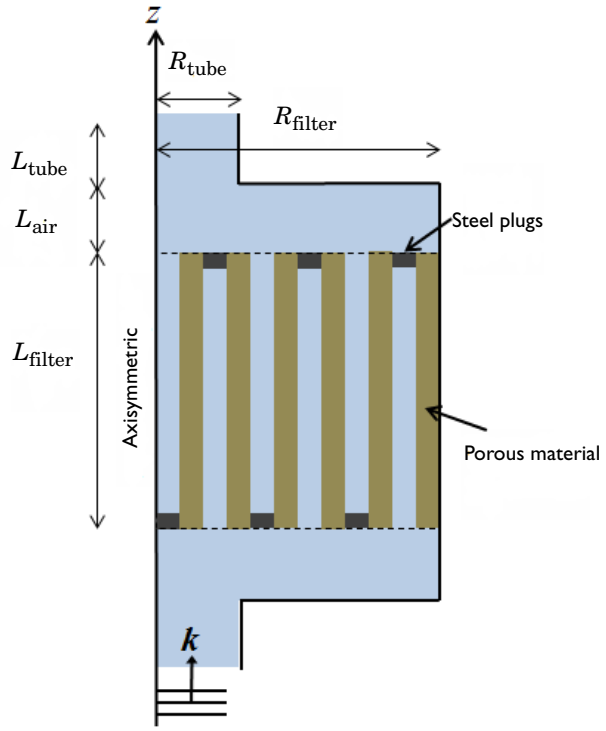


Figure 1: Geometry of the particulate filter with dimensions indicated.

The porous material is assumed to be isotropic with the material parameters as listed in Table 1.

TABLE 1: MATERIAL PARAMETERS OF THE POROUS MATRIX

PARAMETER	VALUE	DESCRIPTION
E	20 GPa	Young's modulus
ν	0.4	Poisson's ratio
ρ_d	1000 kg/m^3	Drained matrix density
α_B	0.3	Biot-Willis coefficient
ε_p	0.3	Porosity
κ_p	10^{-11} m^2	Permeability of porous matrix
τ	1	Tortuosity factor

Note that the Biot-Willis coefficient is equal to the porosity for rigid porous materials and is equal to 1 for a soft porous material (or a suspension of solid in liquid). The fluid parameters are those of air including the compressibility, χ, which for an ideal gas is equal to $(p_0)^{-1}$, where p_0 is the absolute pressure (here 1 atm).

The filter is characterized acoustically by the transmission loss, TL (given in dB), as a function of the frequency, f. It is defined as

$$\mathrm{TL}(f) = 20\log\left(\left|\frac{p_{\mathrm{incident}}}{p_{\mathrm{out}}}\right|\right)$$

where p_{incident} is the incident inlet pressure and p_{out} is the outlet pressure. You solve the model for the frequency interval 20 Hz–2000 Hz.

When setting up the porous material model, you also need to specify whether to use the low-frequency (default) or high-frequency range approximation for the fluid viscosity. The transition between the two ranges is defined by the reference frequency f_c given by the expression

$$f_c = \frac{\varepsilon_p \mu}{2\pi\kappa\rho_f}$$

where ρ_f is the fluid density (for air 1.2 kg/m^3) and μ is the dynamic viscosity of the fluid (for air $1.8\cdot10^{-5}$ Pa·s). Using the above material parameters gives a reference frequency of the order 100 kHz. Thus, the low-frequency range applies to the current problem.

Results and Discussion

The acoustic transmission loss TL through the axisymmetric simplified particulate filter is determined for the frequency range 20 Hz to 2000 Hz and depicted in Figure 2.

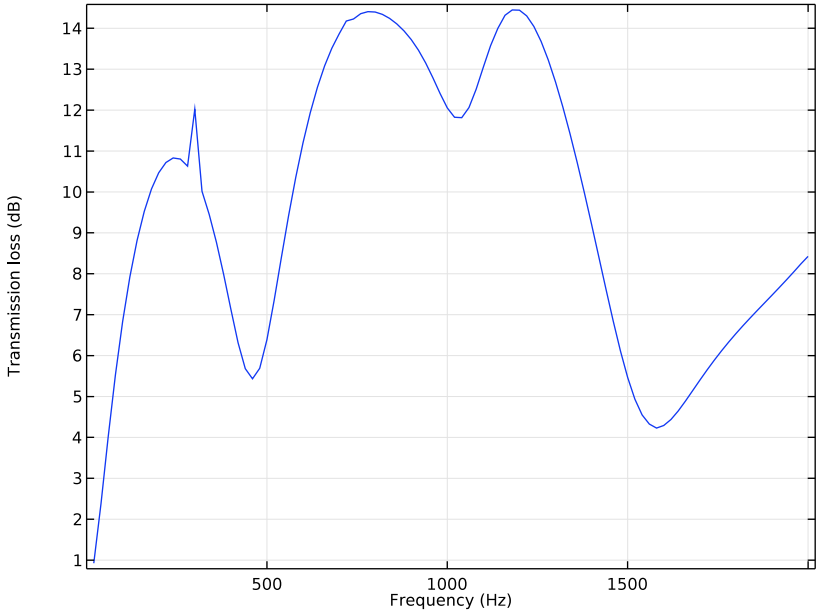


Figure 2: Transmission through the simple particulate filter as a function of frequency.

The loss is seen to be of the same order of magnitude as in real particulate filters (like diesel particulate filters, DPFs) the porous medium is often, as mentioned, structured with long ducts that decrease the acoustic damping while retaining good filtering properties. In this axisymmetric model the ducts take the form of cylindrical slits in 3D, which may introduce some nonstandard resonances in the filter. Moreover, in a real exhaust system there is an interaction between the exhaust flow and the acoustics (here the no-flow situation is studied), and the temperature is higher than 20 °C (as used here). Other physical effects include acoustic-structure and poroelastic-structure interactions with the exhaust pipe system. The present simplified model enables isolating the acoustics problem from other physical phenomena.

Figure 3 depicts the pressure distribution inside the particulate filter model for 20 Hz and for 2 kHz.

.

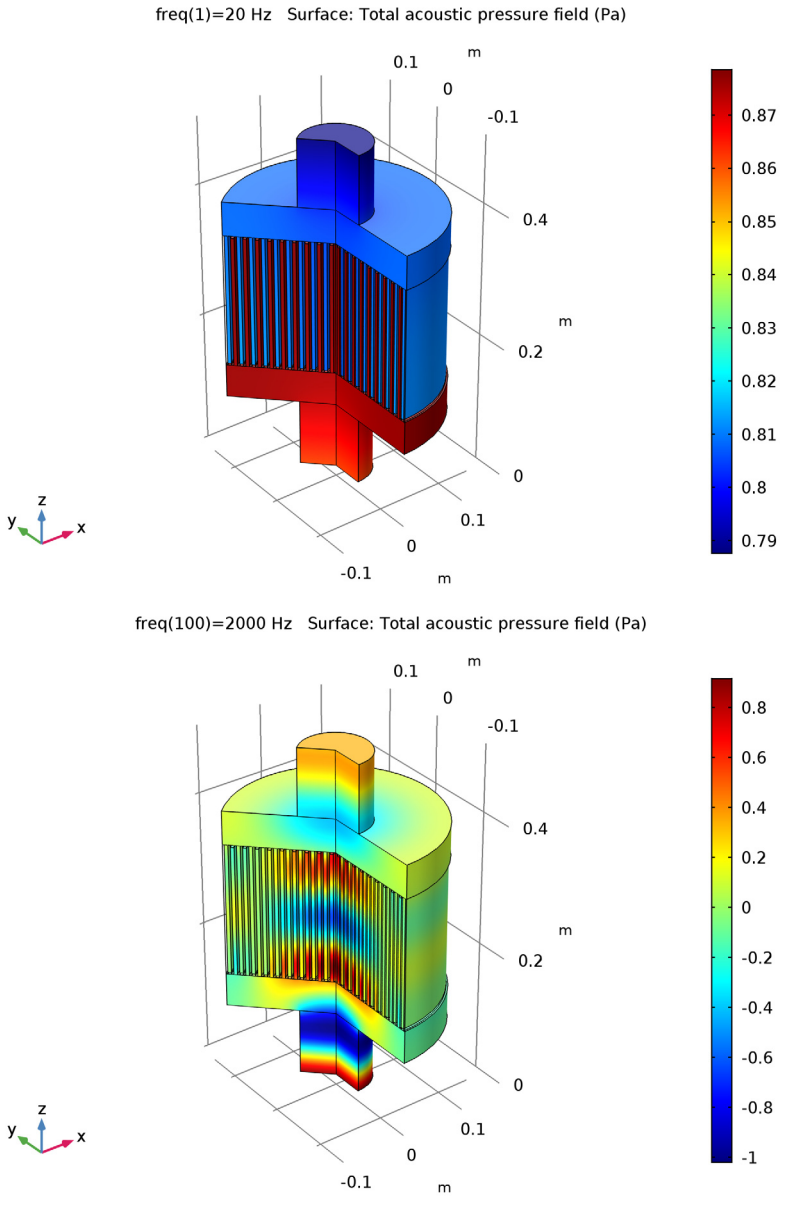


Figure 3: Pressure distribution inside the particulate filter for f = 20 Hz (top) and f = 2 kHz (bottom).

Application Library path: `Acoustics_Module/Automotive/acoustics_particulate_filter`

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-Solid-Poroelastic Waves Interaction**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `acoustics_particulate_filter_parameters.txt`.

   The parameters loaded here define the geometric dimensions and the tortuosity parameter used in the poroelastic model. Because the geometry is now parametrized, changing the dimensions in the parameters list will update the geometry automatically.

GEOMETRY 1

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rtube`.
4. In the **Height** text field, type `Ltube`.

Rectangle 2 (r2)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rfilter`.
4. In the **Height** text field, type `Lair`.
5. Locate the **Position** section. In the **z** text field, type `Ltube`.

Rectangle 3 (r3)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rfilter`.
4. In the **Height** text field, type `Lfilter`.
5. Locate the **Position** section. In the **z** text field, type `Ltube+Lair`.

Rectangle 4 (r4)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rfilter`.
4. In the **Height** text field, type `Lair`.
5. Locate the **Position** section. In the **z** text field, type `Ltube+Lair+Lfilter`.

Rectangle 5 (r5)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rtube`.
4. In the **Height** text field, type `Ltube`.
5. Locate the **Position** section. In the **z** text field, type `Ltube+2*Lair+Lfilter`.

Rectangle 6 (r6)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `dh`.
4. In the **Height** text field, type `Lfilter-ht`.
5. Locate the **Position** section. In the **r** text field, type `ht`.
6. In the **z** text field, type `Ltube+Lair`.

Rectangle 7 (r7)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `dh`.
4. In the **Height** text field, type `ht`.
5. Locate the **Position** section. In the **r** text field, type `ht`.
6. In the **z** text field, type `Ltube+Lair+Lfilter-ht`.

Rectangle 8 (r8)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `dh`.
4. In the **Height** text field, type `Lfilter-ht`.
5. Locate the **Position** section. In the **r** text field, type `2*ht+dh`.
6. In the **z** text field, type `Ltube+Lair+ht`.

Rectangle 9 (r9)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `dh`.
4. In the **Height** text field, type `ht`.
5. Locate the **Position** section. In the **r** text field, type `2*ht+dh`.
6. In the **z** text field, type `Ltube+Lair`.

Array 1 (arr1)

1. On the **Geometry** toolbar, click **Transforms** and choose **Array**.
2. Select the objects **r9**, **r8**, **r7**, and **r6** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **r size** text field, type `9`.

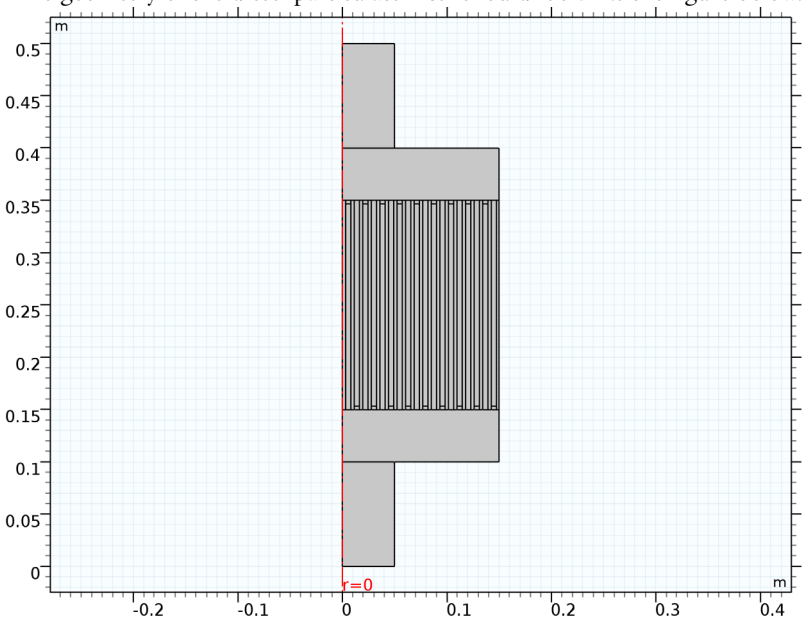
5 Locate the **Displacement** section. In the **r** text field, type `2*(dh+ht)`.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

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

The geometry of the diesel particulate filter should look like the figure below.



ADD MATERIAL

1 On 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>Air**.

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

By default, the first material node applies to all domains except where overridden by subsequent nodes in the Materials branch.

Next, create a poroelastic material with user-defined material parameters for the solid matrix.

MATERIALS

Material 2 (mat2)

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 `SiC matrix` in the **Label** text field.

The material properties for the SiC matrix will be defined after the physics interface settings.

ADD MATERIAL

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

2 In the tree, select **Built-In>Steel AISI 4340**.

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

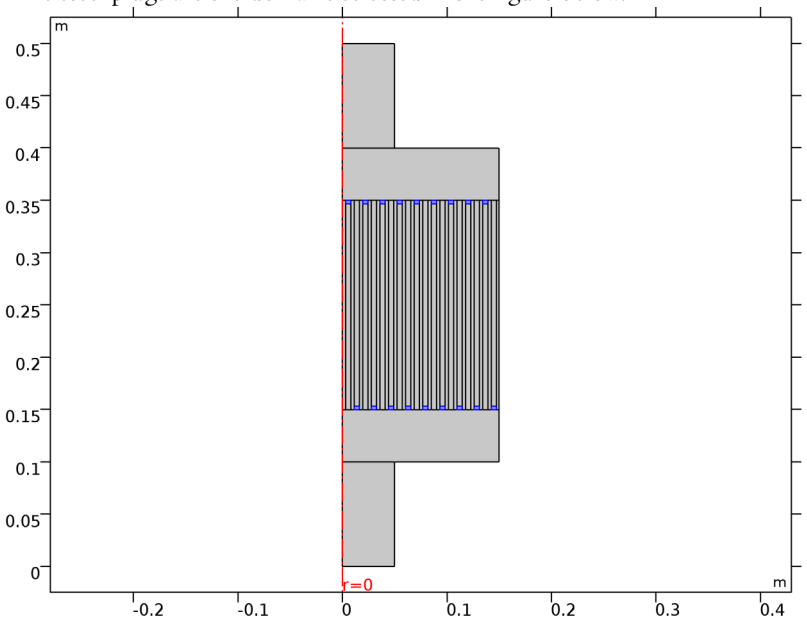
4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Steel AISI 4340 (mat3)

1 Select Domains 7, 9, 13, 15, 19, 21, 25, 27, 31, 33, 37, 39, 43, 45, 49, 51, 55, and 57 only.

The steel plugs are the domains selected in the figure below.



POROELASTIC WAVES (PELW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Poroelastic Waves (pelw)**.

2 Select Domains 3, 8, 11, 14, 17, 20, 23, 26, 29, 32, 35, 38, 41, 44, 47, 50, 53, 56, and 59 only.

Having defined the materials, you can specify the domain settings.

Poroelastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Poroelastic Waves (pelw)** click **Poroelastic Material 1**.

2 In the **Settings** window for **Poroelastic Material**, locate the **Porous Matrix Properties** section.

3 From the **Porous elastic material** list, choose **SiC matrix (mat2)**.

4 From the **Specify** list, choose **Young's modulus and Poisson's ratio**.

Use a linear elastic material model for the steel-plug domains.

SOLID MECHANICS (SOLID)

On the **Physics** toolbar, click **Poroelastic Waves (pelw)** and choose **Solid Mechanics (solid)**.

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

2 Select Domains 7, 9, 13, 15, 19, 21, 25, 27, 31, 33, 37, 39, 43, 45, 49, 51, 55, and 57 only.

POROELASTIC WAVES (PELW)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Poroelastic Waves (pelw)**.

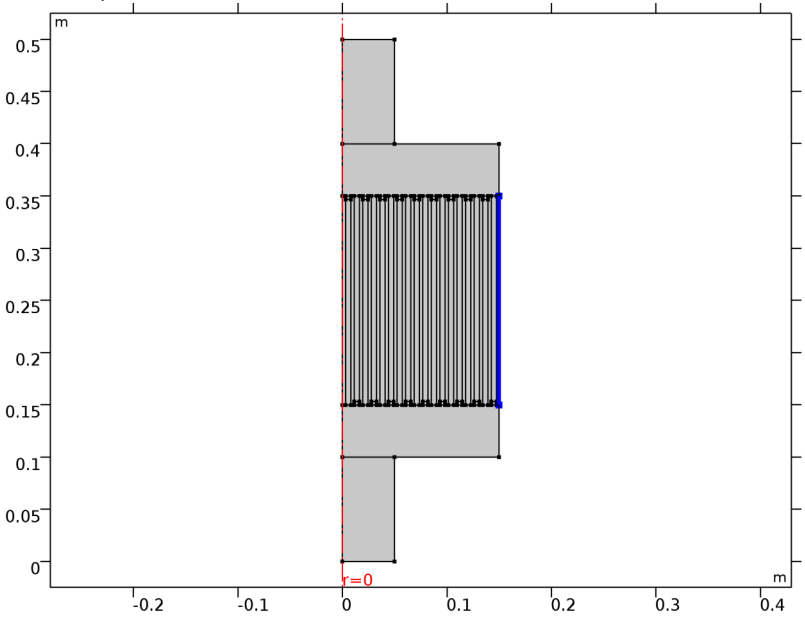
In the **Model Builder** window, under **Component 1 (comp1)** click **Poroelastic Waves (pelw)**.

Fixed Constraint 1

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

2 Select Boundary 179 only.

The porous matrix is assumed to be glued to an outer casing at the boundary highlighted in the figure below; hence the fixed constraint boundary condition. This boundary is also sound hard.

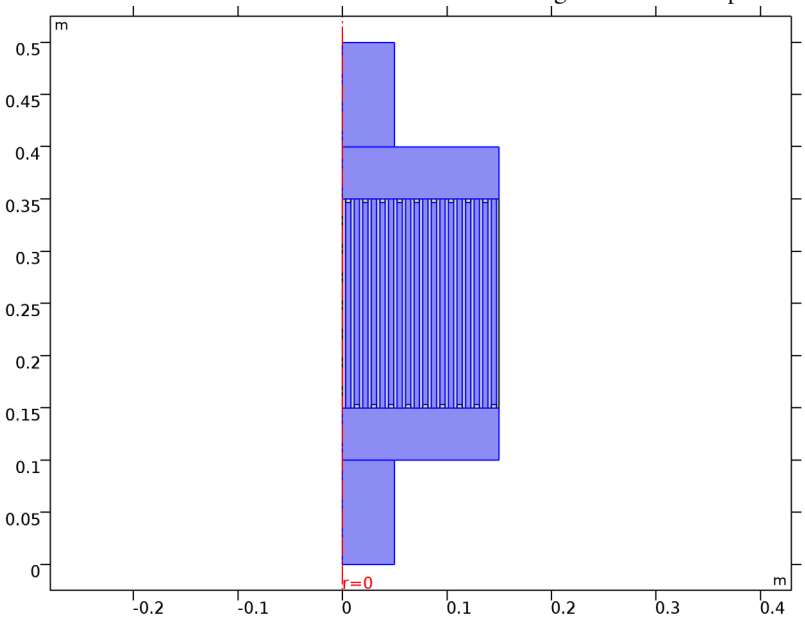


PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

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

2 Select Domains 1, 2, 4–6, 10, 12, 16, 18, 22, 24, 28, 30, 34, 36, 40, 42, 46, 48, 52, 54, and 58 only.

The selected air domain where Pressure Acoustics apply is depicted in the figure below. It consists of the inlet and outlet as well as the thin air groves inside the particulate filter.



3 In the **Model Builder** window, click **Pressure Acoustics, Frequency Domain (acpr)**.

Plane Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundaries 2 and 11 only.

Incident Pressure Field 1

1 On the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 To restrict the incoming plane wave to the inlet, click on Boundary 11 (the outlet), then right-click to remove this boundary from the selection.

3 In the **Settings** window for **Incident Pressure Field**, locate the **Incident Pressure Field** section.

4 In the p_0 text field, type `p0`.

5 From the *c* list, choose **From material**.

6 From the **Material** list, choose **Air (mat1)**.

You have now specified the domain settings. The red cross decoration for the Air and the SiC matrix nodes under Materials indicates that there are still undefined material parameters. To discover what is missing, return to the Materials branch before proceeding with the boundary conditions.

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Compressibility of fluid	chif	`1/1[atm]`	1/Pa	Basic

Recall that the compressibility of an ideal gas at the pressure P0 equals 1/P0.

SiC matrix (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **SiC matrix (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`20[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.4`	1	Basic
Density	rho	`1000`	kg/m³	Basic
Porosity	epsilon	`0.3`	1	Basic
Permeability	kappa	`1e-11`	m²	Basic
Biot-Willis coefficient	alphaB	`0.3`	1	Poroelastic material
Tortuosity factor	tau	`tauP`	1	Poroacoustics model

MULTIPHYSICS

The coupling between the acoustic and the porous domain, and between the acoustic and the elastic domain is automatically set up under the **Multiphysics** node. Click on the node

and then inspect the **Acoustic-Porous Boundary 1** node and the **Acoustic-Structure Boundary 1** node. Both nodes have the selection automatically set to **All boundaries** - this default setting automatically couples all relevant boundaries.

MESH 1

Use a mapped mesh with boundary layers added to resolve the continuity condition between the poroelastic and pressure acoustic domains.

Size

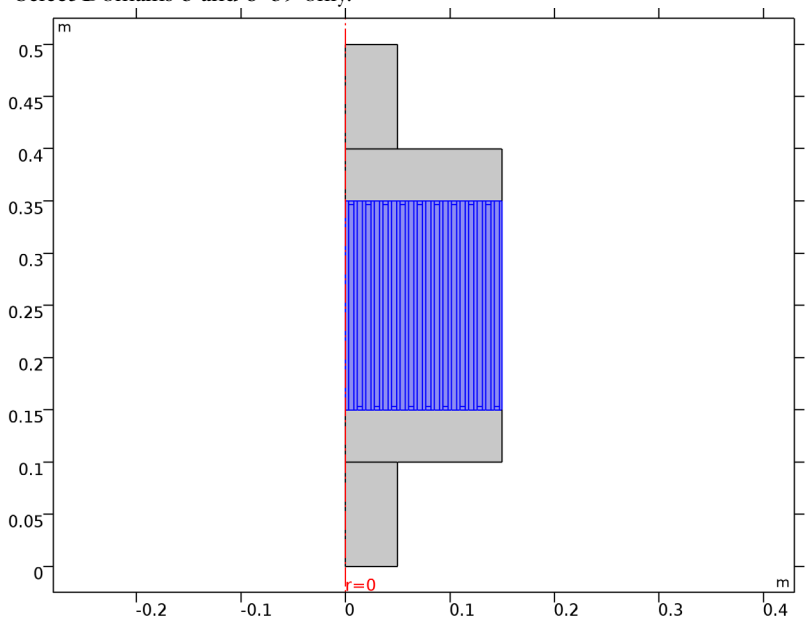
1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Extra fine**.

Size 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 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**.

   Select all the domains that make up the particulate filter, for easy selection use the **Select Box** utility.

4 Select Domains 3 and 6–59 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 `ht`.

8 Click **Build All**.

Boundary Layers 1

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

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

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

4 Select Domains 2 and 4 only.

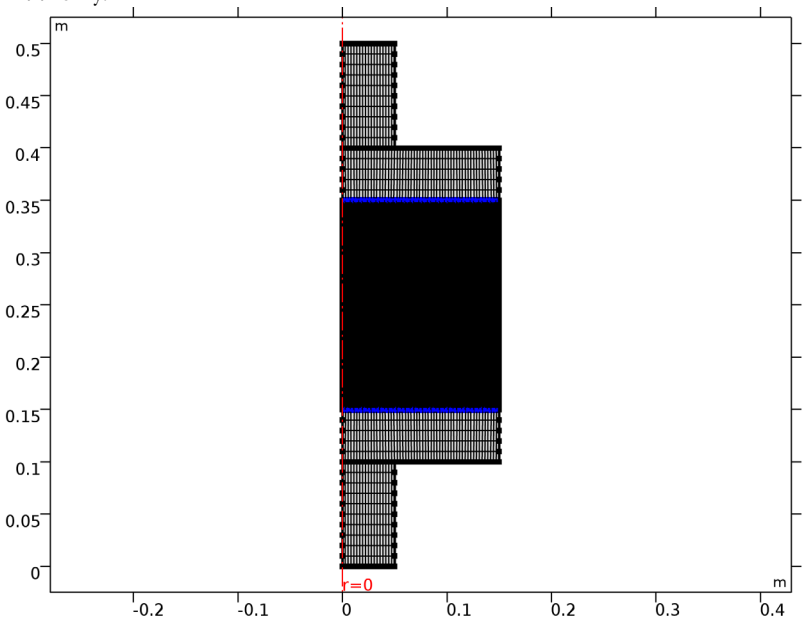
Boundary Layer Properties

Now, select all the boundaries between the particulate filter and the air, for easy selection use the **Select Box** utility. You can also paste in the numbers given below.

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 Select Boundaries 6, 8, 13, 16, 18, 20, 22, 25, 27, 29, 31, 34, 36, 38, 40, 43, 45, 47, 49, 52, 54, 56, 58, 61, 63, 65, 71, 74, 76, 78, 80, 83, 85, 87, 89, 92, 94, 96, 98, 101,

103, 105, 107, 110, 112, 114, 116, 119, 121, 123, 125, 128, 130, 132, 134, 137, 139, 141, 143, 146, 148, 150, 152, 155, 157, 159, 161, 164, 166, 168, 170, 173, 175, and 177 only.



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

4 In the **Number of boundary layers** text field, type `4`.

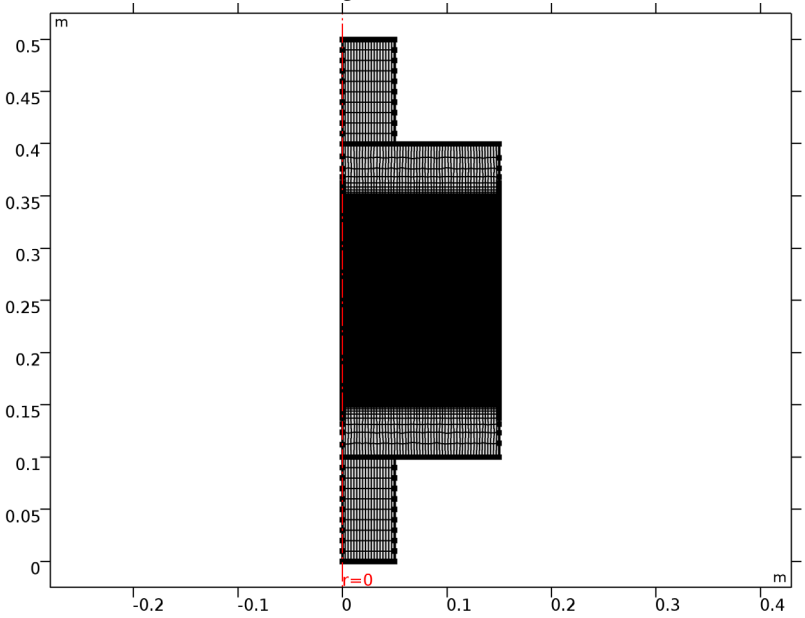
5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `ht/2`.

7 Click **Build All**.

When modeling poroelastic waves coupled to pressure acoustics, it is good practice to include a boundary layer mesh on the interface between the two physics. This is due to the fact that large gradients in the dependent variables appear here.

The mesh should look like the figure below.



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

3 In the **Frequencies** text field, type `range(20,20,2000)`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node, then click **Direct**.

4 In the **Settings** window for **Direct**, locate the **General** section.

5 From the **Solver** list, choose **PARDISO**.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Surface 2

1 In the **Model Builder** window, under **Results** right-click **Displacement (pelw)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.disp`.

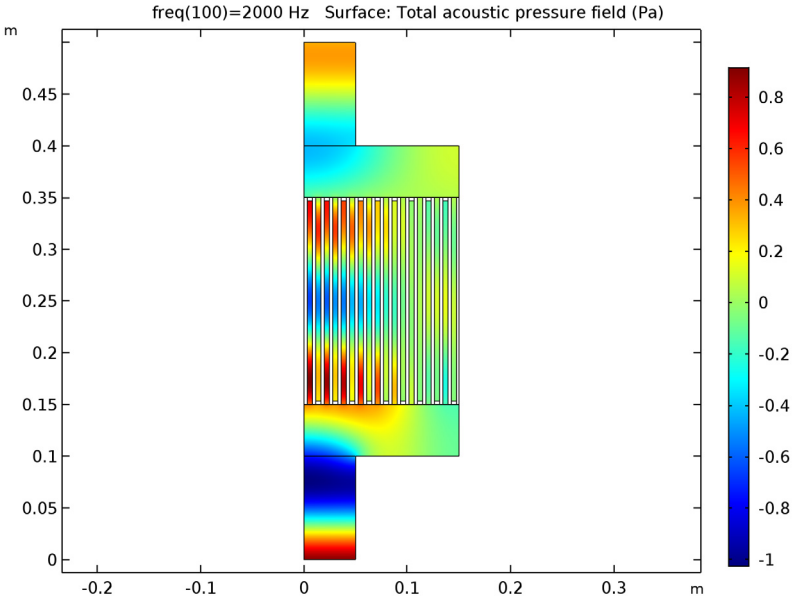
4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Acoustic Pressure (acpr)

1 Right-click **Results>Displacement (pelw)>Surface 2** and choose **Deformation**.

The plot should now look like the figure below. It depicts the total displacement.



Surface 2

1 In the **Model Builder** window, under **Results** right-click **Displacement, 3D (pelw)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.disp`.

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Deformation 1

1 Right-click **Results>Displacement, 3D (pelw)>Surface 2** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

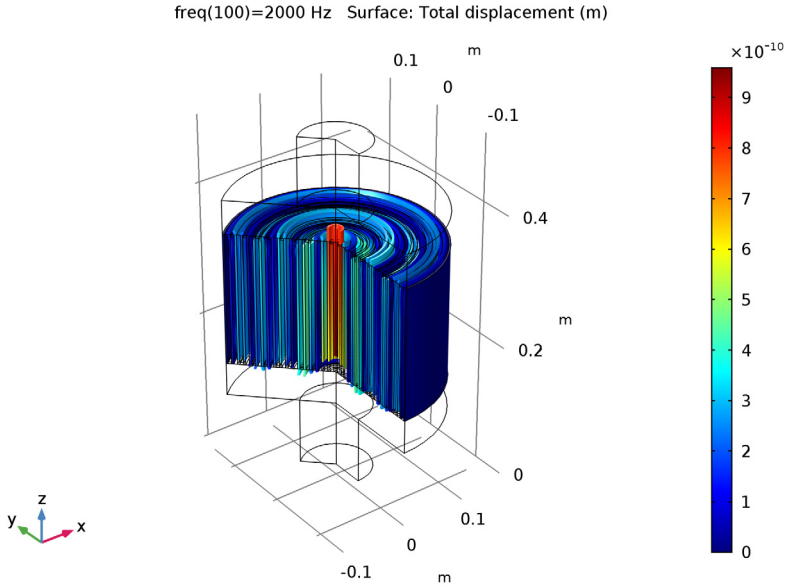
3 In the **R component** text field, type `cos(rev1phi)*u2`.

4 In the **PHI component** text field, type `sin(rev1phi)*v2`.

Displacement, 3D (pelw)

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

Total displacement in the porous matrix at 2 kHz, is depicted in the figure below.



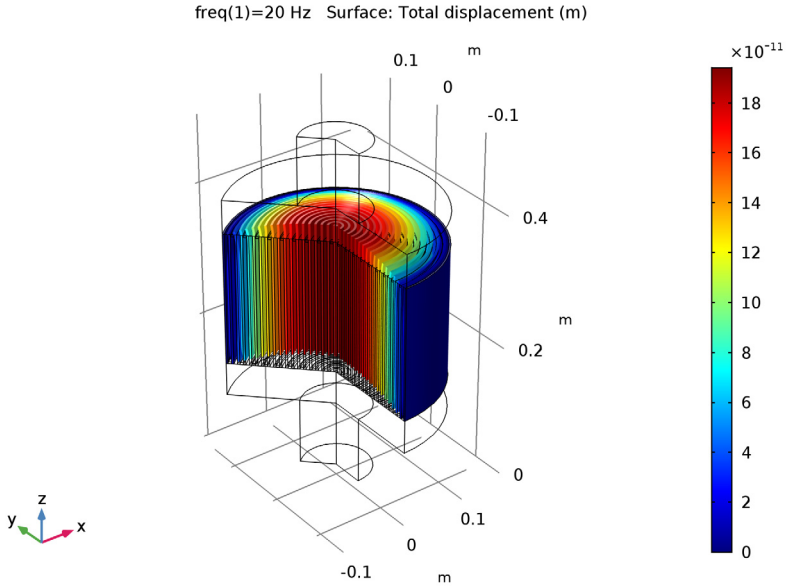
2 In the **Model Builder** window, under **Results** click **Displacement, 3D (pelw)**.

3 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

4 From the **Parameter value (freq (Hz))** list, choose **20**.

5 On the **Displacement, 3D (pelw)** toolbar, click **Plot**.

Depicted below, the total displacement in the porous matrix after changing the evaluation frequency to 20 Hz.

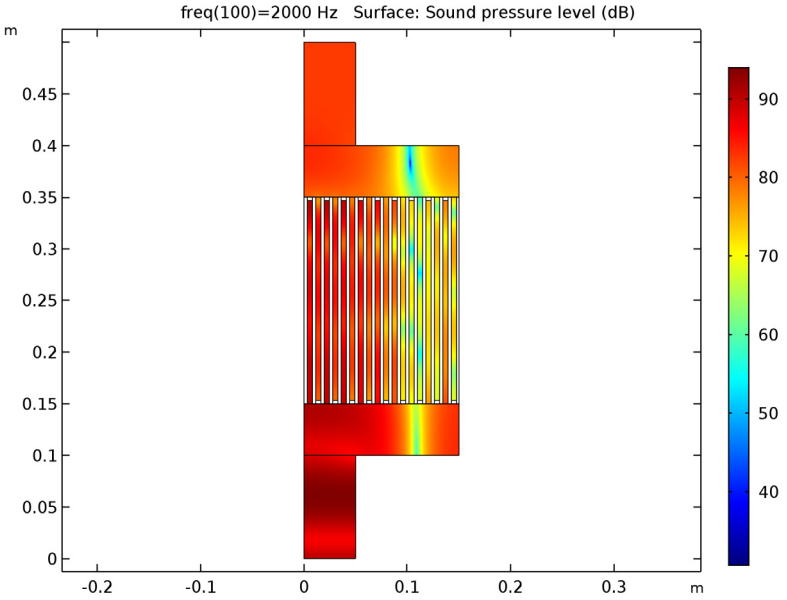


Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

The sound pressure level in dB (relative to 20 μPa) at 2 kHz, is depicted below.



Acoustic Pressure, 3D (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, 3D (acpr)**.

2 On the **Acoustic Pressure, 3D (acpr)** toolbar, click **Plot**.

The plot should look like the one depicted in Figure 3 (bottom).

3 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

4 From the **Parameter value (freq (Hz))** list, choose **20**.

5 On the **Acoustic Pressure, 3D (acpr)** toolbar, click **Plot**.

The plot, now evaluated at 20 Hz, should look like the one depicted in Figure 3 (top).

1D Plot Group 9

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

2 In the **Settings** window for **1D Plot Group**, type `SPL at inlet and outlet` in the **Label** text field.

Octave Band 1

1 On the **SPL at inlet and outlet** toolbar, click **More Plots** and choose **Octave Band**.

Select boundary as the gometric entity and the pressure is automatically averaged on that boundary.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

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

4 Select Boundary 11 only.

5 Locate the **Plot** section. From the **Style** list, choose **Continuous**.

Change the style to octave or 1/3 octave if you like the SPL given in bands.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
SPL at outlet

Octave Band 2

1 Right-click **Octave Band 1** and choose **Duplicate**.

Change the selection for the boundary.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

3 Select the **Active** toggle button.

4 Click **Clear Selection**.

5 Select Boundary 2 only.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
SPL at inlet

SPL at inlet and outlet

1 In the **Model Builder** window, under **Results** click **SPL at inlet and outlet**.

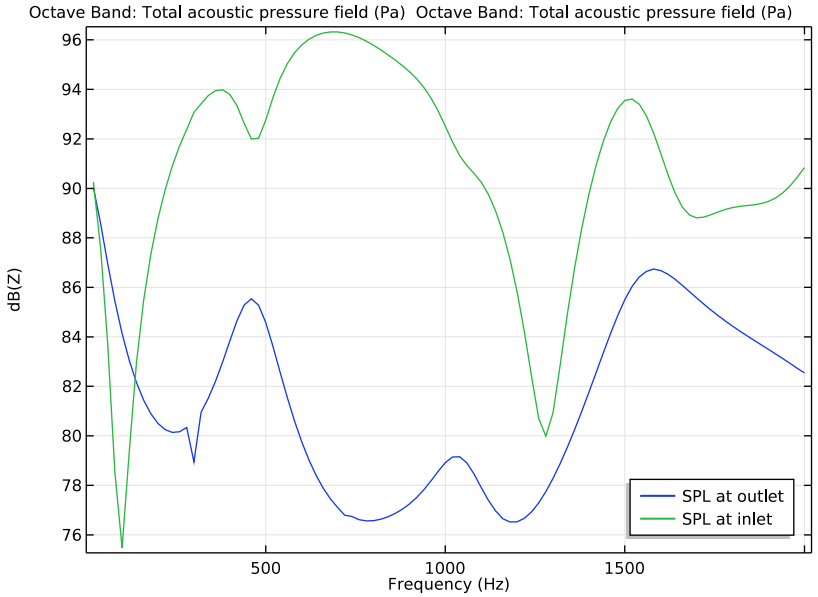
2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

3 From the **Position** list, choose **Lower right**.

4 Locate the **Axis** section. Clear the **x-axis log scale** check box.

5 On the **SPL at inlet and outlet** toolbar, click **Plot**.

The plot should look like the figure below. It depicts the sound pressure level in dB (relative to µPa) at the inlet (blue line) and the outlet (green line).



1D Plot Group 10

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

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

Octave Band 1

1 On the **Transmission Loss** toolbar, click **More Plots** and choose **Octave Band**.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

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

4 Select Boundary 11 only.

5 Locate the **y-Axis Data** section. From the **Expression type** list, choose **Power**.

6 In the **Expression** text field, type `p0^2`.

7 In the **Power reference** text field, type `acpr.p_t^2`.

8 Locate the **Plot** section. From the **Style** list, choose **Continuous**.

Transmission Loss

1 In the **Model Builder** window, under **Results** click **Transmission Loss**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

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

5 In the associated text field, type `Frequency (Hz)`.

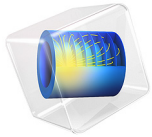
6 Select the **y-axis label** check box.

7 In the associated text field, type `Transmission loss (dB)`.

8 Locate the **Axis** section. Clear the **x-axis log scale** check box.

9 On the **Transmission Loss** toolbar, click **Plot**.

The plot should look like the one depicted in Figure 2.



Baffled Membrane

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This tutorial example demonstrates how to model the acoustic interaction between a vibrating membrane and the surrounding air. In order to focus on the principles, the geometry is kept simple and the driving mechanism is not modeled. For a more advanced loudspeaker modeling project, see the Loudspeaker Driver and Loudspeaker Driver in a Vented Enclosure models.

Note: The application uses the Acoustic-Shell Interaction, Frequency Domain multiphysics interface, which requires the Acoustics Module and the Structural Mechanics Module.

Model Definition

A titanium cone surrounded by air is set in an infinite baffle—a sound hard surface extending to infinity. The cone is modeled as a shell with a thickness of 0.5 mm. A driving mechanism, normally a voice coil, attaches to the cone and makes it vibrate. As illustrated in Figure 1, the model considers a known time-harmonic vibration with an amplitude of 1 μm, applied on the perimeter of the dust cap. The outer perimeter of the cone is left unconstrained. Two frequencies, 1 kHz and 6 kHz, are investigated.

The Acoustic-Shell Interaction multiphysics interface solves for the acoustic pressure in the air and the structural deformations in the shell. The Acoustic-Structure Boundary multiphysics coupling ensures that the structural vibrations automatically act as a sound source in the pressure acoustics equation, and the pressure force is coupled back to act on the shell

Results and Discussion

At 1 kHz, the cone is operating in piston mode, that is approximately as a rigid structure. This is the preferable situation for most loudspeakers and transducers. At 6 kHz, the cone is breaking up. This makes the sound output more difficult to control and is generally an unwanted behavior.

The figures show the displacement and deformation of the piston, the instantaneous pressure distribution, and the sound pressure level at the two frequencies.

I

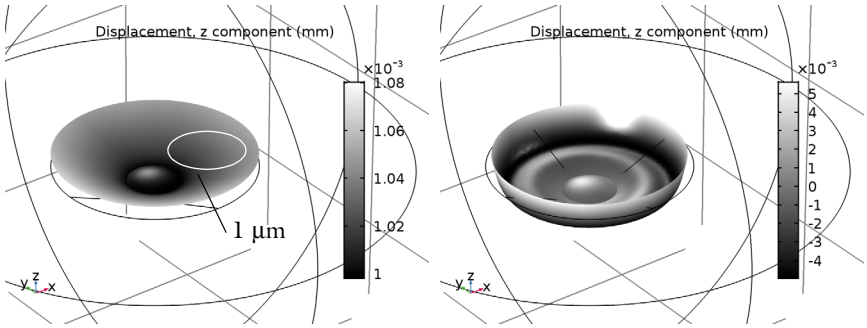


Figure 1: Local instantaneous displacement of the cone, shown as deformations and as a surface plot of the z-displacement. The plots are shown at zero phase, that is as the set displacement on the perimeter of the dustcap is peaking at 1 µm. Note the different color ranges. At 1 kHz (left), the displacement nowhere deviates by more than 8% from where the cone is attached to the driver. At 6 kHz (right), the displacement is more than five times greater at the outer perimeter of the cone.

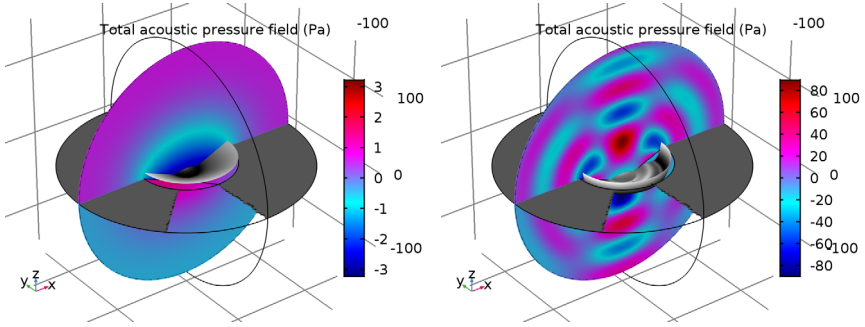


Figure 2: Acoustic pressure field at 1 kHz (left) and 6 kHz (right) shown together with the displaced and deformed membrane at zero phase.

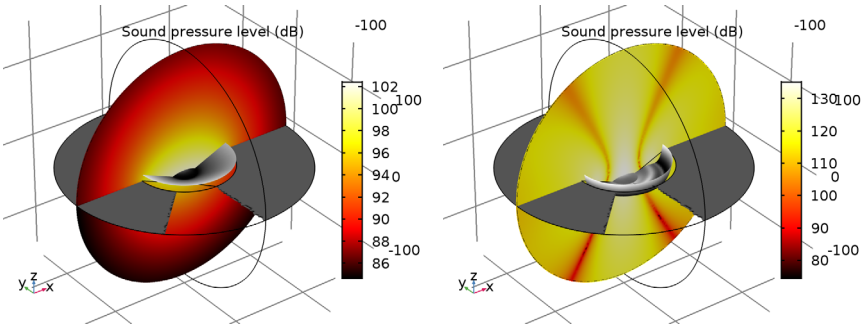


Figure 3: Sound pressure level at 1 kHz (left) and 6 kHz (right), the latter exhibiting a deep valley at around 30 degrees from the z-axis. With the same imposed displacement at both frequencies, the overall sound pressure level becomes greater at the higher frequency.

Application Library path: `Acoustics_Module/Tutorials/baffled_membrane`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Shell Interaction, Frequency Domain**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.
6. Click **Done**.

GEOMETRY 1

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

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

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

Start by creating the cone.

Cone 1 (cone1)

1 On the **Geometry** toolbar, click **Cone**.

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

3 In the **Bottom radius** text field, type `60`.

4 In the **Height** text field, type `20`.

5 From the **Specify top size using** list, choose **Angle**.

6 In the **Semi-angle** text field, type `68`.

7 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

8 In the **z** text field, type `-1`.

9 Right-click **Cone 1 (cone1)** and choose **Build Selected**.

To round off the dust cap, create a large sphere and subtract it from the cone.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

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

3 In the **Radius** text field, type `50`.

4 Locate the **Position** section. In the **z** text field, type `-65`.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **cone1** only to add it to the **Objects to add** list

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

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **sph1** only.

6 Right-click **Difference 1 (dif1)** and choose **Build Selected**.

The effect becomes apparent if you rotate the cone. Next, draw the baffle in a 2D work plane.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `150`.

4 Right-click **Circle 1 (c1)** and choose **Build Selected**.

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

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

Create an air sphere surrounding the geometry.

Sphere 2 (sph2)

1 On the **Geometry** toolbar, click **Sphere**.

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

3 In the **Radius** text field, type `150`.

4 Right-click **Sphere 2 (sph2)** and choose **Build Selected**.

5 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

The geometry is now complete except for an unnecessary surface where the cone meets the baffle. Follow these instructions to remove it.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

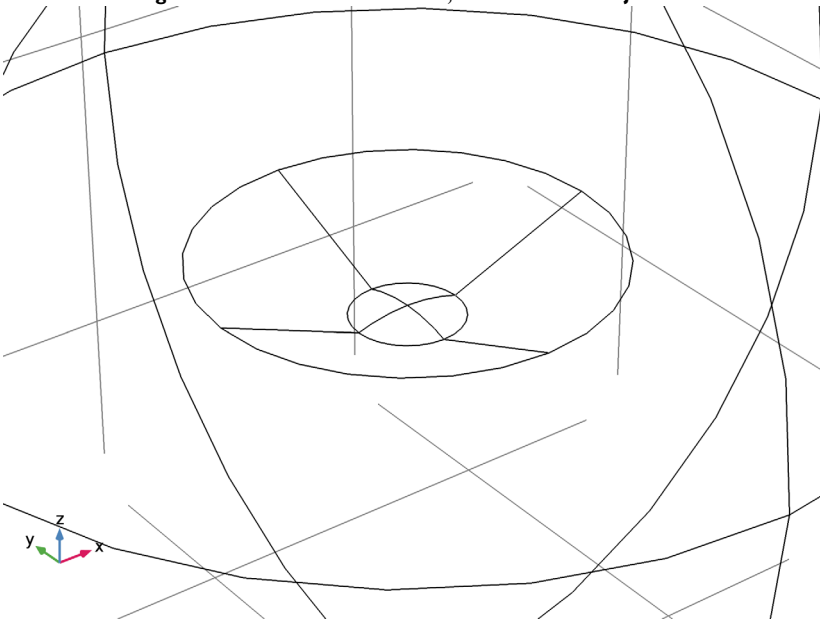
2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

Delete Entities 1 (del1)

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

2 Select the circular upper base boundary of the cone, coplanar with the baffle. The Settings tab should display uni1 8 when you are done.

3 In the **Settings** window for **Delete Entities**, click **Build All Objects**.



MATERIALS

For materials, use air in the volume and titanium on the boundaries constituting the membrane.

ADD MATERIAL

1 On 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>Air**.

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

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

6 In the tree, select **Built-In>Titanium beta-21S**.

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

8 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Titanium beta-21S (mat2)

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

2 From the **Geometric entity level** list, choose **Boundary**.

3 Use the Select Box tool to highlight all the 8 boundaries of the membrane, then right-click to select them. When you are done, the Selection list should show 6-9, 12, 13, 15, and 17. Create a Selection for quick access to these boundaries:

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Membrane` in the **Selection name** text field.

6 Click **OK**.

SHELL (SHELL)

Select the domain of application for each physics. Acoustics is already defined on the entire domain, define the domain for shell.

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

2 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.

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

4 Locate the **Thickness** section. In the d text field, type `0.5[mm]`.

Note: The thickness that you just adjusted by default applies to all shells in the model. Should you need to use different thicknesses on different shell boundaries, you can find a Change Thickness node under **Acoustic-Shell Interaction>Shell**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

On the **Physics** toolbar, click **Shell (shell)** and choose **Pressure Acoustics, Frequency Domain (acpr)**.

As the baffle in surrounded by air, define it as an Interior Sound Hard Boundary. Unlike Sound Hard Boundaries, which have acoustic domain on only one side, Interior Sound Hard Boundary defines boundary conditions on its both sides.

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 Select the baffle boundary. When you are done, the Selection list should read 3.

To complete the settings of pressure acoustics, make the exterior boundaries of the model transparent to sound radiation.

Spherical Wave Radiation 1

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

2 Select all 8 boundaries on the surface of the air sphere. When you are done, the selection list will show boundaries 1, 2, 4, 5, 10, 11, 14, and 16.

SHELL (SHELL)

Use the Prescribed Displacement/Rotation feature to drive the membrane.

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

Prescribed Displacement/Rotation 1

1 On the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement/Rotation**.

2 Select the 4 edges making up the perimeter where the cone meets the dustcap. When you are done, the Selection list should show edges 8, 10, 17, and 25.

3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

6 Select the **Prescribed in z direction** check box.

7 In the u_{0z} text field, type `1[μm]`.

MESH 1

As a rule of thumb, wave models require at least 5 quadratic mesh elements in each direction in order to be well resolved. If resonances are present more element may be needed. If you use the Maximum element size setting to achieve this, you can in practice often specify a size equal to the shortest wavelength divided by 5. The highest frequency that the model uses is 6 kHz, which corresponds to a free space wavelength of 343[m/s]/6[kHz] as wavelength = speed of sound divided by frequency.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

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 `343[m/s]/6[kHz]/5`.

5 In the **Minimum element size** text field, type `5[mm]`.

Free Tetrahedral 1

Use a slightly finer mesh in the membrane to make sure that the structural waves are also well resolved.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 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 **Boundary**.

4 From the **Selection** list, choose **Membrane**.

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 `5[mm]`.

8 Click **Build All**.

STUDY 1

The Acoustic-Shell Interaction multiphysics coupling gives default solver settings that work well for most models. What you need to do before solving is specify the frequencies you want to solve for.

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

2 Clear the **Generate default plots** check box.

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 `1[kHz] 6[kHz]`.

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

RESULTS

Create a plot group to show the deformation of the membrane.

3D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D 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 **Model>Component 1>Shell> Displacement>shell.disp - Total displacement**.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Shell>Displacement>Displacement field>w - Displacement field, Z component**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

Deformation 1

1 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

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

You should now see the cone breaking up at 6 kHz, as in the second plot of Figure 1.

Displacement

1 In the **Model Builder** window, under **Results** click **Displacement**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **1000**.

4 On the **Displacement** toolbar, click **Plot**.

At 1 kHz, the cone operates in piston mode as in the first plot of Figure 1.

Figure 2 contains this same plot but with the pressure field and the baffle superimposed. Duplicate the current plot and continue working from there.

Displacement 1

1 Right-click **Results>Displacement** and choose **Duplicate**.

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

Pressure

In the **Model Builder** window, expand the **Results>Pressure** node.

Slice 1

1 Right-click **Results>Pressure** and choose **Slice**.

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

3 From the **Plane** list, choose **ZX-planes**.

4 In the **Planes** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Disco**.

6 Select the **Symmetrize color range** check box.

Surface 1

1 In the **Model Builder** window, under **Results>Pressure** click **Surface 1**.

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

3 Clear the **Color legend** check box.

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

The plot now shows the membrane and the pressure distribution in the air, the latter using a color range centered at p=0. To color the baffle you need to create a new data set residing only on the baffle boundary.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node.

Selection

1 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

2 On the **Results** toolbar, click **Selection**.

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

4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 3 only.

Surface 2

1 In the **Model Builder** window, under **Results** right-click **Pressure** and choose **Surface**.

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

3 From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Black**.

7 On the **Pressure** toolbar, click **Plot**.

The baffle covers the view a bit unless you rotate the plot. To hide part of it, add a Filter.

Filter 1

1 Right-click **Results>Pressure>Surface 2** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `atan(y/x)<pi/4`.

4 On the **Pressure** toolbar, click **Plot**.

You should now have a reproduction of the first plot in Figure 2. Switching back to 6 kHz will give you the second one.

Pressure

1 In the **Model Builder** window, under **Results** click **Pressure**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **6000**.

4 On the **Pressure** toolbar, click **Plot**.

Figure 3 is achieved by replacing the pressure distribution with the sound pressure level.

Pressure 1

1 Right-click **Results>Pressure** and choose **Duplicate**.

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

Slice 1

1 In the **Model Builder** window, expand the **Results>Sound Pressure Level** node, then click **Slice 1**.

2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

3 Locate the **Coloring and Style** section. Clear the **Symmetrize color range** check box.

4 From the **Color table** list, choose **Thermal**.

5 On the **Sound Pressure Level** toolbar, click **Plot**.

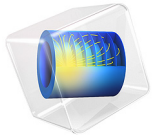
Sound Pressure Level

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **1000**.

4 On the **Sound Pressure Level** toolbar, click **Plot**.



Bessel Panel

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The Bessel panel (patented by Philips, see Ref. 1) is a way to arrange a number of loudspeakers so that the angular sound distribution resembles that of a single speaker. This benchmark model is a study of the near and far sound fields created by 25 loudspeakers arranged as an array. The solution is compared with analytical results.

The sound field outside the computational domain is determined by using the built in BEM-FEM Multiphysics coupling between *Pressure Acoustics, Frequency Domain* and *Pressure Acoustics, Boundary Elements*. The FEM interface is used to model the acoustics in the vicinity of the point sources and the BEM interface models the rest of the radiation problem.

Model Definition

A Bessel panel consists of a number of loudspeakers placed equidistantly in a row. The speakers are driven with different signals, some of them in counter-phase. For a system of five speakers, the input (voltage and current) is weighted by the factors 1, 2, 2, –2, and 1. This results in an approximately homogeneous polar far-field distribution.

This model combines five Bessel panels in the same pattern to approximate a purely radial sound field. Figure 1 is a sketch of this assembly and the input to each speaker.

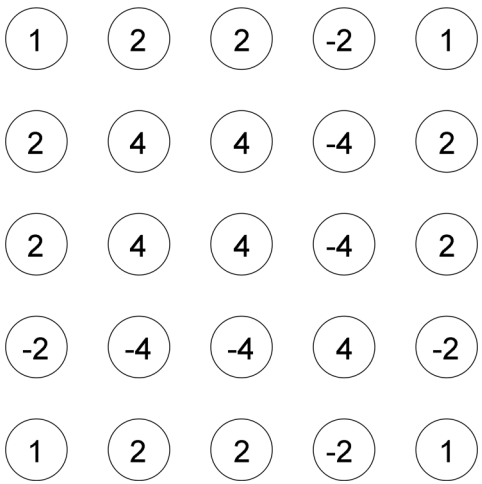


Figure 1: The Bessel panel combination used in the model. The circles represent the speakers and the numbers represent their input. Each row and each column is a Bessel panel in itself.

MODEL EQUATION

The *Pressure Acoustics, Frequency Domain* interface is used to model the pressure field near the point sources in this model. The interface solves the Helmholtz equation

$$\nabla \cdot \left(-\frac{1}{\rho}\nabla p\right) - \frac{\omega^2 p}{\rho c^2} = \sum_L Q_L \tag{1}$$

where p is the pressure, ρ is the density of the medium, $\omega = 2\pi f$ is the angular frequency, c is the speed of sound, and Q_L (SI unit: $1/\mathrm{s}^2$) is a monopole source representing a loudspeaker. In this model a frequency of $f = 200$ Hz is used. Each loudspeaker L is represented by a monopole point source emitting a flow of strength $S_L = 10^{-2}\mathrm{m}^3/\mathrm{s}\, n_L$, where n_L is the weight factor shown in Figure 1. The Helmholtz equation is also solved in the *Pressure Acoustics, Boundary Elements* interface but with all right hand side sources equal to zero.

ANALYTICAL SOLUTION

Each monopole point source is described mathematically as

$$Q_L(\mathbf{x}) = \omega S_L\, \delta(\mathbf{x} - \mathbf{x}_L)$$

where δ refers to the 3D Dirac delta function, $\mathbf{x}_L$ is the location of the speaker L, and $\mathbf{x}$ is the spatial coordinate. Let G be the Green's function for the Helmholtz equation, that is, the solution to the equation

$$(\nabla^2 + k^2)G(\mathbf{x}) = -\delta(\mathbf{x}), \qquad \mathbf{x} \in \mathbf{R}^3$$

where k is the wavenumber given by ω/c. The equation has an outgoing spherical wave solution, given by

$$G(\mathbf{x}) = \frac{e^{-ik|\mathbf{x}|}}{4\pi|\mathbf{x}|}$$

The corresponding analytical solution to Equation 1 (for a constant density) is given by the convolution of ρG and $\sum Q_L$, that is

$$p(\mathbf{x}) = \int d^3x' \rho G(\mathbf{x} - \mathbf{x}') \sum_L Q_L(\mathbf{x}') = \sum_L (\omega S_L \rho) \frac{e^{-ik|\mathbf{x} - \mathbf{x}_L|}}{4\pi|\mathbf{x} - \mathbf{x}_L|}. \tag{2}$$

This analytical solution is set up as a COMSOL Multiphysics variable and compared with the simulation result (see Figure 6).

MODEL GEOMETRY AND SETUP

The model geometry is shown in Figure 2. The distance between two neighboring loudspeakers is 0.5 m. A box extends 0.2 m around the array of points. The void domain exterior of the solid box is modeled with the BEM interface.

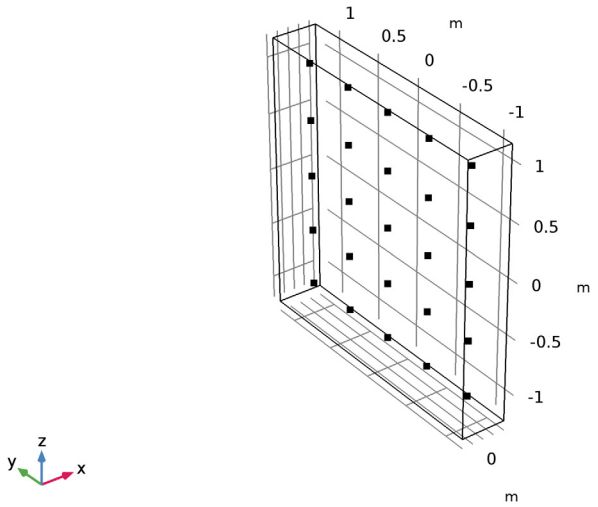


Figure 2: The model geometry.

Results and Discussion

Figure 3 shows the pressure distribution in a slice at x = 0.2 m close to the loudspeaker plane (located at x = 0 m). In this immediate vicinity of the sources, the sound field is still very inhomogeneous. The sound pressure level is shown in a slice at x = 0 m in Figure 4. In both plots the small white region can be removed by increasing the spatial resolution in the **Grid 3D** data set used for the BEM evaluation.

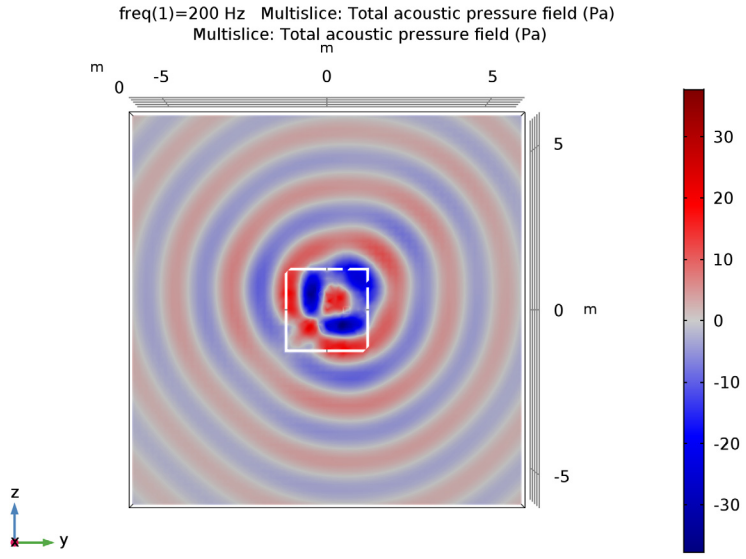


Figure 3: Slice plot of the pressure distribution at 200 Hz. The slice is parallel with the yz-plane and situated at x = 0.2 m.

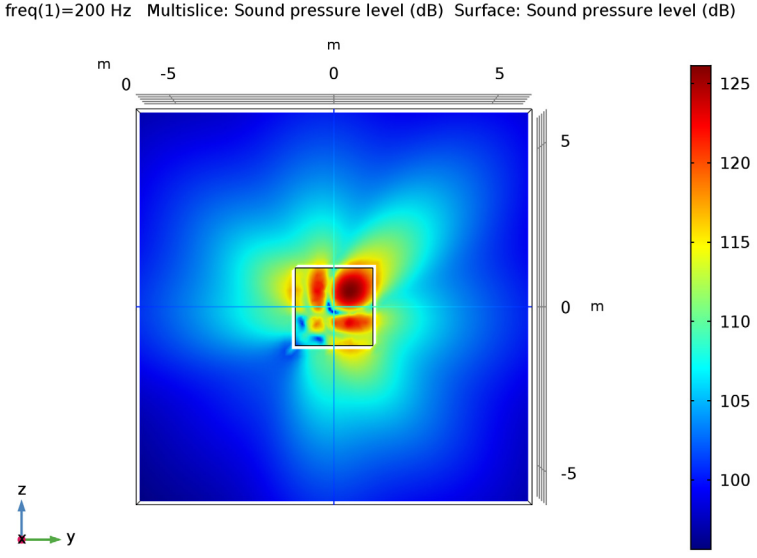


Figure 4: Slice plot of the sound pressure level distribution at 200 Hz. The slice is parallel with the yz-plane and situated at x = 0 m.

Figure 5 shows the far-field sound distribution at a distance of 100 m from the speakers. Note that the scale limits are equal to the global extremes of the sound pressure level. Hence the sound pressure level in any two given directions does not differ by more than 3 dB.

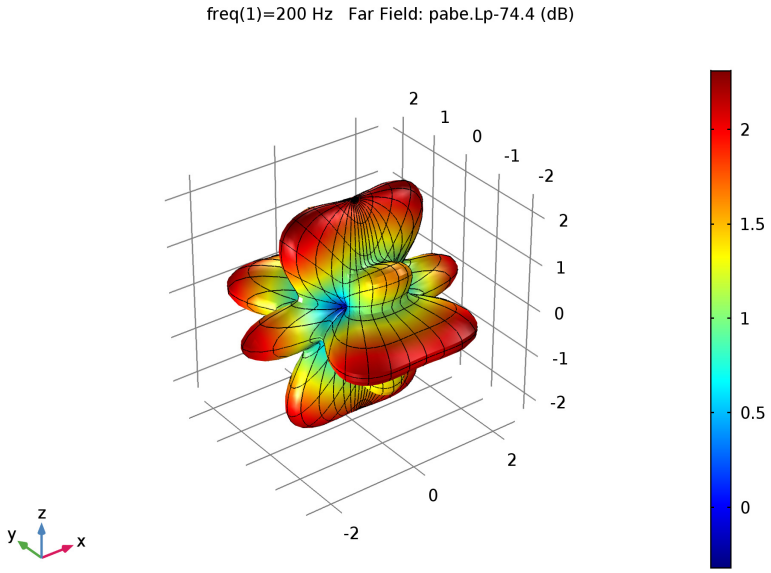


Figure 5: Sound pressure level (dB) at a distance of 100 m from the loudspeakers represented as a 3D far-field polar plot.

Figure 6 plots the computed far-field pressure at a radial distance of 100 m versus polar angle in the positive *xz*-plane and compares it to the analytical solution. As the plot shows, the computed solution is very close to the analytical solution. The accuracy could probably be increased slightly by refining the mesh.

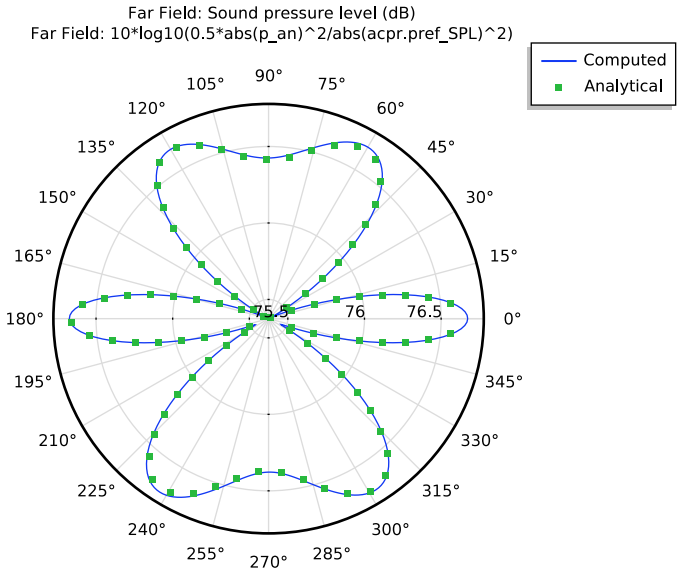


Figure 6: Sound pressure level (dB) at a radial distance of 100 m in the xz-plane (zero azimuthal angle) as a function of the polar angle from the xy-plane. The blue line represents the computed solution and the green line the analytical solution.

Reference

1. "Bessel Panels—High-power Speaker Systems with Radial Sound Distribution," *Technical Publication 091*, Philips Export BV, 1983.

Application Library path: `Acoustics_Module/Tutorials/bessel_panel`

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Boundary Elements (pabe)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

8 Click **Done**.

GEOMETRY 1

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `0.4`.

4 In the **Depth** text field, type `2.4`.

5 In the **Height** text field, type `2.4`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 Click **Build Selected**.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Point**.

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

3 In the **y** text field, type `-1`.

4 In the **z** text field, type `-1`.

5 Right-click **Point 1 (pt1)** and choose **Build Selected**.

Array 1 (arr1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

3 Select the object **pt1** only.

4 In the **Settings** window for **Array**, locate the **Size** section.

5 In the **y size** text field, type `5`.

6 In the **z size** text field, type `5`.

7 Locate the **Displacement** section. In the **y** text field, type `0.5`.

8 In the **z** text field, type `0.5`.

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

10 From the **Show in physics** list, choose **Point selection**.

This makes the object **Array 1** available as a predefined point selection, for example when you later specify flow point sources.

11 Click **Build All Objects**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
S	`0.01[m^3/s]`	0.01 m³/s	`Flow source`
f0	`200[Hz]`	200 Hz	`Frequency`
c0	`343[m/s]`	343 m/s	`Speed of sound`

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
s	`-S*i`	m³/s	`Source strength`

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `5 9 25 29` in the **Selection** text field.

7 Click **OK**.

Variables 2

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
s	2*S*i	m³/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `6 10 14 26` in the **Selection** text field.

7 Click **OK**.

Variables 3

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
s	-2*S*i	m³/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `7 8 15 19 20 24 27 28` in the **Selection** text field.

7 Click **OK**.

Variables 4

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
s	-4*S*i	m³/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type 11 17 18 22 23 in the **Selection** text field.

7 Click **OK**.

Variables 5

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
s	4*S*i	m³/s	Source strength

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type 12 13 16 21 in the **Selection** text field.

7 Click **OK**.

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

Add the material to all domains and the infinite void selection. This is all of the surroundings, where the BEM problem is solved.

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

2 From the **Selection** list, choose **All domains and voids**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Monopole Point Source 1

1 On the **Physics** toolbar, click **Points** and choose **Monopole Point Source**.

2 In the **Settings** window for **Monopole Point Source**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Array 1**.

4 Locate the **Monopole Point Source** section. In the Q_S text field, type `s`.

PRESSURE ACOUSTICS, BOUNDARY ELEMENTS (PABE)

Apply the BEM interface to the surroundings by only selecting the voids, that is, not the solid where the FEM interface is used.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Boundary Elements (pabe)**.

2 In the **Settings** window for **Pressure Acoustics, Boundary Elements**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All voids**.

MULTIPHYSICS

Now, set up the multiphysics coupling between the BEM and the FEM pressure acoustics models.

Acoustic BEM-FEM Boundary 1 (aab1)

1 On the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic BEM-FEM Boundary**.

2 In the **Settings** window for **Acoustic BEM-FEM Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

DEFINITIONS

Now set up the analytical expression for the far field (see Equation 2). Call this variable `p_an`. You will use it later for comparison with the numerical results.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 From the **Selection** list, choose **Array 1**.

Variables 6

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
r	sqrt((dest(x)-x)^2+(dest(y)-y)^2 +(dest(z)-z)^2)	m	Distance between source and observation point
p_an	intop1(s*acpr.omega*acpr.rho*exp(-i*acpr.k*r)/(4*pi*r))	Pa	Analytic pressure at observation point

The operator `dest()` evaluates its argument on the destination side, independently of the integration. In other words, `p_an` is a variable defined on the modeling domain. When evaluated in the far field it gives the analytical acoustic pressure in the point (x,y, z) defined in the far-field plot. The variables are defined globally (entire model).

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

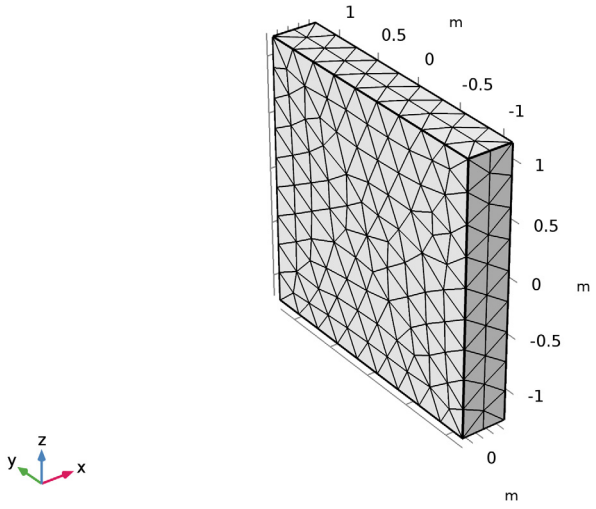
2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `c0/f0/6`.

This setting gives six elements per wavelength.

5 Click **Build All**.



STUDY 1

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `f0`.

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

RESULTS

Acoustic Pressure (acpr)

After solving the model six default plots have been created. The first three stem from the FEM model (Pressure Acoustics, Frequency Domain) and the last three from the BEM model (Pressure Acoustics, Boundary Elements).

The FEM interface creates surface plots of the pressure and the sound pressure level, as well as an isosurface plot.

The BEM interface automatically creates a **Grid 3D** data set where the BEM solution can be evaluated (using kernel evaluation). The first of the three BEM plots shows the pressure

on the BEM boundary, in this case this is the boundary in common with the FEM interface. The last two plots depict the pressure and sound pressure levels on the BEM surface as well as in cross sections through the grid data set.

Take a look at the default plots and then modify them as follows.

Grid 3D 1

1. In the **Model Builder** window, expand the **Data Sets** node, then click **Grid 3D 1**.
2. In the **Settings** window for **Grid 3D**, locate the **Parameter Bounds** section.
3. Find the **First parameter** subsection. In the **Minimum** text field, type `-2`.
4. In the **Maximum** text field, type `2`.
5. Find the **Second parameter** subsection. In the **Minimum** text field, type `-6`.
6. In the **Maximum** text field, type `6`.
7. Find the **Third parameter** subsection. In the **Minimum** text field, type `-6`.
8. In the **Maximum** text field, type `6`.
9. Click to expand the **Resolution** section. In the **y resolution** text field, type `60`.
10. In the **z resolution** text field, type `60`.

Multislice 1

1. In the **Model Builder** window, expand the **Acoustic Pressure (pabe)** node, then click **Multislice 1**.
2. In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.
3. Find the **x-planes** subsection. From the **Entry method** list, choose **Coordinates**.
4. In the **Coordinates** text field, type `0.2`.
5. Find the **y-planes** subsection. In the **Planes** text field, type `0`.
6. Find the **z-planes** subsection. In the **Planes** text field, type `0`.

Surface 1

In the **Model Builder** window, under **Results>Acoustic Pressure (pabe)** right-click **Surface 1** and choose **Disable**.

Multislice 2

1. In the **Model Builder** window, under **Results>Acoustic Pressure (pabe)** right-click **Multislice 1** and choose **Duplicate**.
2. In the **Settings** window for **Multislice**, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.
4. Locate the **Expression** section. In the **Expression** text field, type `acpr.p_t`.

5 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Multislice 1**.

6 On the **Acoustic Pressure (pabe)** toolbar, click **Plot**.

7 Click the **Go to YZ View** button on the **Graphics** toolbar.

Compare the result with that in Figure 3.

Sound Pressure Level (pabe)

Compare the result with that in Figure 4.

Go back to the default 3D view and then create plots of the spatial response of the Bessel panel.

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

3D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

Far Field 1

1 On the **3D Spatial Response** toolbar, click **More Plots** and choose **Far Field**.

2 In the **Settings** window for **Far Field**, locate the **Expression** section.

3 In the **Expression** text field, type `pabe.Lp-74.4`.

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of elevation angles** text field, type `50`.

5 In the **Number of azimuth angles** text field, type `100`.

6 Find the **Sphere** subsection. From the **Sphere** list, choose **Manual**.

7 In the **Radius** text field, type `100`.

8 Locate the **Coloring and Style** section. From the **Grid** list, choose **Finer**.

9 On the **3D Spatial Response** toolbar, click **Plot**.

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

Compare the result with the plot in Figure 5. Note that you may need to click **Zoom Extents** when looking at other figures again.

Polar Plot Group 8

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

2 In the **Settings** window for **Polar Plot Group**, type `Spatial Response in xz-plane` in the **Label** text field.

Far Field 1

1 On the **Spatial Response in xz-plane** toolbar, click **More Plots** and choose **Far Field**.

2 In the **Settings** window for **Far Field**, locate the **Expression** section.

3 In the **Expression** text field, type `pabe.Lp`.

4 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of angles** text field, type `360`.

5 Find the **Evaluation distance** subsection. In the **Radius** text field, type `100`.

6 Find the **Normal** subsection. In the **x** text field, type `0`.

7 In the **y** text field, type `1`.

8 In the **z** text field, type `0`.

The default reference direction (0,0,1) means that 0° in the polar plot corresponds to the positive x-axis direction.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
Computed

Far Field 2

1 Right-click **Far Field 1** and choose **Duplicate**.

2 In the **Settings** window for **Far Field**, locate the **Expression** section.

3 In the **Expression** text field, type `10*log10(0.5*abs(p_an)^2/abs(acpr.pref_SPL)^2)`.

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

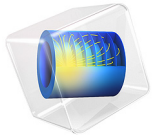
6 In the **Number** text field, type `90`.

7 Locate the **Legends** section. In the table, enter the following settings:

Legends
Analytical

8 On the **Spatial Response in xz-plane** toolbar, click **Plot**.

What you see now is the sound pressure level at a distance of 100 m from the panel as a function of the polar angle at zero azimuthal angle. This plot should resemble the one in Figure 6.



The Brüel & Kjær 4134 Condenser Microphone

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of the Brüel and Kjær 4134 condenser microphone (BK 4134). The BK 4134 is a half inch (1/2") measurement microphone for medium and high level measurements in the audio range and for coupler measurements, see Figure 1. This microphone is of the so-called pressure response type (see Ref. 1 and Ref. 2 for further details). The main output parameter of the model is the microphone sensitivity that relates the electric output of the microphone to the acoustic pressure input. The modeled sensitivity is compared to measurement data from an actual BK 4134 microphone.

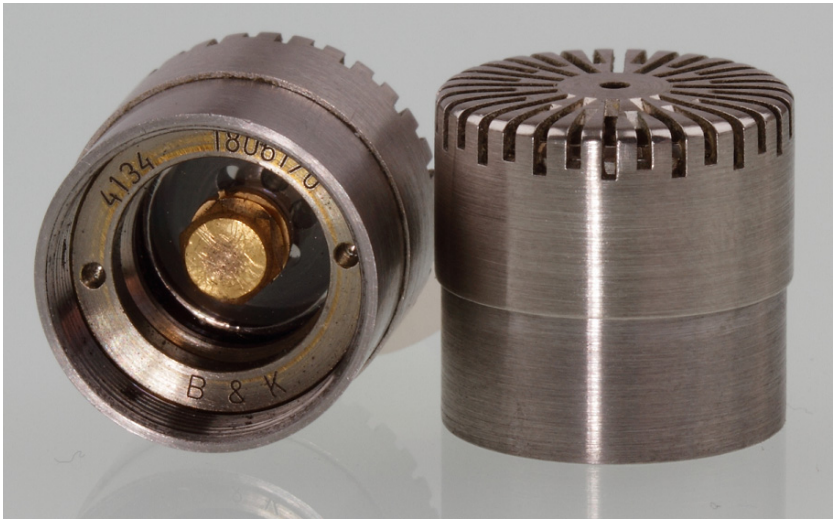


Figure 1: Picture of the Brüel and Kjær 4134 microphone including the protection grid mounted on the housing. Courtesy of Brüel and Kjær.

The BK 4134 microphone has been the subject of many modeling studies including both numerical, semi-analytical, and analytical models, see for example, Ref. 3, Ref. 4, and Ref. 5. In the analytical or semi-analytical approaches, not all effects are included as, for example, the non-trivial edge effects of the electric field and thus electric forces acting on the membrane. In this COMSOL finite element model several physics interfaces are used in a multiphysics approach to capture and couple more physical phenomena. These include:

1. Thermoviscous acoustics which is a detailed acoustic physics interface that explicitly includes and solves for thermal and viscous loss effects.
2. Electrostatics captures the changes in the electric field and electrostatic forces.

3 Membrane for setting up a pretensioned physics for the diaphragm.

4 Moving Mesh for modeling the static deformation of the membrane and computational domain when prepolarizing the microphone.

Note: For information about the different interfaces look in the COMSOL documentation. Select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Note: Many of the working principles of this microphone model are described in the Axisymmetric Condenser Microphone model. Application Library path `Acoustics_Module/Electroacoustic_Transducers/condenser_microphone`.

MICROPHONE WORKING PRINCIPLES

A schematic depiction of the microphone is given in Figure 2 including the diaphragm, backplate (or back-electrode), insulator, vent (or pressure equalization hole), housing, and protection grid. The exterior housing and diaphragm are electrically insulated from the backplate with an insulator that seals the volume inside the microphone. A small vent used for pressure equalization at low frequencies is located in the housing. The distance h between the diaphragm and the backplate is around 19 μm. The radius of the membrane R_{mic} is 4.5 mm and the height H_{mic} is 3.35 mm.

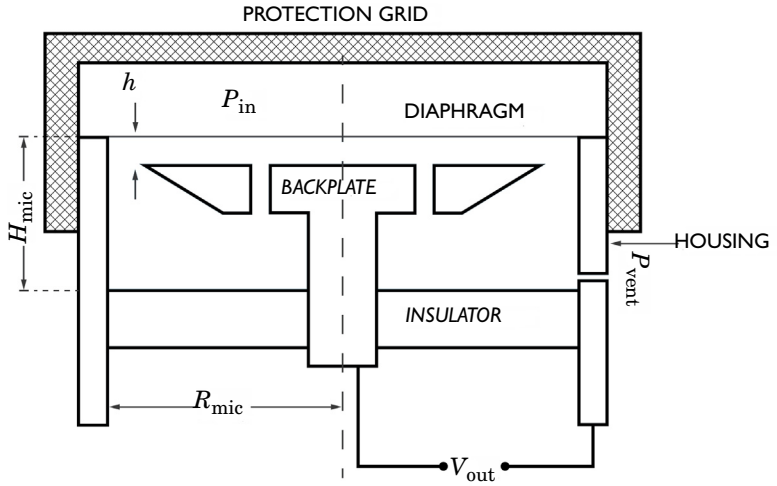


Figure 2: Schematic representation of the Brüel and Kjær 4134 microphone. Comprising the diaphragm (or membrane) and the backplate (or back electrode), the housing, the insulator, and the protection grid. The blue colored region represents the modeled part of the transducer. The microphone radius is R_{mic}, the height of the microphone volume is H_{mic}, and the distance between the diaphragm and backplate is h. The incident pressure on the microphone is P_{in} and the pressure experienced by the vent (pressure equalization hole) is P_{vent}. The output voltage if the microphone is V_{out}.

The microphone works as an electro-mechanic transducer. It transforms the mechanical movement of the diaphragm, induced by an external incident acoustic pressure field P_{in}, into an electric signal V_{out}. The relation between the input pressure and the output voltage is the sensitivity level L, defined as

$$L = 20\left(\log\left[\left|\frac{V_{out}}{P_{in}}\right| / \left(1\frac{V}{Pa}\right)\right]\right) + L_0 \quad (1)$$

where L_0 is the normalization sensitivity, here the level at 250 Hz. A charge Q_0 is applied to the backplate through a very large resistor in series with a DC polarization voltage V_{pol} = 200 V (not in the figure). This produces an electrostatic attraction and constant small DC deformation of the diaphragm. Once the diaphragm is set in motion by the incident acoustic field the gap distance h varies. This creates an AC voltage between the diaphragm/housing and the backplate.

The shape of the electrode, the location of the holes, and the gap thickness all control the viscothermal damping of the diaphragm motion and thus shapes the microphone response.

The low frequency response is influenced by the acoustic impedance of the vent. The vent may either be exposed to the incident pressure field such that $P_{vent} = P_{in} \exp(i\phi)$, where ϕ is a phase change due to distance, or unexposed (shielded) such that $P_{vent} = 0$ Pa. The first configuration is the typical when the microphone is used for field measurements. The second configuration occurs, for example, when the microphone is used for acoustic coupler measurements, where only the membrane is exposed to the sound field. More details are in Ref. 1 and Ref. 2.

MODEL ASSUMPTIONS

- In this model it is assumed that the charge Q_0 is constant. This is not fully correct. Electric interaction between the microphone and the external circuit induces small changes in the surface charge. A constant charge corresponds to charging the microphone through an infinitely large resistor. This means that only the acoustic cut-off is modeled at the low frequencies and not the usual combined electric and acoustic cut-off.
- The incident pressure field P_{in} is constant across the membrane. This is true for normal incidence. For oblique incidence, the diaphragm diameter $2R_{mic}$ becomes comparable to half a wavelength $\lambda/2$ for $f = 20$ kHz.
- The microphone casing and protective grid are not modeled. Only the blue colored region in Figure 2 is modeled.
- Using simple symmetries only the lowest order rotational periodic mode of the diaphragm is computed and modeled.

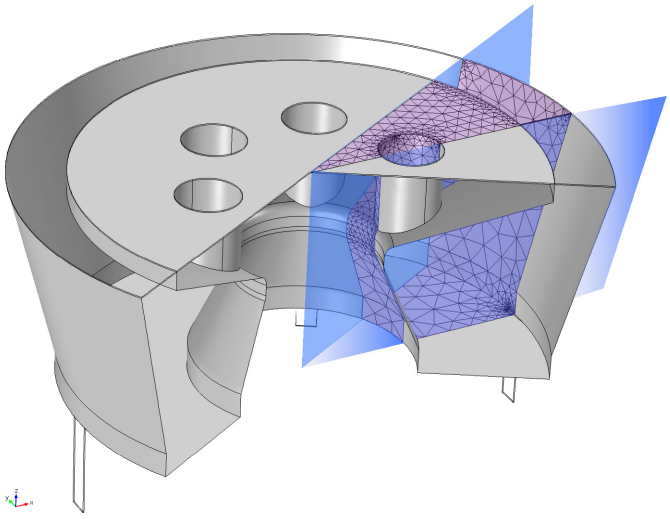


Figure 3: Geometry of the BK 4134 microphone. The computational mesh is reduced to 1/12 *of the geometry using symmetries.*

Model Definition

GEOMETRY

The geometry of the Brüel and Kjær 4134 microphone is shown in Figure 3. The computational mesh is also shown as a 1/12 slice of the geometry. Using the symmetries of the model, the computational domain is reduced. Because of this symmetric construction the vent has been split into 6 slices (each divided into two with the symmetry) with the requirement that the total acoustic impedance of the six slices equals the original single hole.

PARAMETERS

The parameters defined in the model are given in the table below. The properties of air are standard values from the COMSOL air material. The diaphragm is made of nickel and its material parameters are given in the table.

TABLE 1: MODEL PARAMETERS

VARIABLE	VALUE	DESCRIPTION
T_{m0}	3160 N/m	Membrane tension
E_m	$2.21 \cdot 10^{11}$ Ps	Young's modulus of membrane
ν_m	0.4	Poisson's ratio for membrane
t_m	5 µm	Membrane thickness
ρ_m	890 kg/m^3	Membrane density
ρ_{ms}	0.0445 kg/m^2	Membrane surface density (= $t_m \cdot \rho_m$)
Q_0	$3.145 \cdot 10^{-10}$ C	Electrode charge yielding V_{pol} = 200 V
P_{in}	1 Pa	Incident pressure amplitude
P_{vent}	1 Pa / 0 Pa	Vent pressure (exposed/unexposed)
f_{max}	20 kHz	Maximal study frequency
d_{visc}	22 µm(100 Hz/f_{max})$^{0.5}$	Viscous boundary layer thickness at f_{max}
L_0	39.5 dB	Normalization sensitivity

BOUNDARY CONDITIONS

In the exposed vent configuration the pressure at the vent is given by

$$P_{vent} = P_{in} e^{i\phi} \qquad \phi = -kH_{mic} = -\frac{\omega}{c} H_{mic} \tag{2}$$

where H_{mic} is the height of the microphone (see Figure 2), k is the wave number, ω is the angular frequency, and c = 343 m/s is the speed of sound at 20°C and 1 atm. It is here assumed that the incident sound is a plane wave normal to the diaphragm.

Details about the other boundary conditions used in this model are found in the Axisymmetric Condenser Microphone model and in the Modeling Instructions below.

Results and Discussion

The sensitivity of the microphone given by Equation 1 is shown in Figure 4. The modeled curves of the exposed and unexposed vent configurations are plotted in blue and pink, respectively. Three measurement curves of actual responses of a BK 4134 microphone are depicted in green, red, and cyan. The measured curves illustrate the variability in the

sensitivity of a microphone - this is why each measurement microphone is delivered with an individual calibration curve. The measured curves are only valid from 200 Hz and upwards. The microphone sensitivity is also illustrated in 1/3 octaves at the very end of this model description.

The frequency response of the microphone for frequencies below 100 Hz show different behavior depending on the vent configuration (compare this to Fig. 2.7 in Ref. 2). In the exposed configuration the vent equalizes the pressure on both sides of the membrane (the phase lag ϕ is small) and thus reduces the pressure drop across the diaphragm, in turn reducing the sensitivity. In the unexposed configuration the sensitivity is seen to increase at the lowest frequencies. Here the stiffness of the internal air cavity becomes smaller.

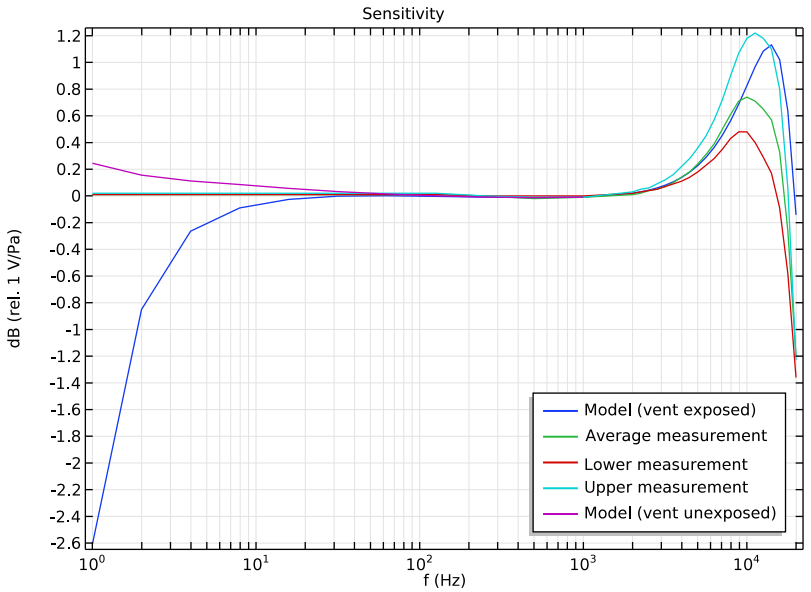


Figure 4: Microphone sensitivity curve from the model with vent exposed (blue) and vent unexposed (pink). Three measurement curves are also added (green, red, and cyan) to illustrate the variability in the microphone sensitivity.

The deformation of the membrane is shown in Figure 5 for 20 kHz and 1 kHz, top and bottom, respectively. At 20 kHz, where the sensitivity starts to fall off, the influence of the holes on the membrane deformation is visible (see for example, Ref. 4 for measured membrane modes).

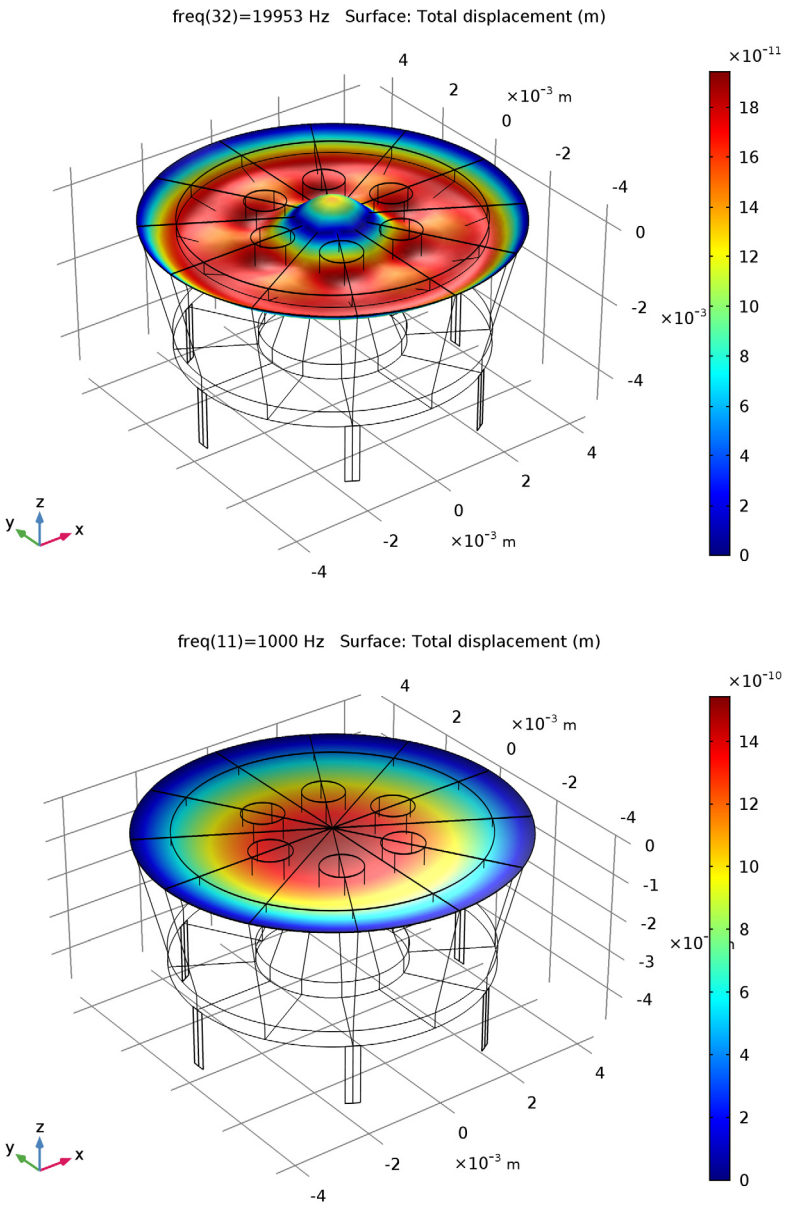


Figure 5: Diaphragm deformation at f = 20 kHz (top) and f = 1 kHz (bottom).

The static electric potential distribution that results from prepolarizing the microphone is shown in Figure 6; notice that the maximal voltage is just over 200 V.

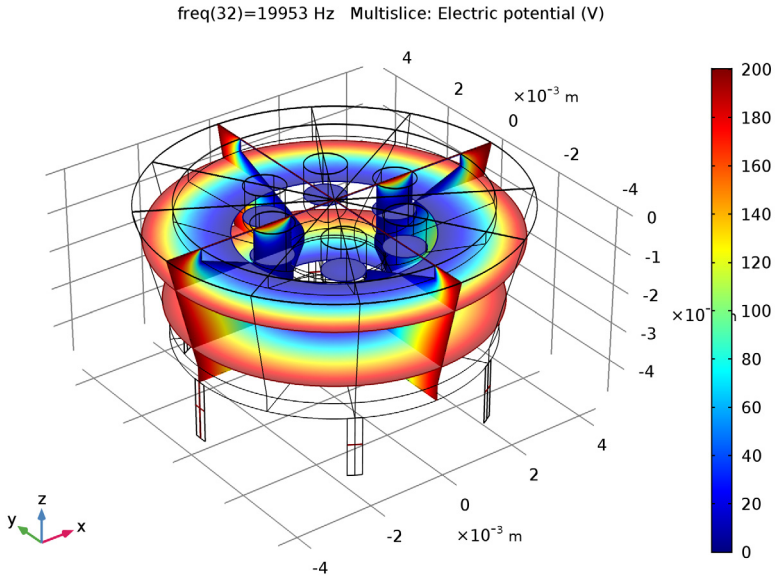


Figure 6: Static electric potential resulting from the prepolarization of the microphone cartridge and diaphragm.

References

1. Brüel & Kjær, "Condenser Microphones and Microphone Preamplifiers for Acoustic Measurements," *Data Handbook be0089,* 1982.

2. Brüel & Kjær, *Microphone Handbook, vol. 1: Theory*, Technical Documentation be1447, 1996.

3. D. Homentcovschi and R. N. Miles, "An analytical-numerical method for determining the mechanical response of a condenser microphone," *J. Acoust. Soc. Am.,* vol. 130, pp. 3698, 2011.

4. T. Lavergne, S. Durand, M. Bruneau, N. Joly, and D. Rodrigues, "Dynamic behavior of the circular membrane of an electrostatic microphone: Effect of holes in the backing electrode," *J. Acoust. Soc. Am.*, vol. 128, pp. 3459, 2010.

5. A.J. Zuckerwar, "Theoretical response of condenser microphones," *J. Acoust. Soc. Am.*, vol. 64, p. 1278, 1978.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/bk_4134_microphone`

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 **Acoustics>Thermoviscous Acoustics> Thermoviscous Acoustics, Frequency Domain (ta)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Structural Mechanics>Membrane (mbrn)**.
5. Click **Add**.
6. In the **Displacement field** text field, type `um`.
7. In the **Displacement field components** table, enter the following settings:

um
vm
wm

   The displacement field of the membrane is (um,vm,wm) while the velocity field in the fluid is (u,v,w).

8. In the **Select Physics** tree, select **AC/DC>Electrostatics (es)**.
9. Click **Add**.
10. In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.
11. Click **Add**.
12. Click **Done**.

GEOMETRY 1

Skip setting up the study types for now because a couple of manual steps are needed to set up the linear perturbation solver properly.

Import the parameters that define the diaphragm material, static surface charge, and incident and vent pressures as well as some mesh related parameters. The parameters are presented in Table 1.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `bk_4134_microphone_parameters.txt`.

   Import the geometry which represents one 12th of the Brüel and Kjær 4134 microphone, see Figure 3. The geometry is courtesy of Brüel and Kjær.

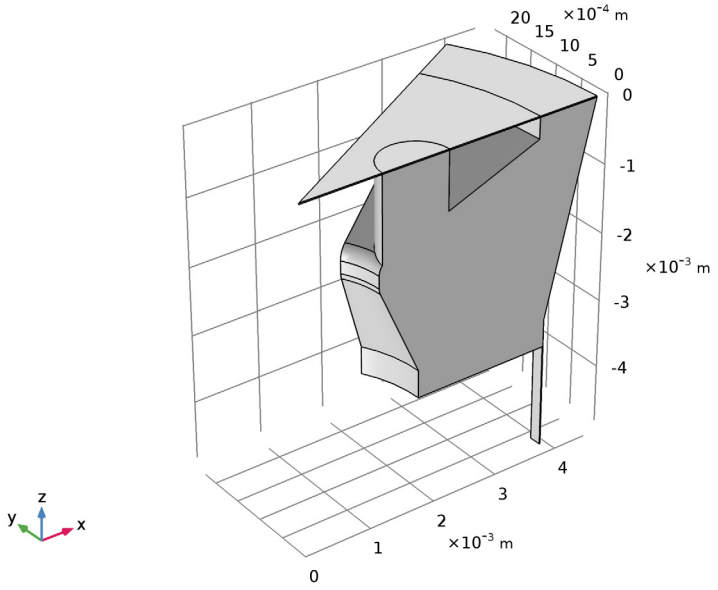
GEOMETRY 1

Import 1 (imp1)

1. On the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `bk_4134_microphone.mphbin`.

5 Click **Import**.

The geometry should look like the one in the figure below.



Next, add three interpolation functions that represent measurement data of the sensitivity of an actual microphone.

DEFINITIONS

Interpolation 1 (int1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `bk_4134_microphone_sensitivity_data.txt`.

6 In the **Number of arguments** text field, type `1`.

7 Click **Import**.

8 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
int_ave	1
int_min	2
int_max	3

Add predefined selections of boundaries to use when setting up the rest of the physics in the model. Rename the selections such that they are easy to use.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundaries 4, 16, and 26 only.

5 In the **Label** text field, type Membrane.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type Symmetry in the **Label** text field.

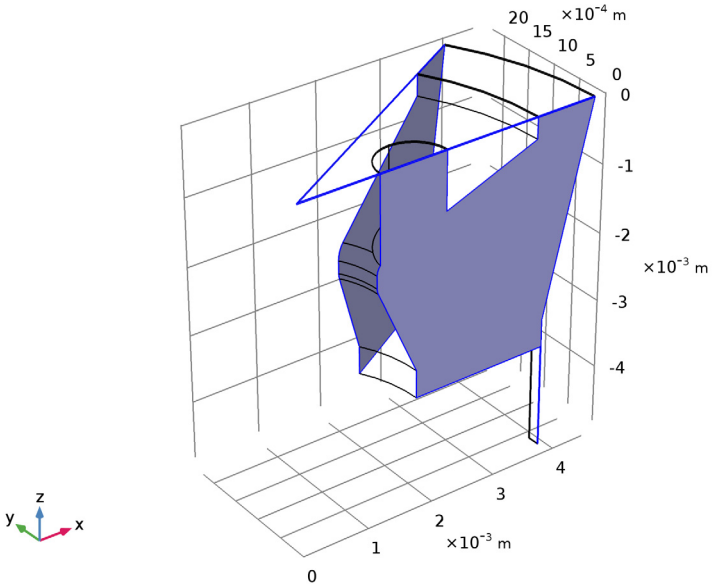
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Use the wireframe rendering for easier visualization of the selections.

5 Select Boundaries 1, 2, 5, 11, 14, 21, 24, 29, and 34 only.

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



Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Pressure Release` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 32 only.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Ground` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 3, 6–10, 12, 17, 19, and 22 only.

Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Terminal` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 4, 16, 26–28, and 36 only.

Select air as the material to be used in the model and set up a material with the membrane properties.

ADD MATERIAL

1 On 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>Air**.

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

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	0	Pa·s	Basic

Material 2 (mat2)

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 `Membrane Material` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Membrane**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	Em	Pa	Basic
Poisson's ratio	nu	num	1	Basic
Density	rho	rhom	kg/m³	Basic

6 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

To set up the acoustic model, set the acoustic velocity equal to the deformation of the diaphragm, provide two pressure boundary conditions at the vent (exposed and

unexposed), and apply symmetry conditions. When solving the model, only one of the pressure boundary conditions at the time will be active.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Symmetry 1

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

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

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

Pressure (Adiabatic) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure (Adiabatic)**.

2 In the **Settings** window for **Pressure (Adiabatic)**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pressure Release**.

4 Locate the **Pressure** section. In the p_{bnd} text field, type `linper(pvent_e*exp(-ta.iomega*Hmic/343[m/s]))`.

See the expression for the vent pressure given in Equation 2.

Pressure (Adiabatic) 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure (Adiabatic)**.

2 In the **Settings** window for **Pressure (Adiabatic)**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pressure Release**.

4 Locate the **Pressure** section. In the p_{bnd} text field, type `linper(pvent_u)`.

The `linper()` operator is used to indicate load terms that should only be included when solving the linear perturbation part of the model, that, is the frequency-dependent terms. For more information look under **Help > Documentation** and search for Special Operators.

Model the diaphragm using the Membrane interface. Constrain the membrane at the outer ridge, add an initial stress equal to the membrane tension T_{m0}, and set up zero displacement in the horizontal plane on the symmetry edges. Finally, add the forces/loads that act on the membrane, that is, the incident pressure field P_{in} and the electrostatic forces given by the Maxwell stress tensor components `(es.dnTex, es.dnTey,es.dnTez)`.

MEMBRANE (MBRN)

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

2 In the **Settings** window for **Membrane**, locate the **Boundary Selection** section.

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

4 Locate the **Thickness** section. In the *d* text field, type `tm`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Membrane (mbrn)** click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 On the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the N_0 table, enter the following settings:

TmO	0
0	TmO

Fixed Constraint 1

1 On the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.

2 Select Edge 74 only.

Symmetry 1

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

2 Select Edges 4, 5, 25, 42, 49, and 57 only.

Face Load 1

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

2 In the **Settings** window for **Face Load**, locate the **Boundary Selection** section.

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

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the *p* text field, type `linper(pin)`.

Face Load 2

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

2 In the **Settings** window for **Face Load**, locate the **Boundary Selection** section.

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

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

es.dnTex	x
es.dnTey	y
es.dnTez	z

Proceed to set up the Electrostatics interface. Add a ground boundary and a terminal boundary with a constant charge Q_0.

ELECTROSTATICS (ES)

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

Ground 1

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

2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

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

Terminal 1

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

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

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

4 Locate the **Terminal** section. In the Q_0 text field, type `Q0`.

Set up the moving mesh interface. This interface allows for a precise calculation of the stationary shape of the membrane, the electric field, and forces. Set the mesh to move/deform with the membrane and set a no-deformation condition on the symmetry boundaries.

MOVING MESH (ALE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Moving Mesh (ale)**.

2 Select Domains 1, 3, and 4 only.

Free Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.

2 Select Domains 1, 3, and 4 only.

Prescribed Mesh Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

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

4 Locate the **Prescribed Mesh Displacement** section. In the d_x text field, type `um`.

5 In the d_y text field, type `vm`.

6 In the d_z text field, type `wm`.

Prescribed Mesh Displacement 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

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

4 Locate the **Prescribed Mesh Displacement** section. Clear the **Prescribed z displacement** check box.

Finally, proceed with coupling the membrane to the acoustics using the predefined multiphysics coupling.

MULTIPHYSICS

Thermoviscous Acoustic-Structure Boundary 1 (tsb1)

1 On the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Thermoviscous Acoustic-Structure Boundary**.

2 In the **Settings** window for **Thermoviscous Acoustic-Structure Boundary**, locate the **Boundary Selection** section.

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

You have now defined all the physics, multiphysics, and boundary conditions of the model. Proceed with defining the computational mesh. Because the model is large and the mesh has to be used for a wide frequency range, some compromise is needed. If the mesh was to resolve the acoustic boundary layer for all frequencies, the model would become extremely large and difficult to solve. To reduce the mesh, the boundary layer is only properly resolved in the thin air slit between the diaphragm and backplate where most damping losses occur. At frequencies below around 1000 Hz the mesh is also adequate in the back volume. Above this frequency the losses in the back volume are minimal.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

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

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 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. Select the **Maximum element size** check box.

5 In the associated text field, type `0.5[mm]`.

Free Triangular 1

To properly resolve the solution near the edges of the holes in the backplate and the outer rim, the mesh size needs to be determined by the thickness of the acoustic boundary layer. More specifically, specify a maximum element size at these edges equal to 4 times the boundary layer thickness at 20 kHz.

Size 2

1 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 **Edge**.

4 Select Edges 24, 35, and 48 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 `4*dvisc`.

8 Click **Build Selected**.

Swept 1

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

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

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

4 Select Domains 1, 3, and 4 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `3`.

4 Click **Build Selected**.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 33 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 61 and 68 only.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 62 and 65 only.

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

4 In the **Number of elements** text field, type 2.

5 Click **Build Selected**.

Swept 2

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

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

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

4 Select Domain 5 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 2** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, click **Build Selected**.

Boundary Layers 1

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

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

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

4 Select Domain 5 only.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 Select Boundary 33 only.

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

4 In the **Number of boundary layers** text field, type `2`.

5 Click **Build Selected**.

Convert 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Convert**.

2 In the **Settings** window for **Convert**, locate the **Geometric Entity Selection** section.

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

4 Select Boundary 33 only.

5 Click **Build Selected**.

6 Click the **Zoom to Selection** button on the **Graphics** toolbar.

Size 1

1 Right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

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

4 Click the **Custom** button.

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

6 In the associated text field, type `0.5[mm]`.

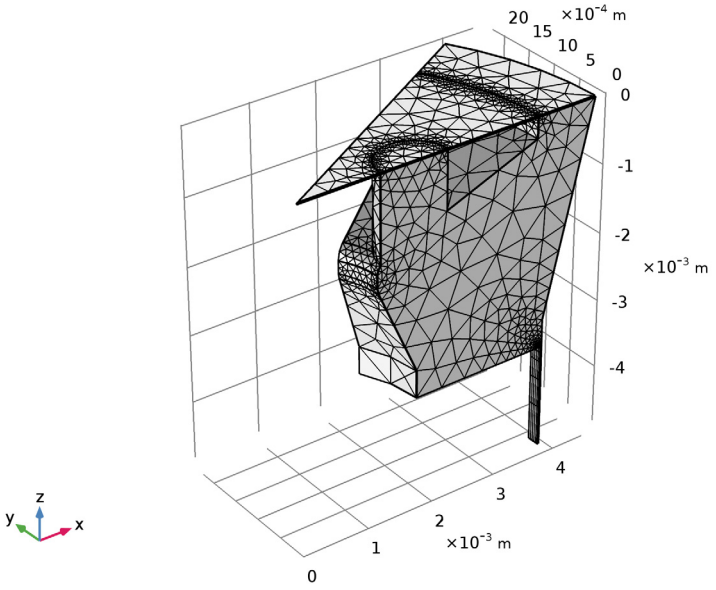
7 Select the **Resolution of narrow regions** check box.

8 In the associated text field, type `1`.

9 Click **Build All**.

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

The final mesh should look like that in the figure below.



Add two studies to solve the model: one for the case where the vent is exposed to the incident pressure P_{in} and another for the case where the vent is unexposed (shielded) from the incident pressure field.

Solve the model using the linear-perturbation solver in the frequency domain. In order for the solver to work, first perform a stationary study to determine the linearization point. This first study deforms the membrane and mesh due to the electrostatic forces (from the DC polarization voltage) and includes the effects of the static membrane tension T_{m0}. The second study models the acoustic perturbation to the static solution, that is the small-parameter harmonic variations of the acoustic pressure, temperature, and velocity.

Because this is a strongly coupled multiphysics problem, set the stationary solver to fully coupled (in contrast to the default segregated type). The frequency domain solver is fully coupeled, per default, when frequency domain perturbation is used. Also set the solvers to be direct and in the frequency domain step use PARDISO. The model should solve on a computer with 6 GB of RAM or more.

ADD STUDY

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

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Thermoacoustics, Frequency Domain (ta)** interface.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

5 Click **Add Study** in the window toolbar.

STUDY 1

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

2 Clear the **Generate default plots** check box.

Step 1: Stationary

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain Perturbation**.

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

3 Select the **Include geometric nonlinearity** check box.

Step 2: Frequency Domain Perturbation

1 In the **Model Builder** window, under **Study 1** click **Step 2: Frequency Domain Perturbation**.

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

3 Select the **Include geometric nonlinearity** check box.

4 In the **Frequencies** text field, type `10^{range(0,3/10,3)} 10^{range(3.3,1/20, 4.3)}`.

This will give you 10 frequencies on a logarithmic scale from 1 Hz to 1 kHz and further 20 frequencies from approximately 2 kHz to 20 kHz.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Thermoviscous Acoustics, Frequency Domain (ta)>Pressure (Adiabatic) 2**.

7 Click **Disable**.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** and choose **Fully Coupled**.

5 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Direct** and choose **Enable**.

6 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** node, then click **Direct 1**.

7 In the **Settings** window for **Direct**, locate the **General** section.

8 From the **Solver** list, choose **PARDISO**.

Solve the model for the case where the vent is exposed to the incoming signal P_{in}. Solving the model for all the desired frequencies may take up to 15 min.

9 On the **Study** toolbar, click **Compute**.

Set up a second study for the unexposed vent case, where P_{vent} = 0 Pa. The procedure is the same as for setting up the first study. This time, solve the model only from 1 Hz to 1 kHz as the effect of the vent exposure is in the lower frequencies. Disable the first pressure boundary condition so that only the unvented case is treated.

ADD STUDY

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

2 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Thermoacoustics, Frequency Domain (ta)** interface.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

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

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

STUDY 2

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

2 Clear the **Generate default plots** check box.

Step 1: Stationary

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain Perturbation**.

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

3 Select the **Include geometric nonlinearity** check box.

Step 2: Frequency Domain Perturbation

1 In the **Model Builder** window, under **Study 2** click **Step 2: Frequency Domain Perturbation**.

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

3 Select the **Include geometric nonlinearity** check box.

4 In the **Frequencies** text field, type `10^{range(0,3/10,3)}`.

This gives 10 frequencies on a logarithmic scale from 1 Hz to 1 kHz.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)>Pressure (Adiabatic) 1**.

7 Click **Disable**.

Solution 3 (sol3)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node.

4 Right-click **Study 2>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** and choose **Fully Coupled**.

5 Right-click **Study 2>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1>Direct** and choose **Enable**.

6 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3 (sol3)>Stationary Solver 2** node, then click **Direct 1**.

7 In the **Settings** window for **Direct**, locate the **General** section.

8 From the **Solver** list, choose **PARDISO**.

Solve the model for the case where the vent is unexposed to the incoming signal P_{in}.

9 On the **Study** toolbar, click **Compute**.

RESULTS

Four data sets have been created automatically:

- Study 1/Solution 1 contains the full solution of the exposed.
- Study 1/Solution Store 1 contains the stationary solution, that is, the linearization point, for Study 1.
- Study 2/Solution 3 contains the full solution of the unexposed vent configuration.
- Study 2/Solution Store 2 contains the stationary solution for Study 2.

1 In the **Model Builder** window, expand the **Results** node.

Data Sets

Create a Sector 3D dataset to visualize the full geometry.

Study 1/Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

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

3 From the **Frame** list, choose **Material (X, Y, Z)**.

Sector 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Sector 3D**.

2 In the **Settings** window for **Sector 3D**, locate the **Symmetry** section.

3 In the **Number of sectors** text field, type `12`.

4 From the **Transformation** list, choose **Rotation and reflection**.

5 Find the **Radial direction of reflection plane** subsection. In the **X** text field, type `0`.

6 In the **Y** text field, type `-1`.

Derived Values

Evaluate the stationary terminal voltage to see if it is equal to the polarization voltage V_{pol} = 200 V as expected.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Electrostatics> Terminals>es.V0_1 - Terminal voltage**.

3 Locate the **Expressions** section. From the **Expression evaluated for** list, choose **Static solution**.

4 Click **Evaluate**.

Next, set up 3D plots to visualize the solution in the computational domain, including membrane deformation, particle velocity, sound pressure levels, acoustic temperature variations, and the static electric potential.

3D Plot Group 1

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Sector 3D 1**.

Surface 1

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

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `mbrn.disp`.

Deformation 1

1 Right-click **Results>Membrane Deformation>Surface 1** and choose **Deformation**.

2 On the **Membrane Deformation** toolbar, click **Plot**.

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

Membrane Deformation

The plot should look like the one in Figure 5 top.

Change the evaluation frequency to 1000 Hz.

1 In the **Model Builder** window, under **Results** click **Membrane Deformation**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **1000**.

4 On the **Membrane Deformation** toolbar, click **Plot**.

The plot should look like the one in Figure 5 bottom.

3D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

Slice 1

1 Right-click **Velocity** and choose **Slice**.

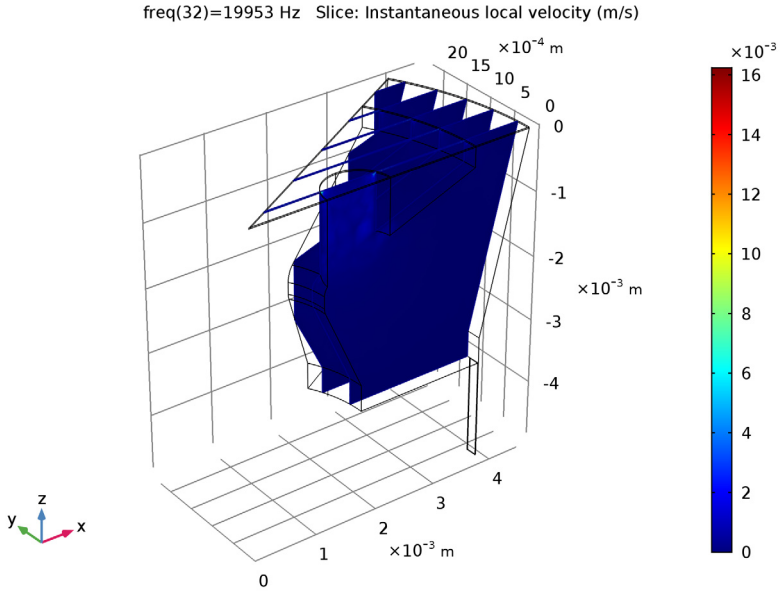
2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `ta.v_inst`.

4 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.

5 On the **Velocity** toolbar, click **Plot**.

The plot shows the instantaneous velocity amplitude. Possibly zoom to the area near the backplate perforation and the membrane to see the highest amplitudes.



3D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

Volume 1

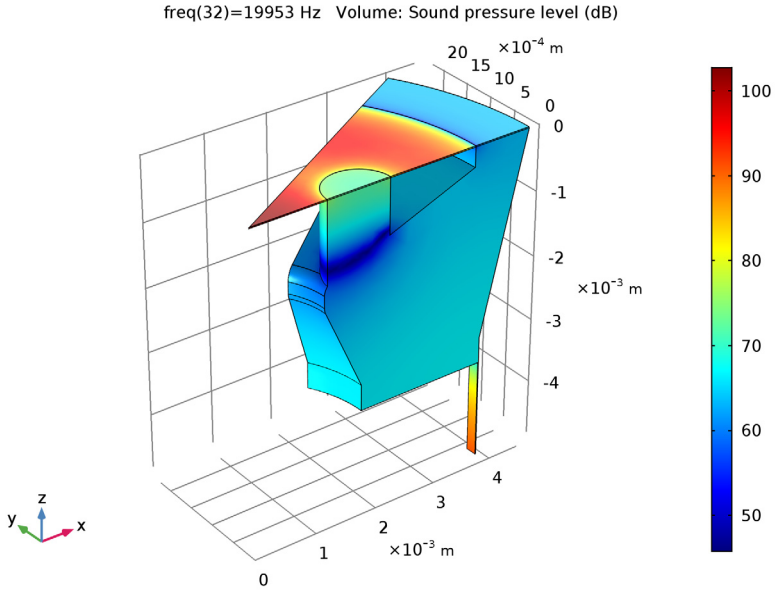
1 Right-click **Sound Pressure Level** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 In the **Expression** text field, type `ta.Lp`.

4 On the **Sound Pressure Level** toolbar, click **Plot**.

The sound pressure level distribution inside the microphone is seen here.



3D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **1000**.

Volume 1

1 Right-click **Temperature** and choose **Volume**.

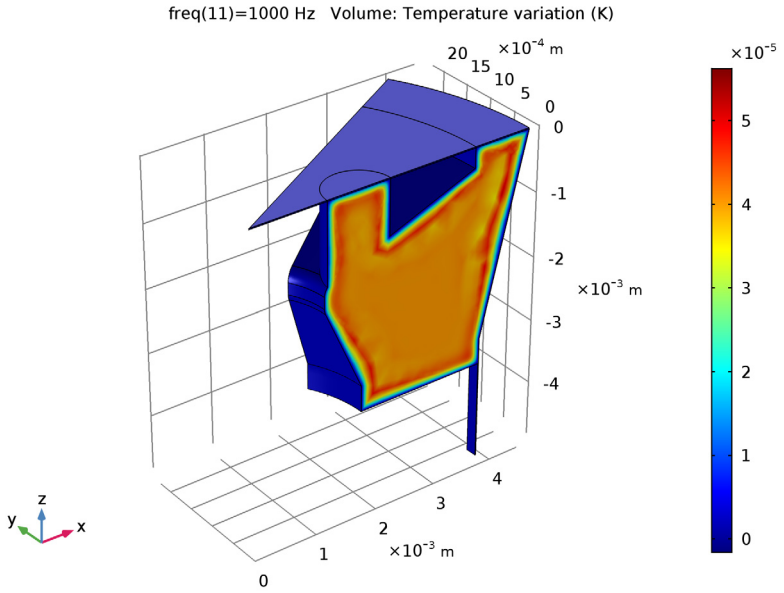
2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 In the **Expression** text field, type `T`.

4 On the **Temperature** toolbar, click **Plot**.

The acoustic temperature variation T inside the microphone is here seen at 1000 Hz. The thermal boundary layer is clearly visible. If you change the evaluation frequency to a lower value you can study the transition to the isothermal behavior. Here T is nearly

constant and 0 inside the microphone (note the min/max numerical values on the color bar).



3D Plot Group 5

1. On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Electric Potential` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Sector 3D 1**.

Multislice 1

1. On the **Electric Potential** toolbar, click **More Plots** and choose **Multislice**.
2. In the **Settings** window for **Multislice**, locate the **Expression** section.
3. In the **Expression** text field, type `V`.
4. From the **Expression evaluated for** list, choose **Static solution**.
5. Locate the **Multiplane Data** section. Find the **Z-planes** subsection. In the **Planes** text field, type `3`.

6 On the **Electric Potential** toolbar, click **Plot**.

The plot should look like the one in Figure 6.

Now create three 1D plots to visualize the microphone sensitivity, membrane deformation, and Maxwell stresses.

1D Plot Group 6

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

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

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

4 In the associated text field, type `f (Hz)`.

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

6 In the associated text field, type `dB (rel. 1 V/Pa)`.

7 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

8 In the **Title** text area, type `Sensitivity`.

9 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 Right-click **Sensitivity** 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
`20*log10(abs(es.V0_1/pin))+L0`		`Model (vent exposed)`
`int_ave(freq)`		`Average measurement`
`int_min(freq)`		`Lower measurement`
`int_max(freq)`		`Upper measurement`

4 On the **Sensitivity** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Global 2

1 In the **Model Builder** window, under **Results** right-click **Sensitivity** and choose **Global**.

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

3 From the **Data set** list, choose **Study 2/Solution 3 (sol3)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(es.V0_1/pin))+L0`		`Model (vent unexposed)`

5 On the **Sensitivity** toolbar, click **Plot**.

The plot of the microphone sensitivity should look like the one in Figure 4.

1D Plot Group 7

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

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

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

4 In the **Title** text area, type `Static Membrane Deformation`.

Line Graph 1

1 Right-click **Static Membrane Deformation** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `wm`.

4 Select Edges 4, 25, 42, and 57 only.

5 From the **Expression evaluated for** list, choose **Static solution**.

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **Static Membrane Deformation** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

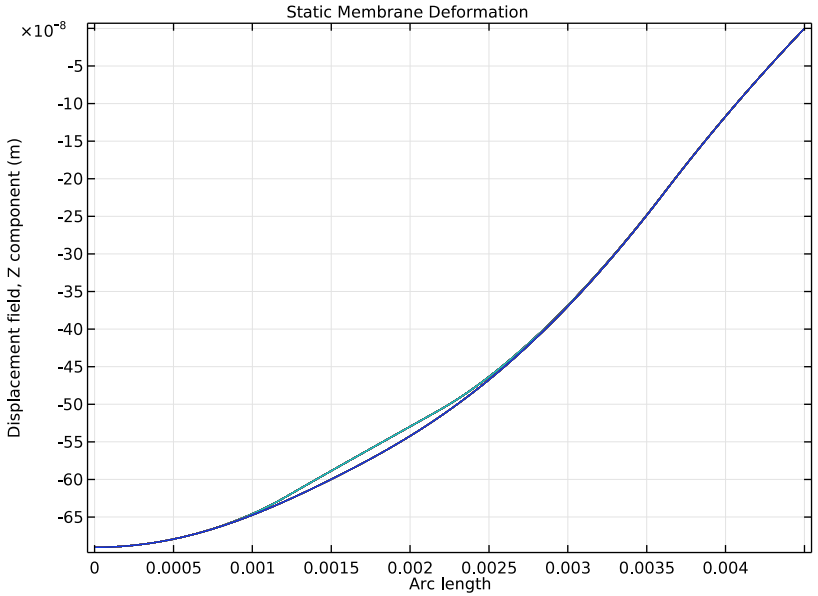
3 In the **Expression** text field, type `wm`.

4 Select Edges 5 and 49 only.

5 From the **Expression evaluated for** list, choose **Static solution**.

6 On the **Static Membrane Deformation** toolbar, click **Plot**.

The figure below shows the static deformation of the membrane due to the pre-polarization, plotted along the two symmetry boundaries. Note the small difference in the curves due to the presence of the hole in the backplate.



Static Membrane Deformation 1

1 Right-click **Static Membrane Deformation** and choose **Duplicate**.

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

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

4 In the **Title** text area, type `Static Maxwell Stress`.

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Maxwell Stress** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `es.dnTz`.

Line Graph 2

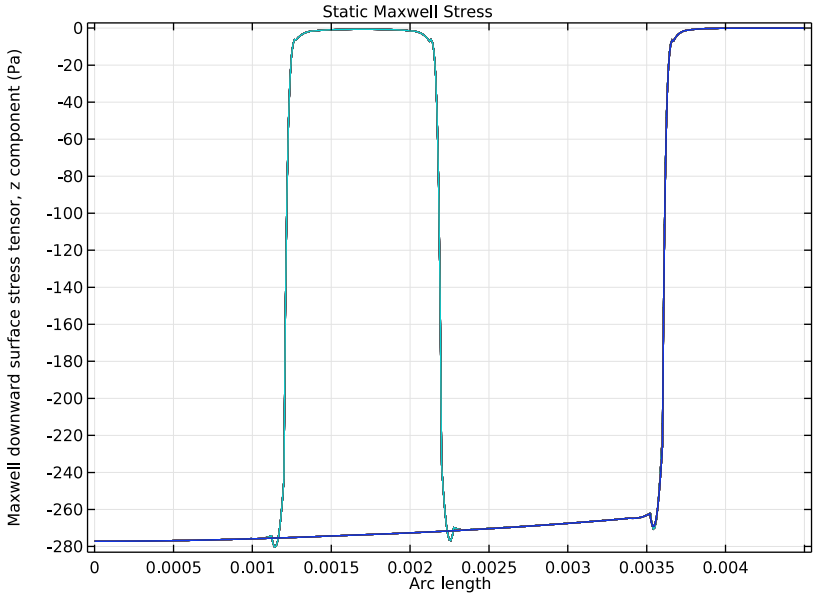
1 In the **Model Builder** window, under **Results>Maxwell Stress** click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `es.dnTz`.

4 On the **Maxwell Stress** toolbar, click **Plot**.

This figure depicts the static electric surface forces (Maxwell stresses) acting on the membrane due to the pre-polarization. Again notice the difference in the two curves, which is due to the presence of the hole in the backplate.



Finally, reproduce the sensitivity curve in 1/3 octave bands (in the exposed vent configuration) using the Octave Band plot.

1D Plot Group 9

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

2 In the **Settings** window for **1D Plot Group**, type `Sensitivity, 1/3 Octave Bands` in the **Label** text field.

Octave Band 1

1 On the **Sensitivity, 1/3 Octave Bands** toolbar, click **More Plots** and choose **Octave Band**.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

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

4 Locate the **y-Axis Data** section. From the **Expression type** list, choose **Transfer function**.

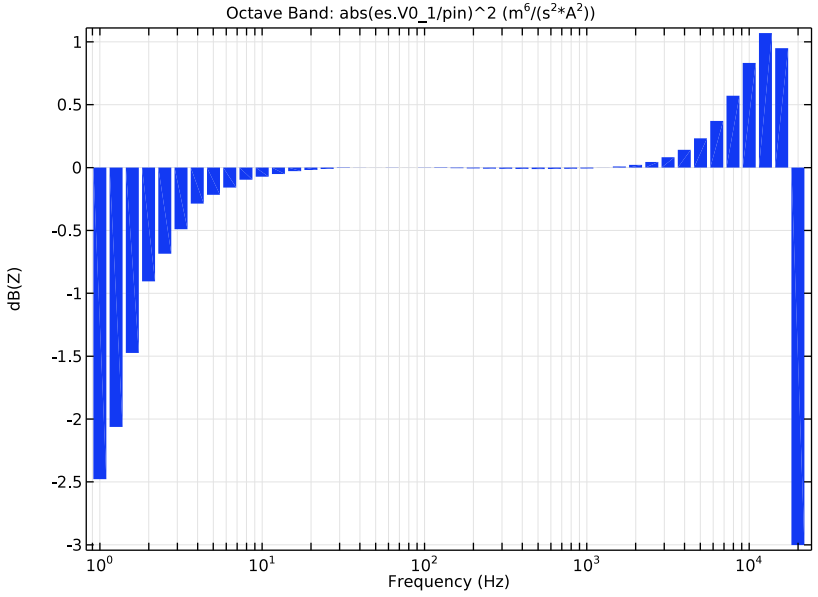
5 In the **Level reference** text field, type `L0`.

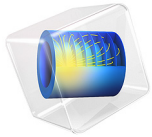
6 In the **Expression** text field, type `abs(es.V0_1/pin)^2`.

The expression represents the power transfer function H from the incident pressure to measured voltage.

7 Locate the **Plot** section. From the **Style** list, choose **1/3 octave bands**.

8 On the **Sensitivity, 1/3 Octave Bands** toolbar, click **Plot**.





Vibrations of a Disk Backed by an Air-Filled Cylinder

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The vibration modes of a thin or thick circular disc are well known, and it is possible to compute the corresponding eigenfrequencies to arbitrary precision from a series solution. The same is true for the acoustic modes of an air-filled cylinder with perfectly rigid walls. A more interesting question to ask is: What happens if the cylinder is sealed in one end not by a rigid wall but by a thin disc? This is the question addressed in this model.

Note: This application uses the Acoustic-Shell Interaction, Frequency Domain interface, which is available if you have both the Acoustics Module and the Structural Mechanics Module.

Model Definition

The geometry is a rigid steel cylinder with a height of 255 mm and a radius of 38 mm. One end is welded to a heavy slab, while the other is sealed with a steel disc only 0.38 mm thick. The disc is modeled using shell elements with the outer edge if the disc fixed. The acoustics in the cylinder is described in terms of the acoustic (differential) pressure. The eigenvalue equation for the pressure is

$$-\Delta p = \frac{\omega^2}{c^2} p$$

where c is the speed of sound and $\omega = 2\pi f$ defines the eigenfrequency, f.

A first step is to calculate the eigenfrequencies for the disc and the cylinder separately and compare them with theoretical values. This way you can verify the basic components of the model and assess the accuracy of the finite-element solution before modeling the coupled system. When computing the decoupled problem, the acoustic domain is completely surrounded by sound hard boundaries. Instead, in the coupled analysis the boundary at the disc, it has the accelerations of the disc as boundary conditions. At the same time, the acoustic pressure supplies a load on the disc. Such a coupling is set up automatically by the Acoustic-Structure Boundary multiphysics coupling.

Results and Discussion

To be able to study the effects of the coupling, we first look at the solution of the uncoupled problem. Figure 1 through Figure 4 show the two first uncoupled structural and acoustic modes.

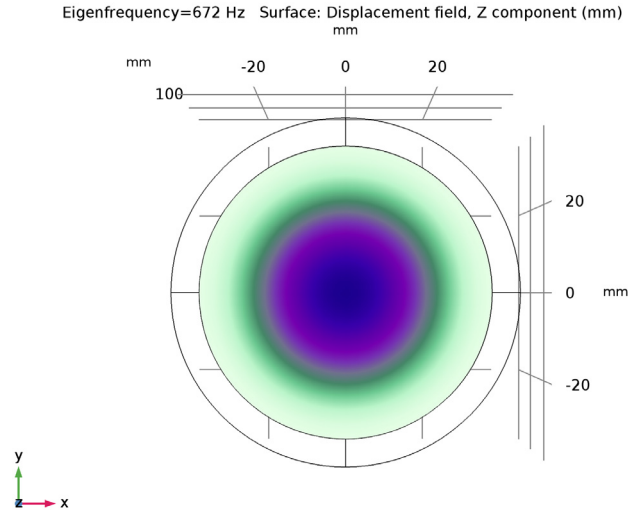


Figure 1: First structural mode displayed by vertical displacement.

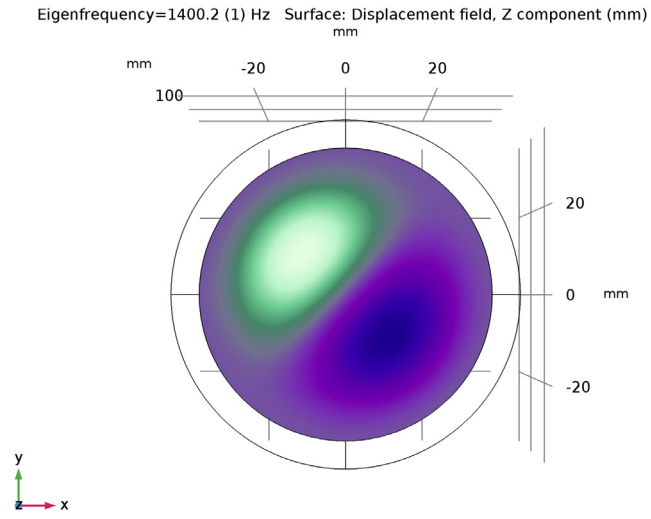


Figure 2: Second structural mode displayed by vertical displacement.

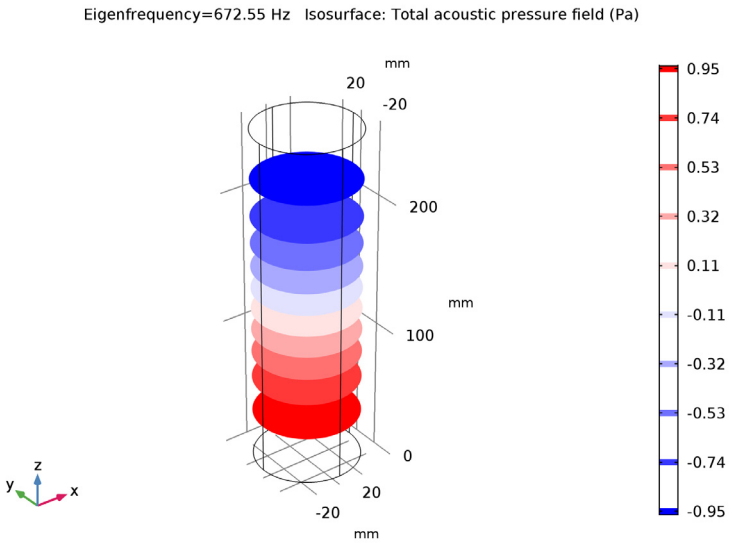


Figure 3: First acoustic mode displayed by pressure isosurfaces.

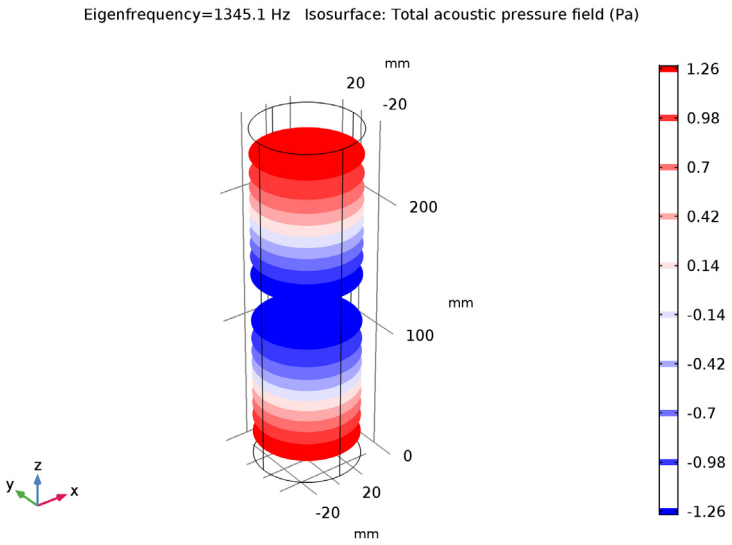


Figure 4: Second acoustic mode displayed by pressure isosurfaces.

In Ref. 1, D.G. Gorman and others have thoroughly investigated the model at hand, and they have developed a semi-analytical solution verified by experiments. Their results for the coupled problem are presented in Table 1, together with the computed results from the COMSOL Multiphysics analysis..

TABLE 1: RESULTS FROM SEMI-ANALYTICAL AND COMSOL MULTIPHYSICS ANALYSIS AND EXPERIMENTAL DATA

Dominated by	Semi-analytical (Hz)	Computed (Hz)	Experimental (Hz)
str/ac	636.9	637.1	630
str/ac	707.7	707.6	685
ac	1347	1347	1348
str	1394	1396	1376
ac	2018	2019	2040
str	2289	2292/2304	2170
str/ac	2607	2623	2596
ac	2645	2645	–
str/ac	2697	2697	2689
ac	2730	2730	2756
ac	2968	2968	2971

As the table shows, the computed eigenfrequencies are in good agreement with both the theoretical predictions and the experimentally measured values. The table also states whether the modes are structurally dominated (str), acoustically dominated (ac), or tightly coupled (str/ac). The eigenfrequency precision is generally better for the acoustically dominated modes.

Most of the modes show rather weak coupling between the structural bending of the disc and the pressure field in the cylinder. It is, however, interesting to note that some of the uncoupled modes have been split into one covibrating and one contravibrating mode with distinct eigenfrequencies. This is, for example, the case for modes 1 and 2 in the FEM solution.

In Figure 5, the first coupled mode is shown in terms of disc displacements and air pressure. The coupling effect can be clearly displayed using a plot of pressure gradients, as in Figure 6.

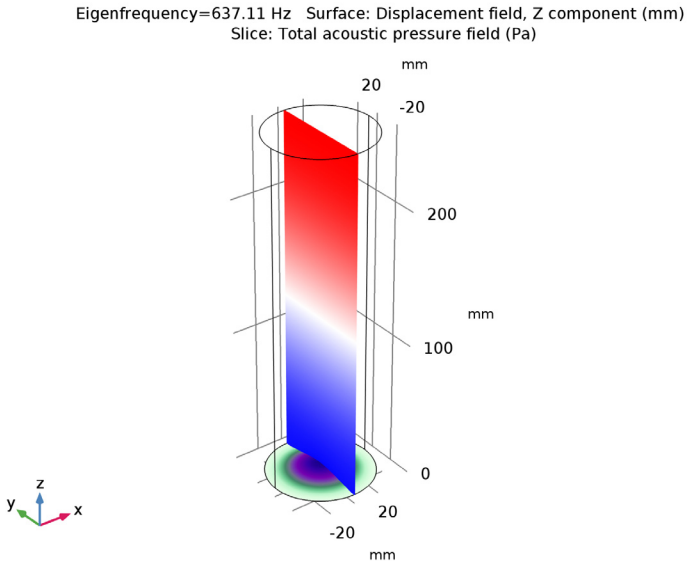


Figure 5: Disc deformation and pressure contours for the first coupled mode.

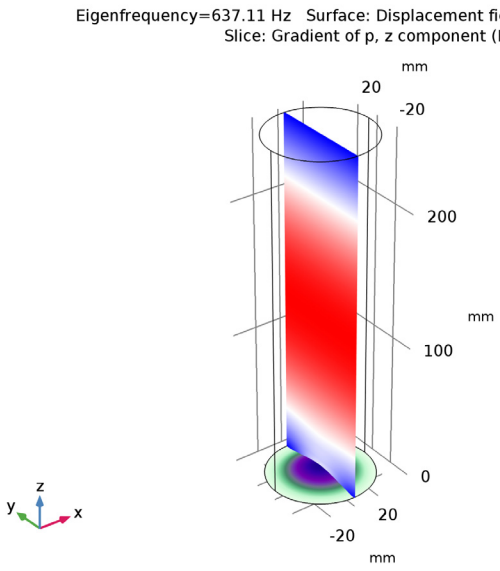


Figure 6: Disc deformation and pressure gradient contours for the first coupled mode.

Reference

1. D.G. Gorman, J.M. Reese, J. Horacek, and D. Dedouch, "Vibration Analysis of a Circular Disk Backed by a Cylindrical Cavity," *Proc. Instn. Mech. Engrs.*, Part C, vol. 215, no11, pp. 1303–-1311. 2001.

Notes About the COMSOL Multiphysics Implementation

Specify the part of the physics for which to compute the uncoupled eigenvalues by selecting the variables for the eigenvalue solver.

This model uses the Acoustic-Shell Interaction, Frequency Domain interface. If you decide to couple Pressure Acoustics, Frequency Domain to Shell manually, be careful when selecting the sign of the coupling terms so that they act in the intended direction. You should specify the acceleration in the inward normal direction for the pressure acoustics domain, which in this case is the positive z-acceleration of the disc. The acceleration is denoted `wtt` as it is the second time-derivative of the variable `w`. The pressure on the shell can be given using global directions, so that a positive pressure acts

as a face load in the negative z-direction. This is handled automatically by the Acoustic-Structure Boundary multiphysics coupling.

Application Library path: `Acoustics_Module/Verification_Examples/coupled_vibrations`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Shell Interaction, Frequency Domain**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Eigenfrequency**.
6. Click **Done**.

GEOMETRY 1

1. In the **Model Builder** window, click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Units** section.
3. From the **Length unit** list, choose **mm**.

Cylinder 1 (cyl1)

1. On the **Geometry** toolbar, click **Cylinder**.
2. In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `38`.
4. In the **Height** text field, type `255`.
5. Click **Build Selected**.

SHELL (SHELL)

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

2 In the **Settings** window for **Shell**, locate the **Thickness** section.

3 In the *d* text field, type `0.38[mm]`.

4 Locate the **Boundary Selection** section. Click **Clear Selection**.

5 Select Boundary 3 only.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.

2 Select Edges 2, 3, 7, and 10 only.

MATERIALS

Material 1 (mat1)

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 `Acoustic Material` in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Density	rho	1.2	kg/m³	Basic
Speed of sound	c	343	m/s	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	210[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7800	kg/m³	Basic

MESH 1

Free Quad 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Quad**.

2 Select Boundary 3 only.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 10.

5 Click **Build Selected**.

Free Quad 1

In the **Settings** window for **Free Quad**, click **Build Selected**.

Swept 1

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

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

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

STUDY 1

In the first study, you solve only the structural problem.

1 In the **Settings** window for **Study**, type `Structural Analysis` in the **Label** text field.

STRUCTURAL ANALYSIS

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Structural Analysis** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type 20.

5 In the **Search for eigenfrequencies around** text field, type 500.

6 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Pressure Acoustics, Frequency Domain (acpr)** interface.

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

RESULTS

Mode Shape (shell)

1 In the **Model Builder** window, under **Results** click **Mode Shape (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape, Structural Analysis` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Mode Shape, Structural Analysis** 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 **Model>Component 1>Shell> Displacement>Displacement field>w - Displacement field, Z component**.

3 On the **Mode Shape, Structural Analysis** toolbar, click **Plot**.

4 Click the **Go to XY View** button on the **Graphics** toolbar.

5 Click the **Scene Light** button on the **Graphics** toolbar.

Mode Shape, Structural Analysis

1 In the **Model Builder** window, under **Results** click **Mode Shape, Structural Analysis**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **1400.2 (1)**.

4 On the **Mode Shape, Structural Analysis** toolbar, click **Plot**.

Undeformed Geometry (shell)

1 In the **Model Builder** window, under **Results** click **Undeformed Geometry (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Undeformed Geometry, Structural Analysis` in the **Label** text field.

Thickness and Orientation (shell)

1 In the **Model Builder** window, under **Results** click **Thickness and Orientation (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Thickness and Orientation, Structural Analysis` in the **Label** text field.

ROOT

Add a second study to solve the acoustics problem only.

ADD STUDY

1 On 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 **Preset Studies> Eigenfrequency**.

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

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

STUDY 2

In the **Settings** window for **Study**, type `Acoustics Analysis` in the **Label** text field.

ACOUSTICS ANALYSIS

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Acoustics Analysis** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `20`.

5 In the **Search for eigenfrequencies around** text field, type `500`.

6 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Shell (shell)** interface.

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

RESULTS

Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure, Acoustics Analysis` in the **Label** text field.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level, Acoustics Analysis` in the **Label** text field.

Acoustic Pressure, Isosurfaces (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Isosurfaces (acpr)**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure, Acoustics Analysis, Isosurfaces` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **672.55**.

4 On the **Acoustic Pressure, Acoustics Analysis, Isosurfaces** toolbar, click **Plot**.

Isosurface 1

1 In the **Model Builder** window, expand the **Results>Acoustic Pressure, Acoustics Analysis, Isosurfaces** node, then click **Isosurface 1**.

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

3 From the **Color table** list, choose **WaveLight**.

4 On the **Acoustic Pressure, Acoustics Analysis, Isosurfaces** toolbar, click **Plot**.

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

Acoustic Pressure, Acoustics Analysis, Isosurfaces

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Acoustics Analysis, Isosurfaces**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **1345.1**.

4 On the **Acoustic Pressure, Acoustics Analysis, Isosurfaces** toolbar, click **Plot**.

ROOT

Finally, add a third study for the coupled problem.

ADD STUDY

1 On 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 **Preset Studies> Eigenfrequency**.

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

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

STUDY 3

In the **Settings** window for **Study**, type `Coupled Analysis` in the **Label** text field.

COUPLED ANALYSIS

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Coupled Analysis** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** check box.

4 In the associated text field, type `20`.

5 In the **Search for eigenfrequencies around** text field, type `500`.

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

RESULTS

Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure, Coupled Analysis` in the **Label** text field.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level, Coupled Analysis` in the **Label** text field.

Acoustic Pressure, Isosurfaces (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Isosurfaces (acpr)**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure, Coupled Analysis, Isosurfaces` in the **Label** text field.

Mode Shape (shell)

1 In the **Model Builder** window, under **Results** click **Mode Shape (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shape, Coupled Analysis` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **637.11**.

4 On the **Mode Shape, Coupled Analysis** toolbar, click **Plot**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Mode Shape, Coupled Analysis** 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 **Model>Component 1>Shell> Displacement>Displacement field>w - Displacement field, Z component**.

Slice 1

1 In the **Model Builder** window, under **Results** right-click **Mode Shape, Coupled Analysis** and choose **Slice**.

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

3 In the **Planes** text field, type `1`.

4 On the **Mode Shape, Coupled Analysis** toolbar, click **Plot**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

Plot the pressure gradient to display the connection to the disk shape.

6 Locate the **Expression** section. In the **Expression** text field, type `pz`.

7 In the **Unit** field, type `Pa/m`.

8 On the **Mode Shape, Coupled Analysis** toolbar, click **Plot**.

Undeformed Geometry (shell)

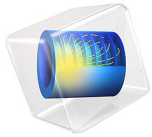
1 In the **Model Builder** window, under **Results** click **Undeformed Geometry (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Undeformed Geometry, Coupled Analysis` in the **Label** text field.

Thickness and Orientation (shell)

1 In the **Model Builder** window, under **Results** click **Thickness and Orientation (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Thickness and Orientation, Coupled Analysis` in the **Label** text field.



Axisymmetric Condenser Microphone

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of a condenser microphone with a simple axisymmetric geometry. The model aims to give a precise description of the physical working principles of such a microphone.

The condenser microphone is considered to be the microphone with highest quality when performing precise acoustical measurements and with high-fidelity reproduction properties when performing sound recordings; see Ref. 2. This electro-mechanical acoustic transducer works by transforming the mechanical deformation of a thin membrane (diaphragm) into an AC voltage signal.

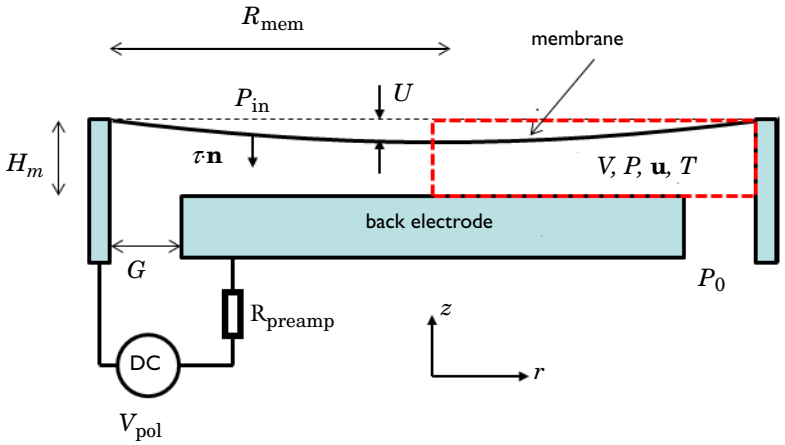


Figure 1: Sketch of the condenser microphone system including variables and coordinate system. The red box indicates the modeled region.

Models for describing condenser microphones have classically been of the equivalent network type; see Ref. 2. Analytical models exist for simpler geometries, but there are also highly advanced analytical models for more complex geometries; see for example Ref. 1 or Ref. 4. In the present detailed finite-element model the thermoviscous acoustic, electric, and structural problem is solved fully coupled using the frequency-domain linear perturbation solver. This includes the DC charging (prepolarization) and deformation of the membrane which makes out the zeroth order linearization point. The system analyzed here is very close to that of the Axisymmetric Condenser Microphone with Electrical Lumping model except that it does not use any lumping approximations. In addition a small external circuit model is added to model the pre-amplifier. In many cases, lumping

of the electrical part is a good approximation, especially for simple geometries such as this one, where the back electrode is flat and has no perforations. During the initial design steps, lumped models are an important tool. In more complex geometries, solving the full set of equations is necessary to get the correct response.

The current model represents a true multiphysics problem that involves several physics interfaces: Thermoviscous Acoustics, Electrostatics, Electrical Circuit, Moving Mesh, and a Membrane model.

Note: This application requires the AC/DC Module and the Structural Mechanics Module in addition to the Acoustics Module.

Model Definition

The geometry and model definitions are shown in Figure 1. In many microphones there is a back-volume below the electrode. In this model a simplified approach is taken and a pressure release condition is applied with p_0 = 0 Pa. The membrane is deformed due to the electrostatic forces from charging the capacitor and because of the pressure variation from the external incoming uniform acoustic signal p_{in}. The chosen dimensions of the microphone are typical generic dimensions. Dimensions and parameters are given in Table 1.

TABLE 1: MICROPHONE DIMENSIONS AND PARAMETERS

SYMBOL	SIZE & UNIT	DESCRIPTION
H_{m}	18 µm	Air gap thickness
R_{mem}	2 mm	Membrane radius
G	54 µm	Slit gap width
T_{m0}	3150 N/m	Membrane static tension
E_{m}	221 GPa	Membrane elastic modulus
t_{m}	7 µm	Membrane thickness
ρ_{m}	8300 kg/m^3	Membrane density
V_{pol}	100 V	Target polarization voltage
R_{preamp}	1 GΩ	Pre-amplifier output impedance
ν_{m}	0.4	Poisson's ratio for the membrane

The membrane is backed by a thin air gap of thickness H_m and a back electrode. Because the gap is so small, the inclusion of thermal and viscous losses in the acoustic model is essential, thus using the thermoviscous acoustics interface. The membrane and back electrode make up a capacitor that is polarized by an external DC voltage source through the pre-amplifier resistance R_{preamp}. This will give rise to a surface charge Q_m. The air gap acts as a damping layer for the membrane vibrations. As the gap between the membrane and the back electrode varies, a voltage change is induced. This AC voltage is the output of the microphone.

The sensitivity of the condenser microphone L, is measured in the unit dB (relative to $1\ \mathrm{V/Pa}$). It is defined as the ratio of the open circuit output voltage V_{out} to the input pressure p_{in} and is given by

$$L = 20\log\left[\left|\frac{V_{out}}{p_{in}}\right| / \left(1\frac{\mathrm{V}}{\mathrm{Pa}}\right)\right]$$

In the present model, the membrane (or diaphragm) is modeled using the dedicated Membrane interface from the Structural Mechanics Module. The membrane is subject to a surface load that is the sum of the external incident pressure p_{in}, the internal pressure $p = p(\mathbf{r})$ (given by the thermoviscous acoustics model), and the electrostatic force given by the Maxwell surface stress, $\mathbf{n} \cdot \tau$. The incident pressure is here assumed to be uniform over the microphone membrane, which is only an approximation. At the highest frequencies modeled, the acoustic wavelength becomes comparable with the membrane radius.

Results and Discussion

This model involves a detailed description of the physical effects in a simple condenser microphone. The sensitivity of the microphone L, is directly determined from the model (voltage on the terminal divided by the incident pressure) and is shown in Figure 2 below. Notice the slight roll-off at the low frequencies, this is due to the interaction with the pre-amplifier circuit. By increasing the output impedance R_{preamt} this effect will disappear, if the value is decreased the roll-off will be more significant.

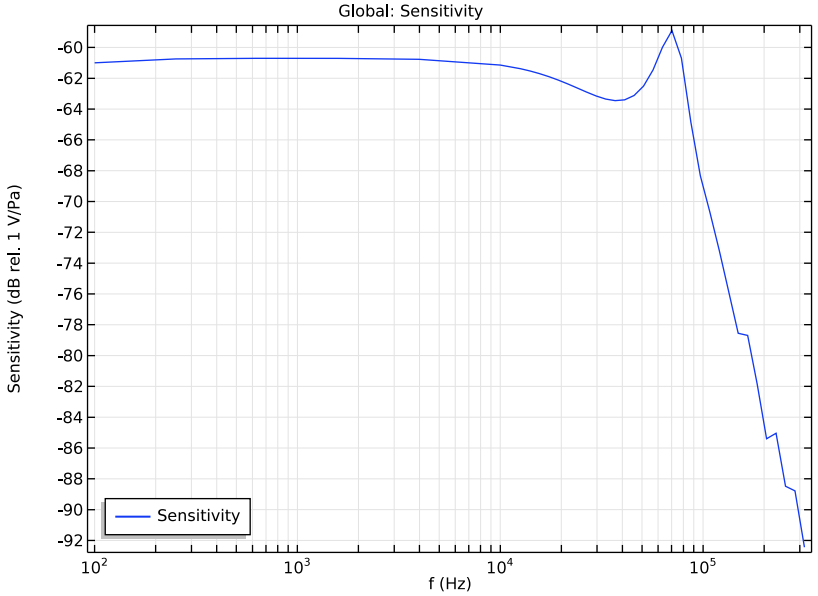


Figure 2: Sensitivity curve of the microphone measured in dB relative to 1 V/Pa.

The shape of the deformed membrane is plotted for $f = 0.3$ GHz as a 3D surface in Figure 3, using a revolution 2D data set. At this frequency it is clear to see how higher order modes in the membrane are the cause for the poor sensitivity.

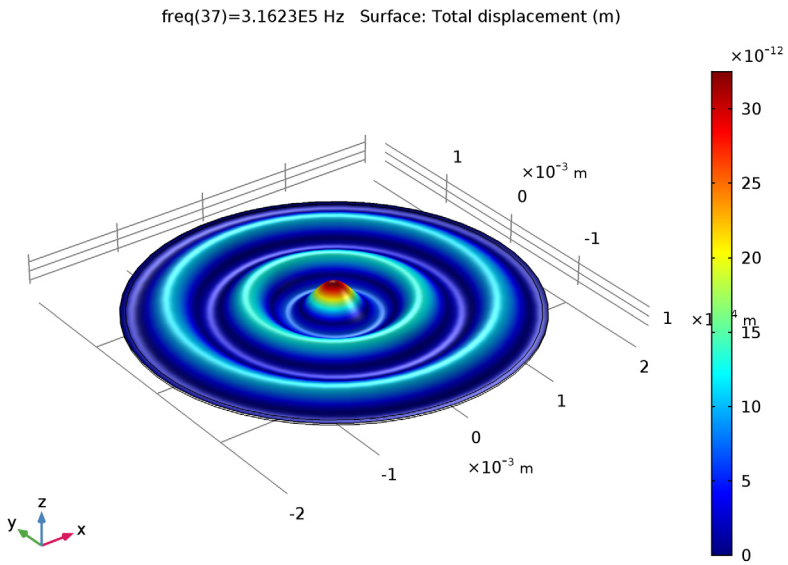


Figure 3: 3D representation of the harmonic membrane deformation at 0.3 GHz.

The principles described in this model can be extended to 3D models with more complex geometries. Because the full set of equations is solved, such a model includes all physical effects to a high degree of detail. It can be used to optimize the performance of microphones, to make virtual tests of new geometries, or to investigate the relative importance of different parameters.

Notes About the COMSOL Implementation

COUPLED STATIC AND FREQUENCY-DOMAIN MODEL USING THE LINEAR PERTURBATION SOLVER

The current model solves a fully coupled problem using the linear perturbation frequency-domain solver. A stationary model determines the linearization point and the full system of equations is then linearized and solved around this point to determine the harmonic small-signal response.

The first step is to determine the linearization point for the problem which requires solving a static model that determines the shape of the membrane after the polarization voltage is

applied. The current model does not consider the transient charging of the condenser. The first step solves the electrostatic model (using the Electrostatics interface of the AC/DC Module) coupled to the membrane model. The acoustic model is automatically deactivated as it is, per construction, a small perturbation and thus has no contribution to the static solver step. To determine the correct capacitance (and electric fields) a Moving Mesh interface is needed. The capacitance is a geometry-dependent quantity.

The second step is to solve the linear perturbation frequency-domain model that describes the time-harmonic small-signal deformation of the membrane and the interaction with the fluid (described by a Thermoviscous Acoustics, Frequency Domain interface) within the microphone.

References

1. T. Lavergne, S. Durand, M. Bruneau, N. Joly, and D. Rodrigues, "Dynamic behavior of the circular membrane of an electrostatic microphone: Effect of holes in the backing electrode," *J. Acoust. Soc. Am.*, vol. 128, p. 3459, 2010.

2. W. Marshall Leach, Jr., *Introduction to Electroacoustics and Audio Amplifier Design*, 3rd ed., Kendall/Hunt Publishing Company, 2003.

3. P.M. Morse and K. Uno Ignard, *Theoretical Acoustics*, Princeton University Press, 1968.

4. V.C. Henriquez, *Numerical Transducer Modelling*, PhD Thesis, DTU, November 2001.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/condenser_microphone`

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 **AC/DC>Electrostatics (es)**.

3 Click **Add**.

This is an appropriate choice because it is a good assumption that the electric processes in this model are quasistatic.

4 In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics> Thermoviscous Acoustics, Frequency Domain (ta)**.

7 Click **Add**.

8 In the **Select Physics** tree, select **Structural Mechanics>Membrane (mbrn)**.

9 Click **Add**.

10 In the **Displacement field** text field, type `um`.

11 In the **Displacement field components** table, enter the following settings:

um
vm
wm

The displacement field of the membrane is (um,vm,wm) while the velocity field in the fluid is (u,v,w).

12 In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.

13 Click **Add**.

14 Click **Study**.

15 In the **Select Study** tree, select **Empty Study**.

16 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `condenser_microphone_parameters.txt`.

   These are the parameters specifying the geometry and the physical properties of the microphone and membrane.

GEOMETRY 1

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Rmem`.
4. In the **Height** text field, type `Hm`.
5. Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `G`.
4. In the **Height** text field, type `Hm`.
5. Locate the **Position** section. In the **r** text field, type `Rmem-G`.
6. Click **Build All Objects**.

DEFINITIONS

Create a selection corresponding to the membrane for use when adding features to the membrane edge.

Explicit 1

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, locate the **Input Entities** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 3 and 6 only.

5 In the **Label** text field, type `Membrane`.

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	muB0	Pa·s	Basic

Having set up the materials, proceed to setting up and defining the physics. Begin with the electric problem: Electrostatics and Circuit.

ELECTROSTATICS (ES)

Terminal 1

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

2 Select Boundary 2 only.

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

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

Ground 1

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

2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

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

Now, set up the small external electrical circuit that is depicted in Figure 1.

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

External I Vs. U 1

1 On the **Electrical Circuit** toolbar, click **External I Vs. U**.

2 In the **Settings** window for **External I Vs. U**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **External Device** section. From the *V* list, choose **Terminal voltage (es/term1)**.

Resistor R1

1 On the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the R text field, type `Rpreamp`.

Voltage Source V1

1 On the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the V_{src} text field, type `Vpol`.

Next, set up the acoustics before turning to the membrane model and the moving mesh interface.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

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

Pressure (Adiabatic) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure (Adiabatic)**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Pressure (Adiabatic)**, locate the **Pressure** section.

4 In the p_{bnd} text field, type `p0`.

MULTIPHYSICS

Thermoviscous Acoustic-Structure Boundary 1 (tsb1)

1 On the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Thermoviscous Acoustic-Structure Boundary**.

2 In the **Settings** window for **Thermoviscous Acoustic-Structure Boundary**, locate the **Boundary Selection** section.

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

The last step couples the membrane to the thermoviscous acoustic domain. The multiphysics coupling feature sets the acoustic velocity equal to i\omega times the deformation of the membrane (this is the time derivative in the frequency domain). This condition is a bidirectional constraint, meaning that a reaction force is added to the membrane equation, ensuring a two-way coupling.

Now, set up the membrane model, constrain it at the outer perimeter where it is fixed and add the forces that act on it. They are the electrostatic forces given by the Maxwell stress tensor `(es.dnTer,es.dnTez)` and the incident pressure field `pin`. Use the `linper()` operator to tell COMSOL that the incident pressure is only a harmonic frequency dependent quantity (not a static load).

MEMBRANE (MBRN)

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

2 In the **Settings** window for **Membrane**, locate the **Boundary Selection** section.

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

4 Locate the **Thickness** section. In the d text field, type `tm`.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Membrane (mbrn)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the E list, choose **User defined**. In the associated text field, type `Em`.

4 From the ν list, choose **User defined**. In the associated text field, type `num`.

5 From the ρ list, choose **User defined**. In the associated text field, type `rhom`.

Initial Stress and Strain 1

1 On the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the N_0 table, enter the following settings:

TmO	0
0	TmO

Fixed Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 6 only.

Face Load 1

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

2 In the **Settings** window for **Face Load**, locate the **Boundary Selection** section.

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

4 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

es.dnTer	r
es.dnTez	z

Face Load 2

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

2 In the **Settings** window for **Face Load**, locate the **Boundary Selection** section.

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

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the *p* text field, type `linper(pin)`.

MOVING MESH (ALE)

In the **Model Builder** window, under **Component 1 (comp1)** click **Moving Mesh (ale)**.

Free Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.

2 Click in the **Graphics** window and then press Ctrl+A to select both domains.

Prescribed Mesh Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

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

4 Locate the **Prescribed Mesh Displacement** section. In the d_r text field, type `um`.

5 In the d_z text field, type `wm`.

Prescribed Mesh Displacement 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.

4 Clear the **Prescribed z displacement** check box.

The above Moving Mesh interface ensures that the computational mesh deforms according to the membrane deformation (um,wm).

MESH 1

The final step before setting up the solvers is to build the computational mesh. To mesh the narrow region between the membrane and the backplate use a structured Mapped mesh.

Mapped 1

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

2 In the **Settings** window for **Mapped**, click to expand the **Advanced settings** section.

3 Locate the **Advanced Settings** section. Select the **Adjust evenly distributed edge mesh** check box.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Boundaries 2 and 3 only.

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

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

5 In the **Number of elements** text field, type `40`.

6 In the **Element ratio** text field, type `5`.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 1 and 7 only.

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

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

5 In the **Number of elements** text field, type 10.

6 In the **Element ratio** text field, type 2.

7 Select the **Symmetric distribution** check box.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 5 and 6 only.

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

4 In the **Number of elements** text field, type 4.

5 Click **Build All**.

STUDY 1

Some manual work is required to set up the solvers. First, a stationary solver determines the linearization point for the frequency-domain perturbation solver. The first stationary problem is highly non-linear. Also select the PARDISO solver for this as one of the ODE variables is zero in this step. The PARDISO solver handles this better than the default MUMPS. Second, set up the frequency-domain perturbation solver and choose the PARDISO solver also here; the PARDISO solver is better suited for solving the acoustic problem than the default MUMPS solver.

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

2 Clear the **Generate default plots** check box.

Step 1: Stationary

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

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

3 Select the **Include geometric nonlinearity** check box.

Enabling geometric nonlinearity is necessary for the Membrane interface to work.

Notice the small orange warning sign next to Thermoviscous Acoustics and the Multiphysics coupling indicating that these are not solved in the stationary study step (as expected).

Step 2: Frequency Domain Perturbation

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain Perturbation**.

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

3 Select the **Include geometric nonlinearity** check box.

4 In the **Frequencies** text field, type `10^{range(2,2/5,4)} 10^{range(4.1,1.4/30, 5.5)}`.

This gives a frequency range of 100 Hz - 316 kHz. The reason for including such high frequencies is to be able to observe the fall-off in sensitivity.

Generate the default solver sequence. This is a good starting point when doing modifications to a solver. First modify the stationary solver then the perturbation solver.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 2** node, then click **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1>Fully Coupled 1**.

5 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

6 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Automatic highly nonlinear (Newton)**.

You have selected the **Automatic highly nonlinear (Newton)** solver for this fully coupled nonlinear problem. Finally, select PARDISO as the direct solver for both study steps.

7 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 1** click **Direct 1**.

8 In the **Settings** window for **Direct**, locate the **General** section.

9 From the **Solver** list, choose **PARDISO**.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 2** click **Direct 1**.

11 In the **Settings** window for **Direct**, locate the **General** section.

12 From the **Solver** list, choose **PARDISO**.

13 Click **Compute**.

RESULTS

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
es.V0_1	V	Terminal voltage
es.Q0_1	C	Terminal charge

4 From the **Expression evaluated for** list, choose **Static solution**.

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The static polarization voltage across the membrane should equal 100.0 V (as defined). The resulting static membrane charge is 5.9e-10 C.

RESULTS

2D Plot Group 1

1 On the **Results** toolbar, click **2D Plot Group**.

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

Surface 1

1 Right-click **Velocity** 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>Thermoviscous Acoustics, Frequency Domain>Acceleration and velocity>ta.v_inst - Instantaneous local velocity**.

3 On the **Velocity** toolbar, click **Plot**.

To get a better view of the long slender geometry, disable the Preserve aspect ratio option.

DEFINITIONS

Axis

1 In the **Model Builder** window, expand the **View 1** node, then click **Axis**.

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

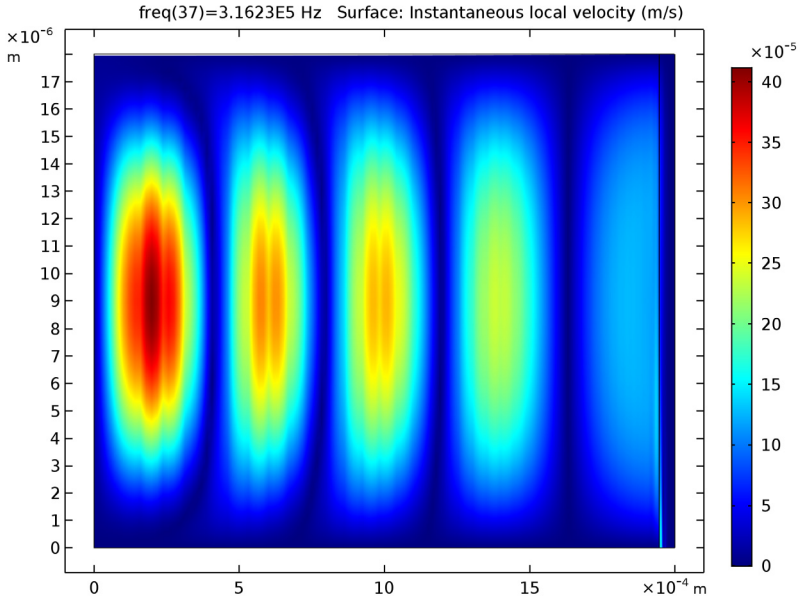
3 From the **View scale** list, choose **Automatic**.

4 Click **Update**.

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

RESULTS

Velocity



In the same way you can also plot the acoustic pressure (replacing `ta.v_inst` with `ta.p_t`) or acoustic temperature variations (replacing `ta.v_inst` with `ta.T_t`).

2D Plot Group 2

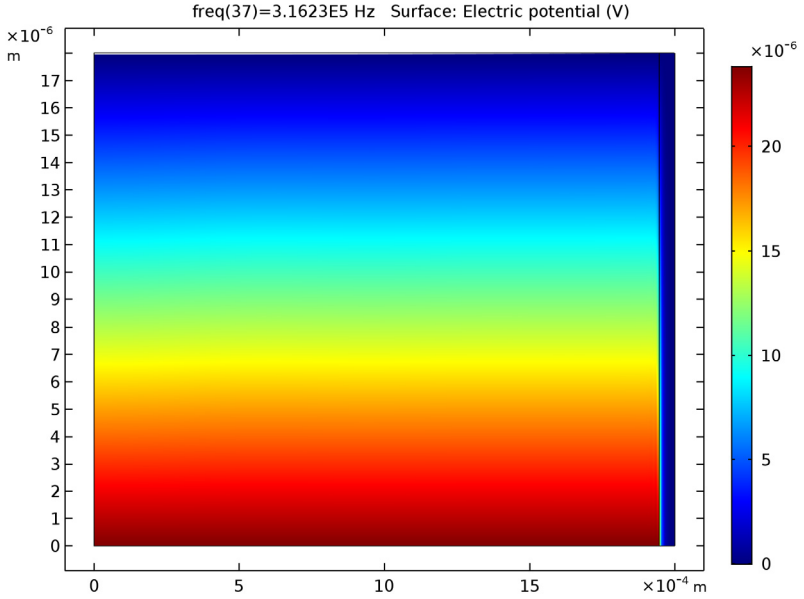
1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

Potential

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

2 On the **Potential** toolbar, click **Plot**.



1D Plot Group 3

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

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

Global 1

1 Right-click **Sensitivity** 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
`20*log10(abs(es.V0_1/pin))`		`Sensitivity`

4 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Sensitivity

1 In the **Model Builder** window, under **Results** click **Sensitivity**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

3 From the **Position** list, choose **Lower left**.

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

5 In the associated text field, type `f (Hz)`.

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

7 In the associated text field, type `Sensitivity (dB rel. 1 V/Pa)`.

8 On the **Sensitivity** toolbar, click **Plot**.

The microphone sensitivity curve should look like Figure 2.

Now, plot the membrane deformation in a 1D plot, both the static and the harmonic components, as function of the radial coordinate.

1D Plot Group 4

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

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

Line Graph 1

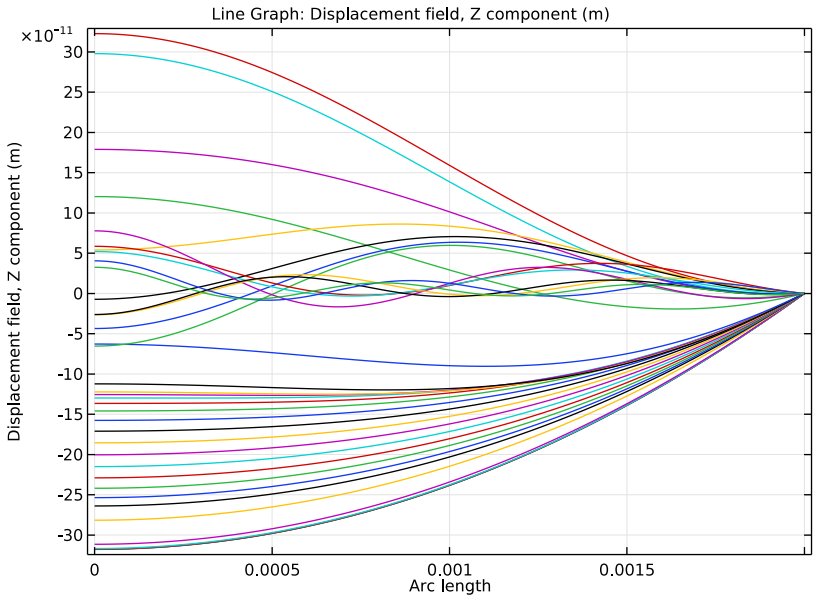
1 Right-click **Membrane deformation** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `wm`.

5 On the **Membrane deformation** toolbar, click **Plot**.



Membrane deformation 1

1 In the **Model Builder** window, under **Results** right-click **Membrane deformation** and choose **Duplicate**.

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

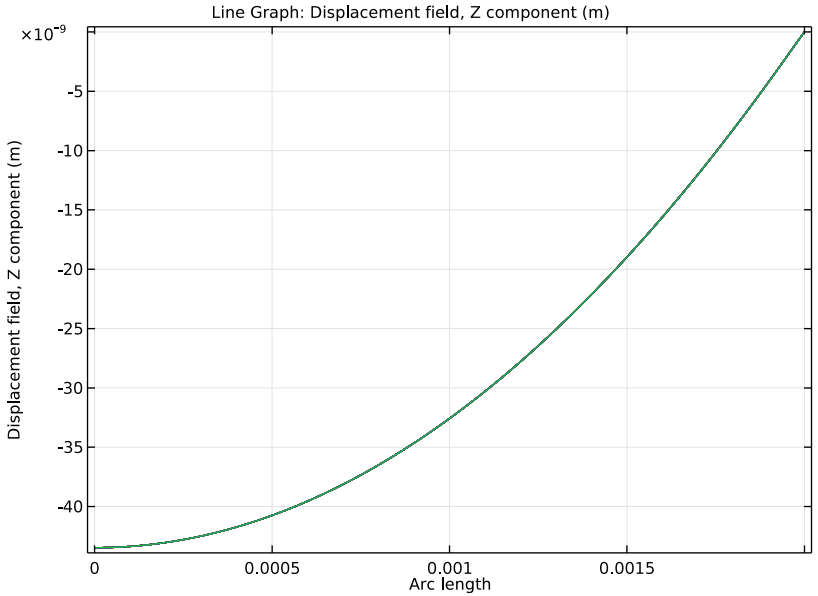
Line Graph 1

1 In the **Model Builder** window, expand the **Results>Static membrane deformation** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 From the **Expression evaluated for** list, choose **Static solution**.

4 On the **Static membrane deformation** toolbar, click **Plot**.



Finally, plot the membrane deformation on a 3D revolved geometry using a 2D revolution data set, to reproduce Figure 3.

3D Plot Group 6

1 On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

2 On the **Results** toolbar, click **3D Plot Group**.

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

Surface 1

1 Right-click **3D Membrane deformation** and choose **Surface**.

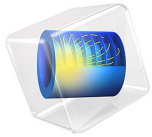
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Membrane>Displacement> mbrn.disp - Total displacement**.

Deformation 1

1 Right-click **Results>3D Membrane deformation>Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Membrane> Displacement>um,...,wm - Displacement field (material and geometry frames)**.

3 On the **3D Membrane deformation** toolbar, click **Plot**.



Axisymmetric Condenser Microphone with Electrical Lumping

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of a condenser microphone with a simple axisymmetric geometry. The model aims to give a precise description of the physical working principles of such a microphone, using lumped descriptions for some of the physical phenomena. Lumping certain parts of a model can give additional insight to physical phenomena using a simplified description. A fully coupled approach (without lumping internal physics) can be seen in the Axisymmetric Condenser Microphone model, also located in the Application Library of the Acoustics Module.

The condenser microphone is considered to be the microphone with highest quality when performing precise acoustical measurements and with high-fidelity reproduction properties when performing sound recordings, see Ref. 2. This electromechanical acoustic transducer works by transforming the mechanical deformation of a thin membrane (diaphragm) into an AC voltage signal.

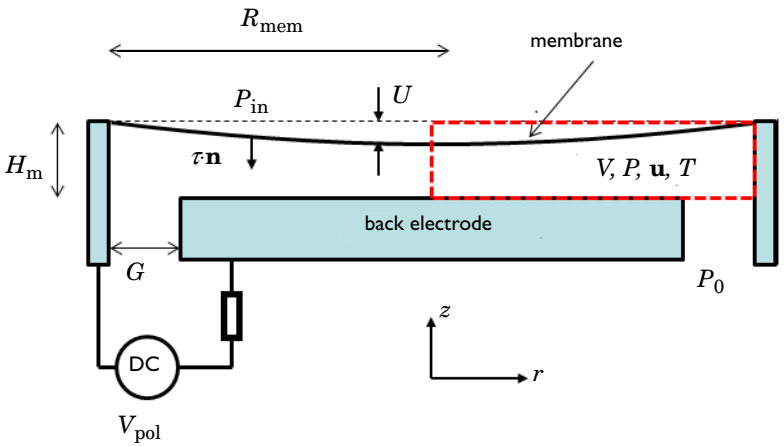


Figure 1: Sketch of the condenser microphone system including variables and coordinate system. The red box indicates the modeled region.

Models for describing condenser microphones have classically been of the equivalent network type (see Ref. 2). Analytical models exist for simpler geometries, but there are also highly advanced analytical models for more complex geometries; see for example Ref. 1. In the present detailed finite-element (FE) model, you model the microphone including a static (quiescent) analysis of the DC charging (prepolarization) and deformation of the membrane. You then perform time-harmonic (small-signal) finite element (FE) analysis of

the dynamics of the membrane coupled to thermoviscous acoustics. The small-signal electric model for the system is solved as a lumped (electric equivalent) model coupled to the FE model. The model is a true multiphysics problem that involves several physics interfaces: Thermoviscous Acoustics, Electrostatics, Moving Mesh, two user-defined PDE interfaces, a Global ODEs and DAEs interface, as well as an Electrical Circuit model.

Note: This application requires the AC/DC Module.

Model Definition

The geometry and model definitions are shown in Figure 1. The membrane is deformed due to the electrostatic forces from charging the capacitor and because of the pressure variation from the external incoming uniform acoustic signal p_{in}. The chosen dimensions of the microphone are typical generic dimensions. Dimensions and parameters are given in Table 1.

TABLE 1: MICROPHONE DIMENSIONS AND PARAMETERS

PARAMETER	VALUE	DESCRIPTION
H_{m}	18 μm	Air gap thickness
R_{mem}	2 mm	Membrane radius
G	54 μm	Slit gap width
T_{m}	3150 N/m	Membrane tension
t_{m}	7 μm	Membrane thickness
ρ_{m}	8300 kg/m^3	Membrane density
V_{pol}	100 V	Polarization voltage

The membrane is backed by a thin air gap of thickness H_{m} and a back electrode. Because the gap is so small, the inclusion of thermal and viscous losses in the acoustic model is essential, thus using the thermoviscous acoustics interface. The membrane and back electrode makes up a capacitor that is polarized by an external DC voltage source. The air gap acts as a damping layer for the membrane vibrations. As the gap between the membrane and the back electrode varies a voltage change is induced and is coupled to the capacitor (the quiescent DC capacitance of the system C_0) and an external very large resistive load R_L.

The electric circuit for this coupling is shown in Figure 2.

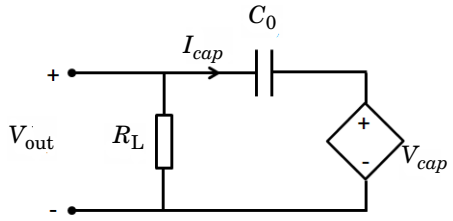


Figure 2: Analogous circuit for the electrical part of the condenser microphone.

The sensitivity L of the condenser microphone is measured in the unit dB (relative to 1 V/Pa). It is defined as the ratio of the open circuit output voltage V_{out} to the input pressure p_{in} and is given by

$$L = 20\log\left(\frac{V_{\text{out}}}{p_{\text{in}}}\right)$$

Next, consider a small-signal (linearized) analysis of the electric part of the condenser microphone. The total voltage is the sum of the quiescent polarization voltage V_{pol} and the small-signal output voltage V_{out}. The total charge on the condenser is the sum of the quiescent charge Q and the small-signal charge q. The distance between the back electrode and the membrane is the sum of the initial distance H_{m}, the quiescent average deformation $U_{0,\text{av}}$ and the small-signal deformation U_{av}.

The capacitance C of a parallel plate capacitor, with a fixed air gap distance h and area A is given by

$$C = \frac{\varepsilon_0\varepsilon_r A}{h}$$

If the air gap h is varied with an average deformation U_{av} around the initial static gap distance, $H_{\text{m}} + U_{0,\text{av}}$, the expression becomes

$$C = \frac{\varepsilon_0\varepsilon_r A}{H_{\text{m}} + U_{0,\,\text{av}} + U_{\text{av}}} = \frac{\varepsilon_0\varepsilon_r A}{H_{\text{m}} + U_{0,\,\text{av}}}\left(1 - \frac{U_{\text{av}}}{H_{\text{m}} + U_{0,\,\text{av}}}\right) + O(U_{\text{av}}^2) \qquad (1)$$

where the expression has been expanded to first order. Again, the distance $U_{0,\text{av}}$ stems from the initial deformation from equilibrium due to the electrostatic forces of the prepolarization of the condenser. A first approximation for the average deformation in axisymmetric coordinates is given as

$$U_{\text{av}} = \frac{1}{A}\int U dA = \frac{2}{R_{\text{mem}}^2}\int_0^{R_m - G} U(r) r dr \tag{2}$$

The capacitance is by its definition (see Ref. 4) given by

$$Q = CV \tag{3}$$

where Q is the charge on and V the potential across the capacitor. Inserting Equation 1 into Equation 3 and retaining only first-order terms yields

$$V_{\text{pol}} + V_{\text{out}}(t) = \frac{Q + q(t)}{C} \cong \frac{Q}{C_0}\left(1 + \frac{U_{\text{av}}(t)}{H_{\text{m}} + U_{0,\text{av}}}\right) + \frac{q(t)}{C_0} \tag{4}$$

Differentiating Equation 4 with respect to time, switching to frequency domain, and using $V_{\text{pol}} = Q/C_0$ yields

$$V_{\text{out}} = \frac{I_{\text{cap}}}{i\omega C_0} + \frac{u_{\text{av}} V_{\text{pol}}}{i\omega(H_{\text{m}} + U_{0,\text{av}})} \tag{5}$$

where I_{cap} is the induced current through the capacitor, the second term on the right is the electro-mechanic coupling V_{cap}, and the average membrane velocity is

$$u_{\text{av}} = \frac{1}{A}\int u_{\text{m}} dA = \frac{2}{R_{\text{mem}}^2}\int_0^{R_m - G} u_{\text{m}}(r) r dr \tag{6}$$

where u_{m} is the axial membrane velocity. The circuit model equivalent to equation Equation 5 is shown in Figure 2.The governing equation of the membrane is described in the next section, Membrane Model. The final element necessary to couple the lumped small parameter model of the electric model to the mechanical FE model is the back coupling via the small parameter electrostatic force f_{es}. The force is approximated by the spatial derivative (z direction) of the electric energy stored in a parallel plate condenser; the small parameter component is (see Ref. 2)

$$f_{\text{es}} = -\frac{V_{\text{pol}} I_{\text{cap}}}{i\omega(H_{\text{m}} + U_{0,\text{av}})}$$

The force is applied evenly over the membrane as a surface normal stress $f_{\text{es}}/(2\pi R_{\text{mem}}^2)$.

Note that the integrals in Equation 2 and Equation 6 are over the area of the membrane that is backed by the back electrode plus a possible small correction for edge effects. This

is especially important if the back electrode has holes. This is the case in many commercials condenser microphones where the holes are placed in order to produce a special sensitivity characteristic of the microphone. In this model the back electrode is flat and uniform.

MEMBRANE MODEL

The displacement U of a thin axisymmetric membrane of thickness $\mathrm{t_m}$, under constant tension $\mathrm{T_m}$, and with a density ρ_m is governed by the following equation

$$\mathrm{T_m}\frac{\partial}{\partial r}\left(r\frac{\partial U}{\partial r}\right) - \rho_\mathrm{ms} r\frac{\partial^2 U}{\partial t^2} - rF_s = 0 \tag{7}$$

where r is the radial coordinate, t is time, $\rho_\mathrm{ms} = \rho_\mathrm{m}/\mathrm{t_m}$ is the surface density, and F_s is the sum of surface forces; see for example Ref. 3. In the present model, the surface force is the sum of the external incident pressure p_in (it is assumed to be uniform over the microphone membrane), the internal pressure $p = p(\mathbf{r})$ (given by the thermoviscous acoustics model), and the electrostatic force which is the sum of the quiescent Maxwell surface stress $\mathbf{n}\cdot\tau$ (given by the electrostatic model) and the small-signal force f_es. The variations of the deformation U is assumed to be small and harmonic on top of the static contribution U_0 from the DC polarization, such that

$$\begin{aligned} U(\mathbf{r},t) &= U_0(\mathbf{r}) + U(\mathbf{r})e^{i\omega t} \\ p_\mathrm{in}(\mathbf{r},t) &= p_\mathrm{in}e^{i\omega t} \\ p(\mathbf{r},t) &= p(\mathbf{r})e^{i\omega t} \\ F_\mathrm{es}(\mathbf{r},t) &= \mathbf{n}\cdot\tau + f_\mathrm{es}/(2\pi R_\mathrm{m}^2)\cdot e^{i\omega t} \end{aligned}$$

Using these expressions, Equation 7 is reformulated into a static and a time-harmonic equation as

$$\begin{aligned} \mathrm{T_m}\frac{\partial}{\partial r}\left(r\frac{\partial U_0}{\partial r}\right) - r(\mathbf{n}\cdot\tau) &= 0 \\ \mathrm{T_m}\frac{\partial}{\partial r}\left(r\frac{\partial U}{\partial r}\right) + \rho_\mathrm{ms} r\omega^2 U - r(f_\mathrm{es}/(2\pi R_\mathrm{mem}^2) + p_\mathrm{in} - p) &= 0 \end{aligned} \tag{8}$$

The latter equation may be rewritten in terms of the axial velocity, $u_\mathrm{m} = i\omega U$, of the membrane in the form of a Helmholtz equation:

$$\mathrm{T_m}\frac{\partial}{\partial r}\left(r\frac{\partial u_\mathrm{m}}{\partial r}\right)+\mathrm{T_m}k_\mathrm{m}^2 r u_\mathrm{m}-i\omega r(p_\mathrm{in}-p)\;=\;0$$
$$k_\mathrm{m}^2\;=\;\frac{\omega^2\rho_\mathrm{ms}}{\mathrm{T_m}} \tag{9}$$

Here k_m is the membrane wave number. In this model you disregard the change in tension due to the movement of the membrane, which is a nonlinear effect that is small compared to the tension T_m.

Results and Discussion

This model involves a detailed description of the physical effects at play in a simple condenser microphone. The lumping of the small-signal analysis of the electrical part is a good approximation for this simple geometry, where the back electrode is flat and has no perforations. The sensitivity, L, of the microphone is directly determined from the model (voltage across the load resistance divided by the incident pressure) and is shown in Figure 3.

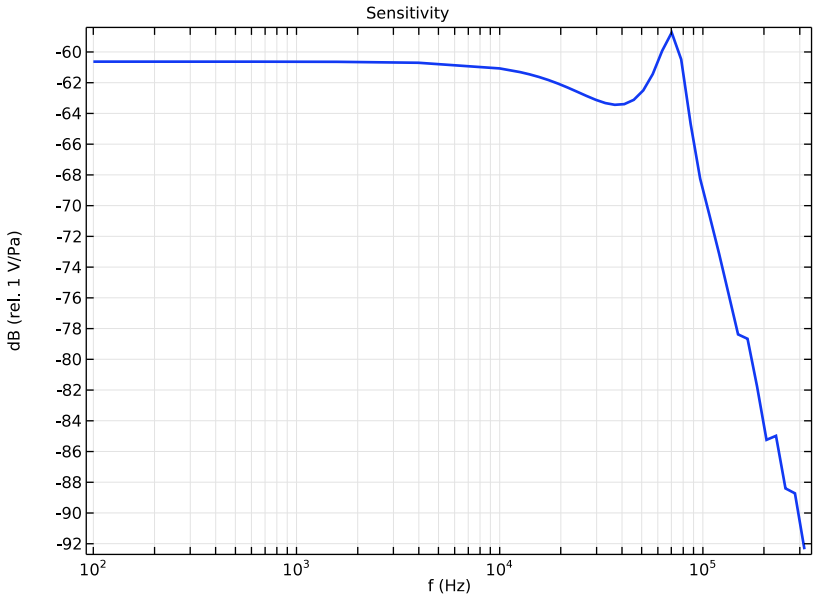


Figure 3: Sensitivity curve of the microphone measured in dB relative to 1 V/Pa.

For the case of the simple geometry used in this model, an analytical solution exists for the dynamics of the undamped membrane; see Ref. 5. The axial displacement is given by

$$U_{\mathrm{th}}(r) = \frac{p_{\mathrm{in}}}{T_{\mathrm{m}} k_{\mathrm{m}}^2}\left(1 - \frac{J_0(k_{\mathrm{m}} r)}{J_0(k_{\mathrm{m}} R_{\mathrm{mem}})}\right)$$

where k_{m} is the wave number defined in Equation 9. The analytical approximation is compared to the model results in Figure 4, which shows the average deformation versus frequency. The results agree well below the resonance frequency of the system. The average behavior above the first resonance (in between resonances) is also well captured by the approximative theoretical model. In the real system the damping introduced by the thermal and viscous losses in the air gap is important, especially at the resonances. This is also seen from the figure, where the resonance of the full (real) system is damped and shifted in frequency. The comparison of the two models is used as an extra indicator for the correctness of the FE model.

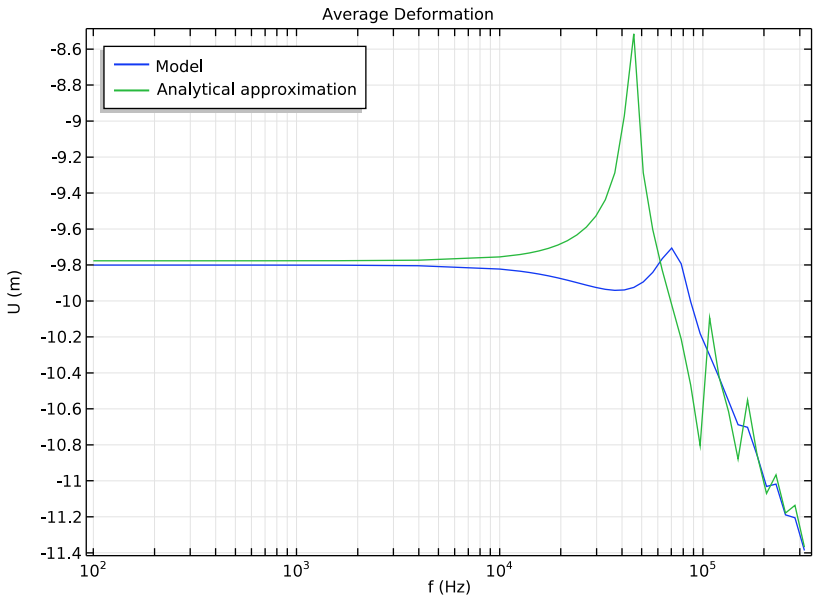


Figure 4: Comparison of the average membrane deformation given by the COMSOL model and by the theoretical approximation for the undamped membrane.

The shape of the deformed membrane is plotted for $f = 0.3$ GHz as a 3D surface in Figure 5 using a revolution 2D data set. At this frequency it is clear to see how higher order modes in the membrane are the cause of the poor sensitivity.

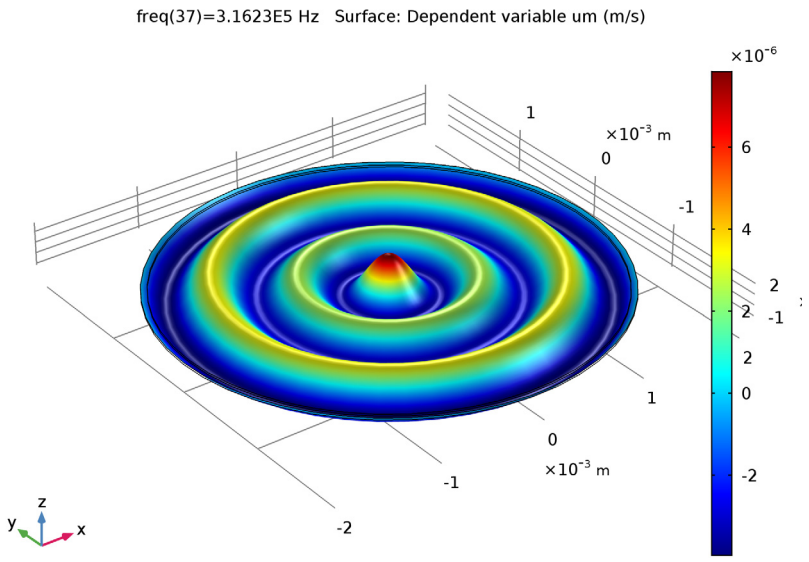


Figure 5: 3D representation of the harmonic membrane deformation at 0.3 GHz.

The principles described in this model may be extended to 3D models with more complex geometries where, for example, the back electrode is perforated or has a convex shape. Such a model may be used to optimize the performance of microphones, to make virtual tests of new geometries, or to investigate the relative importance of different parameters.

Notes About the COMSOL Implementation

COUPLED STATIC AND FREQUENCY DOMAIN MODEL

The implementation couples a static model that determines the quiescent shape of the membrane after the polarization voltage is applied to the time harmonic small-signal response. The current model does not consider the transient charging of the condenser. The first step requires solving an electrostatic model (AC/DC interface) coupled to the static membrane model (user defined PDE model). In order for the model to determine the correct quiescent capacitance C_0, a Moving Mesh interface is necessary, as the capacitance is a geometric dependent quantity.

The second step is to solve the frequency domain model that describes the time harmonic small-signal deformation of the membrane (user defined PDE) and the interaction with the fluid (thermoviscous acoustics model) within the microphone. This solution is superposed to the static solution. The small-signal electric components of the microphone and the sensitivity is determined by a small lumped AC/DC circuit model.

STATIONARY SURFACE CHARGE AND CAPACITANCE

In the variables list the static capacitance variable `C0` is defined as:

```
es.term1.int(es.nD*2*pi*r)/es.V0_1
```

This term is equal to the stationary terminal charge Q_0 divided bu the stationary voltage V_0. The stationary charge needs to be calculated as a surface integral of the D field and not evaluated using the variable `es.Q0_1`. This variable evaluates to 0 in the frequency domain step due to the current way reaction forces are transferred between study steps.

WEAK FORM OF THE MEMBRANE EQUATION

The membrane equations are implemented in COMSOL using the general weak form formulation of a partial differential equation. This is an integral form of the strong formulation of Equation 8. The equation is multiplied by a test function and integration by parts is performed (using Green's first identity). The resulting equation for the static deformation becomes

$$\int_{\partial\Omega} r\left(F_s\Phi - T_\mathrm{m}\frac{\partial U_0}{\partial r}\frac{\partial\Phi}{\partial r}\right)dl + \sum_N rT_\mathrm{m}\frac{\partial U_0}{\partial r}\Phi$$

where Φ is the test function of the displacement U_0. In COMSOL Multiphysics this is formulated as

```
r*((es.dnTez)*test(U0)-Tm*dtang(U0,r)*test(dtang(U0,r)))
```

The resulting weak form equation for the time-harmonic variation of the membrane reads

$$\int_{\partial\Omega} r\left((T_\mathrm{m}k_\mathrm{m}^2u_\mathrm{m} - i\omega(p_\mathrm{in}-p))\Phi - T_\mathrm{m}\frac{\partial u_\mathrm{m}}{\partial r}\frac{\partial\Phi}{\partial r}\right)dl + \sum_N rT_\mathrm{m}\frac{\partial u_\mathrm{m}}{\partial r}\Phi$$

which is formulated as

```
r*((Tm*kmsq*um-ta.iomega*(pin-p))*test(um)-
    Tm*dtang(um,r)*test(dtang(um,r)))
```

The sum over N in both expressions are the boundary (point) contribution at the center $r = 0$ and the edge $r = R_\mathrm{mem}$ of the membrane.

References

1. T. Lavergne, S. Durand, M. Bruneau, N. Joly, and D. Rodrigues, "Dynamic behavior of the circular membrane of an electrostatic microphone: Effect of holes in the backing electrode," *J. Acoust. Soc. Am.*, vol. 128, p. 3459, 2010.

2. W. Marshall Leach, Jr., *Introduction to Electroacoustics and Audio Amplifier Design*, 3rd ed., Kendall/Hunt Publishing Company, 2003.

3. P.M. Morse and K. Uno Ignard, *Theoretical Acoustics*, Princeton University Press, 1968.

4. D.J. Griffiths, *Introduction to Electrodynamics*, 3rd ed., Pearson Education, 2008.

5. V.C. Henriquez, *Numerical Transducer Modelling*, PhD Thesis, DTU, November 2001.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/condenser_microphone_lumped`

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>Thermoviscous Acoustics> Thermoviscous Acoustics, Frequency Domain (ta)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **AC/DC>Electrostatics (es)**.
5. Click **Add**.
6. In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.
7. Click **Add**.
8. In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh (ale)**.

9 Click **Add**.

10 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>Lower Dimensions> Weak Form Boundary PDE (wb)**.

11 Click **Add** twice.

12 Click **Done**.

GLOBAL DEFINITIONS

A set of parameters defining the material properties and the geometry are available in a text file that can be loaded.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `condenser_microphone_lumped_parameters.txt`.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `Rmem`.

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

5 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `G`.

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

5 Locate the **Position** section. In the **r** text field, type `Rmem-G`.

6 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

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

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 Click **Load from File**.

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

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_be` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

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

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 3 and 6 only.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Thermoviscous Acoustics Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)** click **Thermoviscous Acoustics Model 1**.

2 In the **Settings** window for **Thermoviscous Acoustics Model**, locate the **Thermoviscous Acoustics Model** section.

3 From the $\rho_0(p_{0,T0})$ list, choose **Ideal gas**.

Pressure (Adiabatic) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure (Adiabatic)**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Pressure (Adiabatic)**, locate the **Pressure** section.

4 In the p_{bnd} text field, type `p0`.

Velocity 1

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

2 In the **Settings** window for **Velocity**, locate the **Boundary Selection** section.

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

4 Locate the **Velocity** section. Select the **Prescribed in r direction** check box.

5 Select the **Prescribed in z direction** check box.

6 In the u_{0z} text field, type `um`.

7 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

8 Click to expand the **Constraint settings** section. Locate the **Constraint Settings** section. From the **Apply reaction terms on** list, choose **Individual dependent variables**.

This setting is necessary because an influence of the acoustics on the membrane (the reaction force) is introduced via the pressure term, p, in the equation for the membrane that you will set up shortly. If this option were not selected, the pressure (or a scaled version thereof) would act twice on the membrane.

The thermal boundary condition for the membrane should be isothermal. This conditions is automatically selected in the wall condition for the walls.

Isothermal 1

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

2 In the **Settings** window for **Isothermal**, locate the **Boundary Selection** section.

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

ELECTROSTATICS (ES)

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

Terminal 1

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

2 Select Boundary 2 only.

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

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

5 In the V_0 text field, type `Vpol`.

Ground 1

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

2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

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

You have now set up the electrical circuit representing the small-signal part of the electrical model in accordance with the circuit depicted in Figure 2.

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

Capacitor C1

1 On the **Electrical Circuit** toolbar, click **Capacitor**.

2 In the **Settings** window for **Capacitor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the *C* text field, type `C0`.

Resistor R1

1 On the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the *R* text field, type `RL`.

Voltage Source V1

1 On the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. From the **Source type** list, choose **AC-source**.

5 In the V_{src} text field, type `Vcap`.

MOVING MESH (ALE)

In the **Model Builder** window, under **Component 1 (comp1)** click **Moving Mesh (ale)**.

Free Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.

2 Click in the **Graphics** window and then press Ctrl+A to select both domains.

Prescribed Mesh Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Boundary Selection** section.

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

4 Locate the **Prescribed Mesh Displacement** section. In the d_z text field, type `U0`.

Prescribed Mesh Displacement 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.

4 Clear the **Prescribed z displacement** check box.

WEAK FORM BOUNDARY PDE (WB)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Weak Form Boundary PDE (wb)**.

2 In the **Settings** window for **Weak Form Boundary PDE**, locate the **Boundary Selection** section.

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

4 Locate the **Units** section. In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1

5 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Field name** text field, type `U0`.

6 In the **Dependent variables** table, enter the following settings:

U0

7 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

8 In the **Physical Quantity** dialog box, In the associated text field, type `id:displacement`.

9 Click **Filter**.

10 In the tree, select **Solid>Displacement field (m)**.

11 Click **OK**.

Weak Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>** **Weak Form Boundary PDE (wb)** click **Weak Form PDE 1**.

2 In the **Settings** window for **Weak Form PDE**, locate the **Weak Expressions** section.

3 In the weak text field, type `r*((es.dnTez)*test(U0)-Tm*dtang(U0,r)*test(dtang(U0,r)))`.

You can ignore the unexpected unit warning.

4 In the **Model Builder** window, click **Weak Form Boundary PDE (wb)**.

Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Constraint**.

2 Select Point 6 only.

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

4 In the R text field, type `-U0`.

WEAK FORM BOUNDARY PDE 2 (WB2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Weak Form Boundary PDE 2 (wb2)**.

2 In the **Settings** window for **Weak Form Boundary PDE**, locate the **Boundary Selection** section.

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

4 Locate the **Units** section. In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	1

5 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Field name** text field, type `um`.

6 In the **Dependent variables** table, enter the following settings:

`um`

7 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

8 In the **Physical Quantity** dialog box, In the associated text field, type `id:velocity`.

9 Click **Filter**.

10 In the tree, select **Transport>Velocity field (m/s)**.

11 Click **OK**.

Weak Form PDE 1

1 In the **Model Builder** window, expand the **Weak Form Boundary PDE 2 (wb2)** node, then click **Weak Form PDE 1**.

2 In the **Settings** window for **Weak Form PDE**, locate the **Weak Expressions** section.

3 In the weak text field, type `r*((Tm*kmsq*um-ta.iomega*(Fes+pin-p))*test(um)-Tm*dtang(um,r)*test(dtang(um,r)))`.

Ignore the inconsistent unit warning.

4 In the **Model Builder** window, click **Weak Form Boundary PDE 2 (wb2)**.

Constraint 1

1 On the **Physics** toolbar, click **Points** and choose **Constraint**.

2 Select Point 6 only.

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

4 In the R text field, type `-um`.

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, click to expand the **Material properties** section.

2 Locate the **Material Properties** section. From the **Material type** list, choose **Nonsolid**.

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

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	muB0	Pa·s	Basic
Specific gas constant	Rs	Rs0	J/(kg·K)	Ideal gas

MESH 1

Mapped 1

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

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

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

4 Select Domain 1 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Boundaries 2 and 3 only.

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

4 In the **Number of elements** text field, type 30.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 1 and 4 only.

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

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

5 In the **Number of elements** text field, type 10.

6 In the **Element ratio** text field, type 2.

7 Select the **Symmetric distribution** check box.

Distribution 1

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

2 Right-click **Mapped 2** and choose **Distribution**.

3 Select Boundaries 5 and 6 only.

Boundary Layer Properties

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

2 Select Boundary 4 only.

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

4 In the **Number of boundary layers** text field, type `6`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `2e-6`.

7 Click **Build Selected**.

The mesh is built such that it resolves the acoustic boundary layer at the maximal frequency of 320 kHz. At this frequency the viscous boundary layer is about 4 μm thick, corresponding to roughly 1/5 of the air-gap thickness.

ADD STUDY

1 On 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 **Empty Study**.

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

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

STUDY 1

Step 1: Stationary

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the following interfaces:

Physics interface
Thermoviscous Acoustics, Frequency Domain
Electrical Circuit
Weak Form Boundary PDE 2

Step 2: Frequency Domain

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain**.

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

3 In the **Frequencies** text field, type `10^{range(2,2/5,4)} 10^{range(4.1,1.4/30, 5.5)}`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the following interfaces:

Physics interface
Electrostatics
Weak Form Boundary PDE
Moving Mesh

5 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Study 1, Stationary**.

8 In the **Model Builder** window, click **Study 1**.

9 In the **Settings** window for **Study**, locate the **Study Settings** section.

10 Clear the **Generate default plots** check box.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 2** node, then click **Direct 1**.

5 In the **Settings** window for **Direct**, locate the **General** section.

6 From the **Solver** list, choose **PARDISO**.

The PARDISO solver is the most efficient one for solving problems involving thermoviscous acoustics.

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

RESULTS

2D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

Surface 1

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

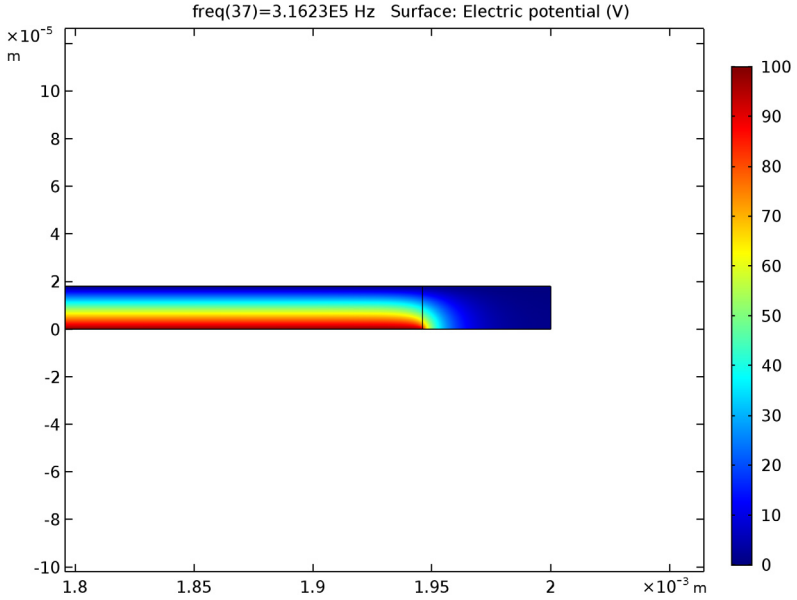
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `V`.

4 On the **Potential** toolbar, click **Plot**.

Potential

You can examine the plot in greater detail by zooming around the edge of the model using the **Zoom Box** tool.



2D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

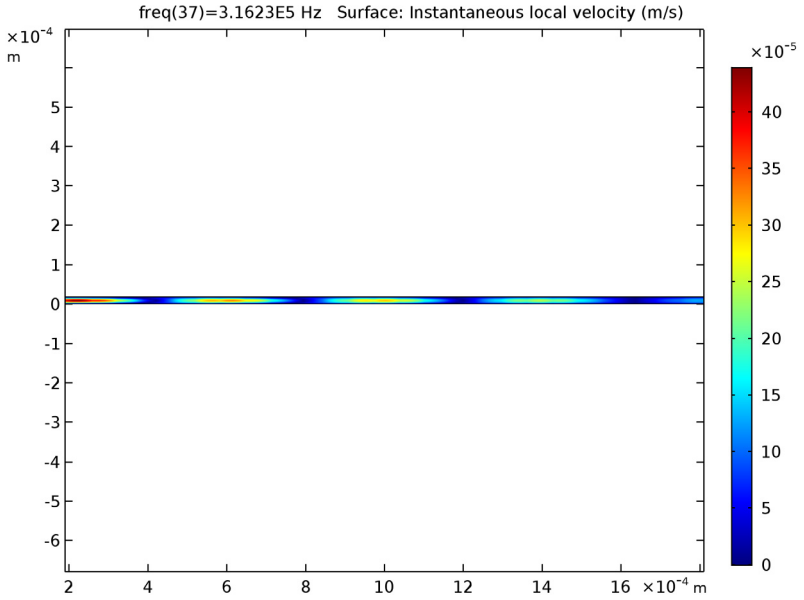
Surface 1

1 Right-click **Velocity** 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>Thermoviscous Acoustics, Frequency Domain>Acceleration and velocity>ta.v_inst - Instantaneous local velocity**.

3 On the **Velocity** toolbar, click **Plot**.

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



1D Plot Group 3

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

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

Line Graph 1

1 Right-click **Membrane Deformation** and choose **Line Graph**.

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

3 In the **Settings** window for **Line Graph**, locate the **Selection** section.

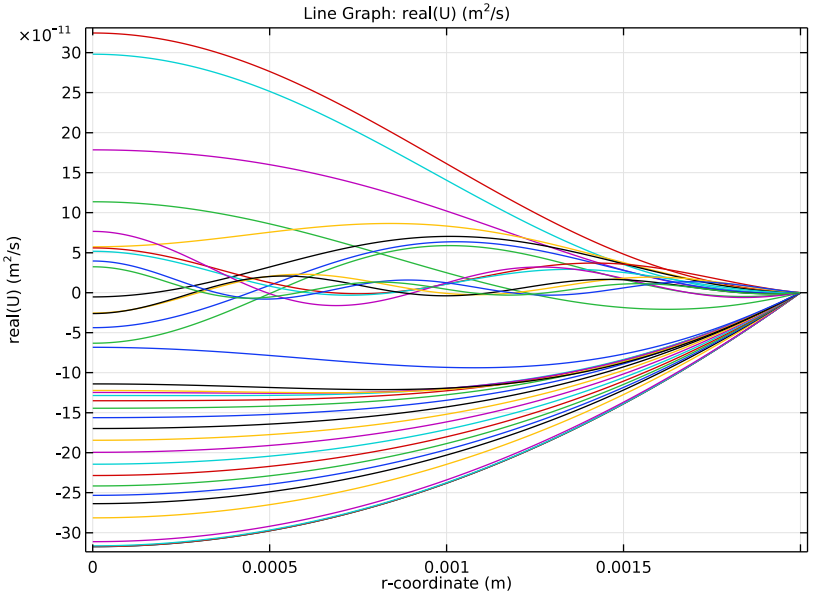
4 From the **Selection** list, choose **Membrane**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(U)`.

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

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

8 On the **Membrane Deformation** toolbar, click **Plot**.



1D Plot Group 4

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

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

Line Graph 1

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

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

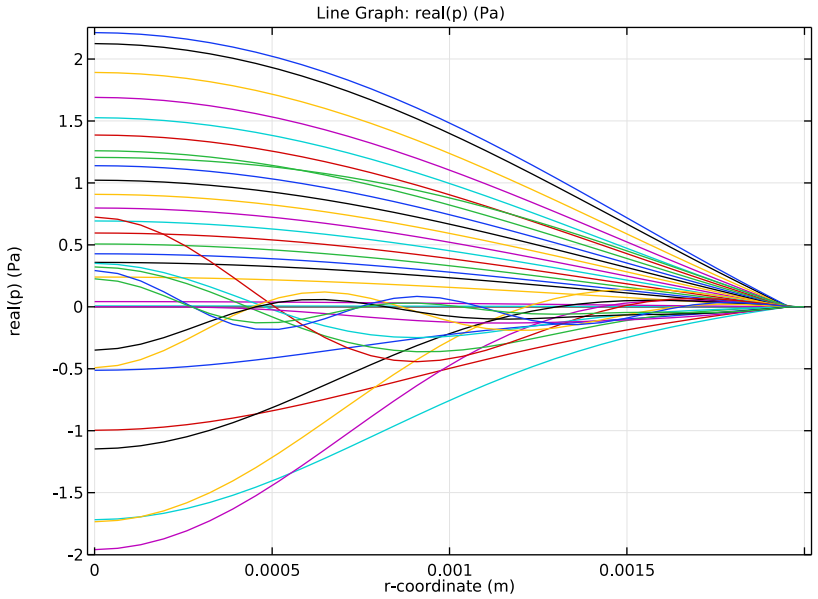
3 From the **Selection** list, choose **Membrane**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(p)`.

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

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

7 On the **Pressure Under Membrane** toolbar, click **Plot**.



1D Plot Group 5

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

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

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

4 In the **Title** text area, type `Average Membrane Velocity`.

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

6 In the associated text field, type `f (Hz)`.

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

8 In the associated text field, type `um_av (m/s)`.

9 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Average Membrane Velocity** and choose **Global**.

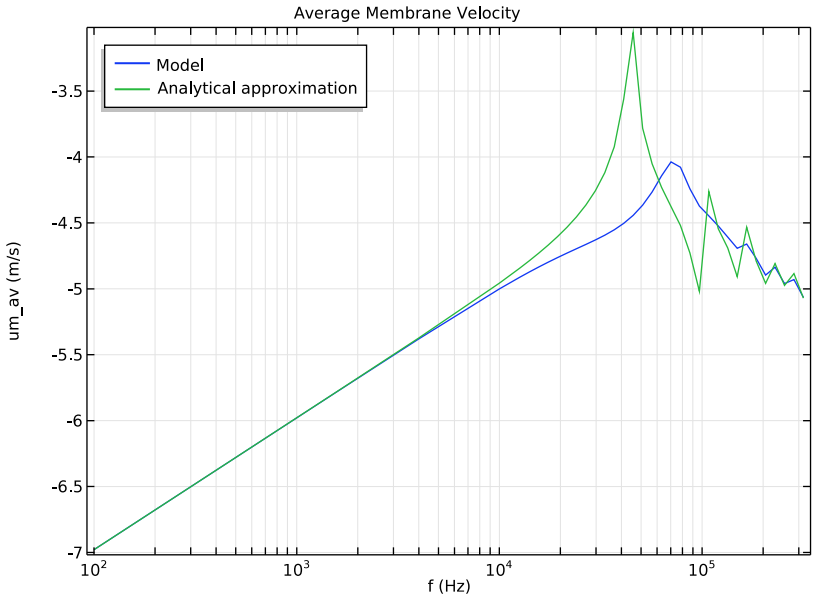
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
log10(abs(um_av))		Model
log10(abs(uth_av))		Analytical approximation

4 On the **Average Membrane Velocity** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.



1D Plot Group 6

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

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

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

4 In the **Title** text area, type `Average Deformation`.

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

6 In the associated text field, type `f (Hz)`.

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

8 In the associated text field, type `U (m)`.

9 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Average Membrane Deformation** 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
`log10(abs(U_av))`		`Model`
`log10(abs(Uth_av))`		`Analytical approximation`

4 On the **Average Membrane Deformation** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This reproduces the average membrane velocity plot depicted in Figure 4.

1D Plot Group 7

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

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

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

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

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

6 In the associated text field, type `f (Hz)`.

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

8 In the associated text field, type `dB (rel. 1 V/Pa)`.

Global 1

1 Right-click **Sensitivity** 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
`20*log10(abs(cir.R1_v/pin))`		`Sensitivity`

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

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

6 On the **Sensitivity** toolbar, click **Plot**.

7 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This should reproduce the sensitivity curve depicted in Figure 3.

Finally, create a 2D revolution data set to plot the membrane deformation on the revolved 3D geometry.

3D Plot Group 8

1 On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

2 On the **Results** toolbar, click **3D Plot Group**.

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

Surface 1

1 Right-click **3D Membrane Deformation** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `um`.

Deformation 1

1 Right-click **Results>3D Membrane Deformation>Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

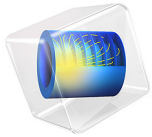
3 In the **r component** text field, type `0`.

4 In the **phi component** text field, type `0`.

5 In the **z component** text field, type `um`.

6 On the **3D Membrane Deformation** toolbar, click **Plot**.

The plot should reproduce Figure 5.



Cylindrical Subwoofer

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is an introduction to modeling with the Acoustics Module. The step-by-step instructions take you through the process of setting up a model of the sound field created by a cylindrical down-firing subwoofer placed on a floor. The important result of the simulation is the contribution to the system's mechanical impedance induced by the coupling between the speaker membrane and the air inside and outside the speaker enclosure. A polar plot of the far field is also presented to demonstrate the subwoofer's uniform intensity distribution.

Model Definition

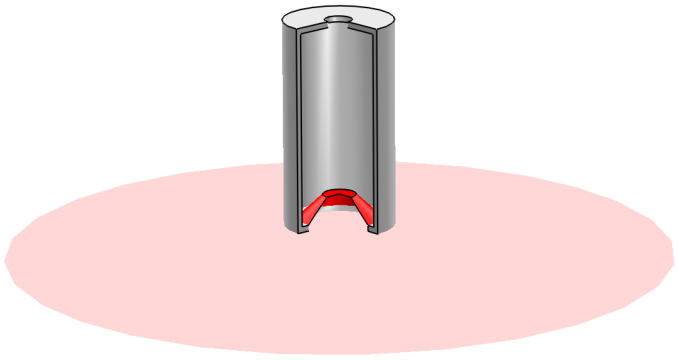


Figure 1: The geometry of the subwoofer with a slice removed to show its interior. In the illustration the floor is cut off but in the model it is assumed to extend to infinity.

Figure 1 shows the geometry simulated in this model. A down-firing cylindrical subwoofer is placed on a floor bounding an infinite half-space. The subwoofer enclosure has a height of 0.7 m and a diameter of 0.4 m. The bottom of the subwoofer is dominated by a 12-inch (0.3 m diameter) membrane. The top has a vent with a diameter of 0.1 m.

The membrane vibrates harmonically with the acceleration $a = a_0 e^{i\omega t}$ where $a_0 = 10$ m/s^2 and $\omega = 2\pi f$ is the angular frequency (rad/s). The frequencies considered are $f = 50$–70 Hz, which are in the high end of the frequency range that subwoofers are typically used for. It is assumed that the walls of the enclosure are perfectly rigid. The acoustic medium is air.

The feet that the speaker would need to stand on are judged to have a negligible effect on the sound field. With this assumption, all geometric features and physics have a rotational symmetry with respect to the axis of the speaker. This makes it natural to set the model up in a 2D axisymmetric physics interface.

Results and Discussion

By assuming the membrane is inflexible, you can write Newton's second law for the voice coil and membrane as

$$ma = F_{\text{ext}} + F_{\text{emf}} + F_m + F_a$$

where the forces acting on the system have been split into four components. The applied force external force F_{ext}, an electromagnetic force opposing the movement of the coil F_{emf}, a mechanical part F_m independent of the acoustic environment (elastic behavior of the suspensions, spider and surround in a speaker), and a term F_a containing only the effects of the fluid loading on the membrane. The external force is typically proportional to the applied voltage, while the latter three contributions are proportional to the velocity and directed to oppose the movement. Introducing corresponding mechanical impedances (with proper normal directions assumed), and using the time-harmonic assumption, it holds that

$$F_{\text{ext}} = v(i\omega m + Z_{\text{emf}} + Z_m + Z_a)$$

If you know the three impedances as functions of frequency, you can predict the system's electrical and mechanical behavior. The electrical force can be measured or simulated with a blocked voice coil, while Z_m is directly related to the mass of the voice coil and membrane as well as to the stiffness of the baffle keeping the voice coil centered. These contributions are therefore independent of the environment in which the speaker operates.

The acoustic contribution to the mechanical impedance, on the other hand, depends on the enclosure geometry and on reflecting surfaces in the immediate surroundings. In Figure 2 you can study the real and imaginary parts of Z_a as functions of frequency in the simulated range. The real part represents radiation and acts as a resistance in the electromechanical system while the imaginary part represents a reactance.

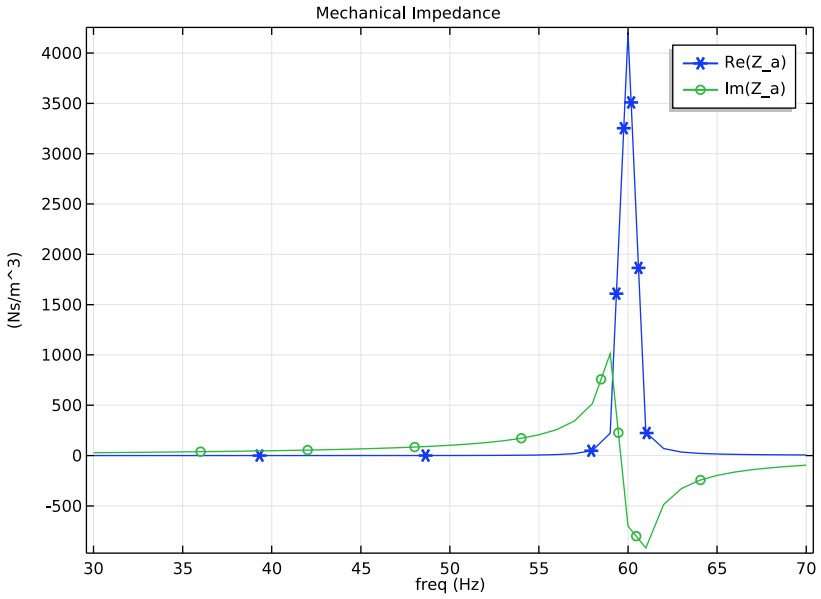


Figure 2: The resistive (real) and reactive (imaginary) parts of the contribution of the air to the speaker's mechanical impedance.

Note that the reactance switches sign from positive to negative at approximately 60 Hz. This means that at frequencies below this visible resonance the surrounding air acts as an added mass on the membrane, while at higher frequencies its action is spring-like.

Figure 3 displays the far-field sound pressure level in the *rz*-plane as a polar plot evaluated at 2 m from the speaker.

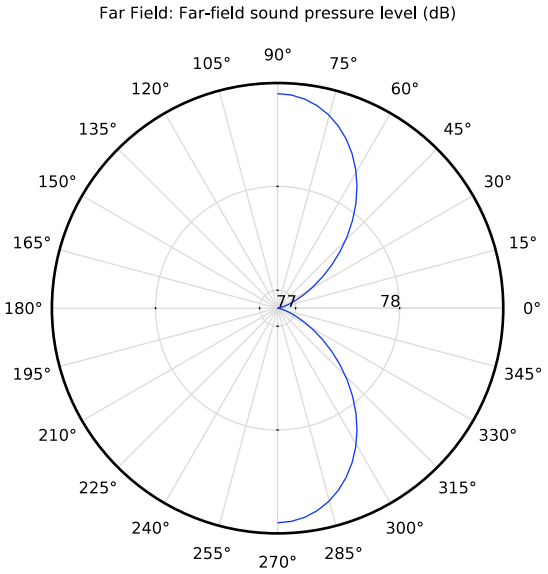


Figure 3: Far-field sound level in dB at a distance of 2 m from the floor below the subwoofer, 0^0 corresponds to the vertical z-direction. Note the 77 dB offset. The large wavelength compared to the size of the speaker system makes the response rather uniform.

Notes About the COMSOL Implementation

This model is set up in 2D axisymmetry using the Pressure Acoustics, Frequency Domain interface, which is described in the *Acoustics Module User's Guide*. The modeled physical domain is a hemisphere with a radius of 1 m. To minimize the effect of nonphysical reflections at the exterior boundary of this domain, an absorbing perfectly matched layer (PML) is added outside of it. For more information about PMLs see the *COMSOL Multiphysics Reference Manual*.

The membrane is modeled with zero thickness. The Interior Normal Acceleration boundary condition allows the pressure field to be discontinuous across the membrane.

Application Library path: `Acoustics_Module/Tutorials/cylindrical_subwoofer`

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>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6. Click **Done**.

GEOMETRY 1

Create the geometry.

To simplify this step, you can insert a prepared geometry sequence. On the **Geometry** toolbar, click **Insert Sequence**. Browse to the application's Application Library folder and double-click the file `cylindrical_subwoofer.mph`. Click **Build All** on the **Geometry** toolbar. Then, continue with the instruction after the geometry plot below.

Otherwise, proceed with the following instructions to create the geometry from scratch:

Begin by drawing the air domain and the surrounding PML.

Circle 1 (c1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `1.5`.
4. In the **Sector angle** text field, type `90`.
5. Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`0.5`

6. Click **Build Selected**.

Next, add the walls of the subwoofer.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.02`.

4 In the **Height** text field, type `0.66`.

5 Locate the **Position** section. In the **r** text field, type `0.18`.

6 In the **z** text field, type `0.12`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.05`.

4 In the **Height** text field, type `0.02`.

5 Locate the **Position** section. In the **r** text field, type `0.15`.

6 In the **z** text field, type `0.1`.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.15`.

4 In the **Height** text field, type `0.02`.

5 Locate the **Position** section. In the **r** text field, type `0.05`.

6 In the **z** text field, type `0.78`.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c1** onlyto add it to the **Objects to add** list.

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

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the objects **r1**, **r2**, and **r3** only.

6 Click **Build All Objects**.

Finally, create the membrane.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Linear**.

5 Find the **Control points** subsection. In row **1**, set **z** to `0.2`.

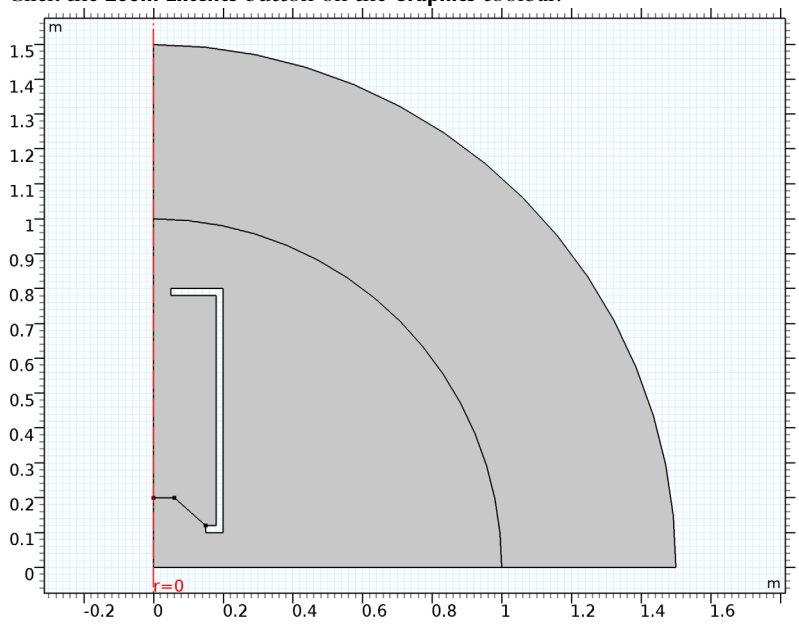
6 In row **2**, set **r** to `0.06` and **z** to `0.2`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **r** to `0.15` and **z** to `0.12`.

9 Click **Build All Objects**.

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



GLOBAL DEFINITIONS

This model uses a parameter to represent the membrane's peak acceleration and variables for the total force on the membrane and the mechanical impedance of the membrane-air system. An integral operator takes part in the force computation.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
a0	10[m/s^2]	10 m/s²	Peak acceleration

DEFINITIONS

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `mem_int` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 4 and 9 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
F_a	mem_int(2*pi*r*nz*(down(p)-up(p)))	N	Net force on membrane
Z_a	-F_a/(a0/acpr.iomega)	N·s/m	Mechanical impedance

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 2 only.

Note that the default PML center coordinate (0,0) applies because the spherical mantle that it constitutes is centered in the origin. The evanescent part of the wave leaving from the vent is expected to decay with a characteristic length much shorter than the wavelength.

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Interior Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Normal Acceleration**.

2 Select Boundaries 4 and 9 only.

3 In the **Settings** window for **Interior Normal Acceleration**, locate the **Interior Normal Acceleration** section.

4 Specify the $\mathbf{a}_0$ vector as

0	r
a0	z

The membrane boundaries are considered interior because they have a pressure distribution on both sides. The Sound Hard Boundary (Wall) condition applies per default to all the remaining exterior boundaries. This includes the surface of the enclosure.

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 Select Boundary 18 only.

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

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

5 From the **Type of integral** list, choose **Full integral**.

The hard floor is excluded from the integral by noting that the radiation pattern is exactly the same as if replacing the floor with a mirror image of the subwoofer that is instead included in the integral. The far-field integral is set to use the full integral version, this means that far field can be evaluated in any point outside the computational. This option will give the full information about both amplitude and phase, not only in the mathematical far-field.

MESH 1

In acoustic models, the wavelength must be resolved by the mesh. In addition, small features in the geometry can induce high local pressure gradients that must be resolved properly to obtain a consistent global solution. In this particular model, the wavelength is greater than 5 m, so the focus is on resolving the geometry. Use a mapped mesh in the PML domain.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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

3 From the **Predefined** list, choose **Finer**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.

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

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

4 Select Domain 1 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>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 **Boundary**.

4 Select Boundaries 4, 6, and 9 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click the **Custom** button.

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

8 In the associated text field, type `1e-3`.

9 Select the **Maximum element growth rate** check box.

10 Click **Build All**.

MESH 1

In the **Model Builder** window, expand the **Study 1** node.

Mapped 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Mapped**.

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

STUDY 1

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `range(30,1,70)`.

This gives you frequencies from 30 to 70 Hz with a pitch of 1 Hz.

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

RESULTS

Acoustic Pressure (acpr)

The first default plot shows the pressure field for the last parameter value, 70 Hz. The postprocessing algorithms automatically try to smooth discontinuous fields. You can see this effect if you zoom in on the membrane, where the field really should be discontinuous. To turn off the smoothing, follow these steps:

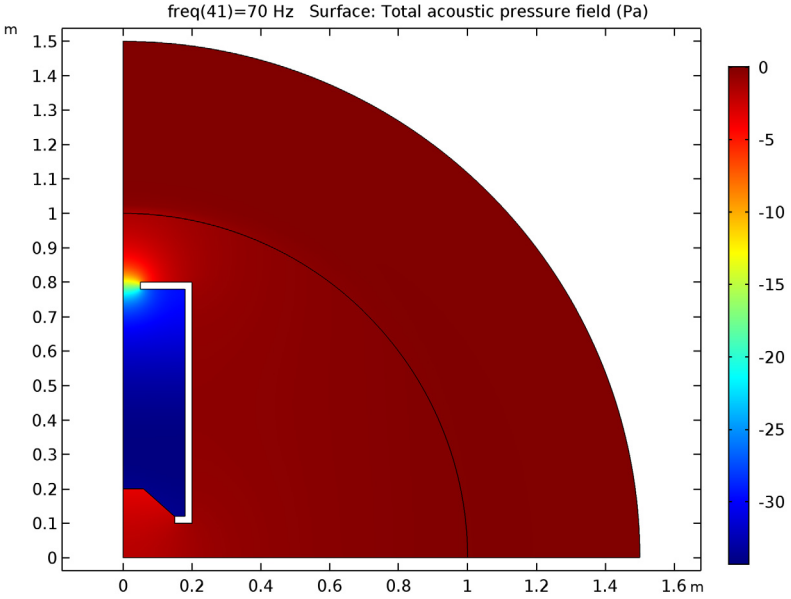
Surface 1

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

2 In the **Settings** window for **Surface**, click to expand the **Quality** section.

3 From the **Smoothing** list, choose **None**.

4 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.



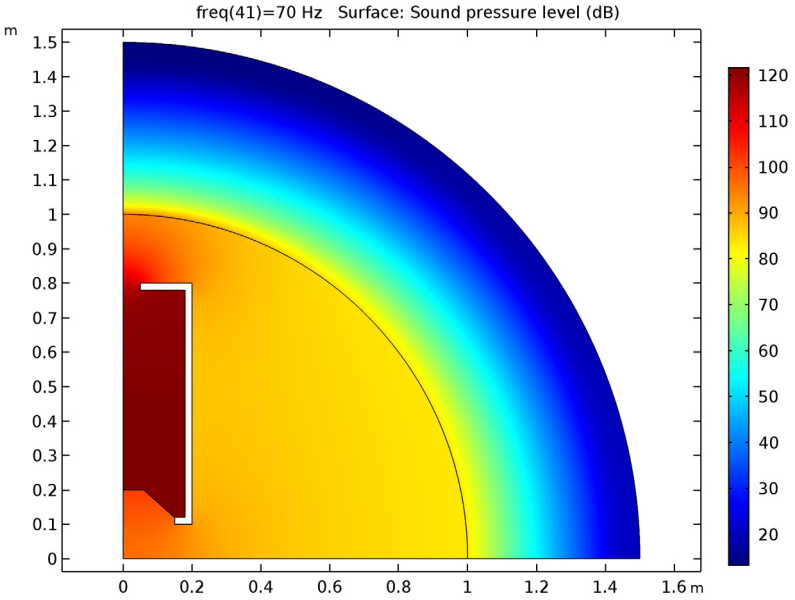
Note that the pressure drops rapidly toward zero in the PML domain.

For a better view of the damping, look at the second default plot, which shows the sound pressure level in dB.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.



With a visual inspection you can see that the pressure drop in the PML is roughly 80 dB. This means that the part of the wave that is reflected at the exterior boundary experiences a total of 160 dB damping before it returns to the physical domain. In other words, the reflected wave has virtually no effect on the solution.

Study 1/Solution 1 (1) (sol1)

For a 3D representation of the sound pressure level, look at the fourth default plot.

First create a selection to hide the PML domain:

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (1) (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domain 1 only.

Contour 1

1 In the **Model Builder** window, under **Results** right-click **Sound Pressure Level, 3D (acpr)** and choose **Contour**.

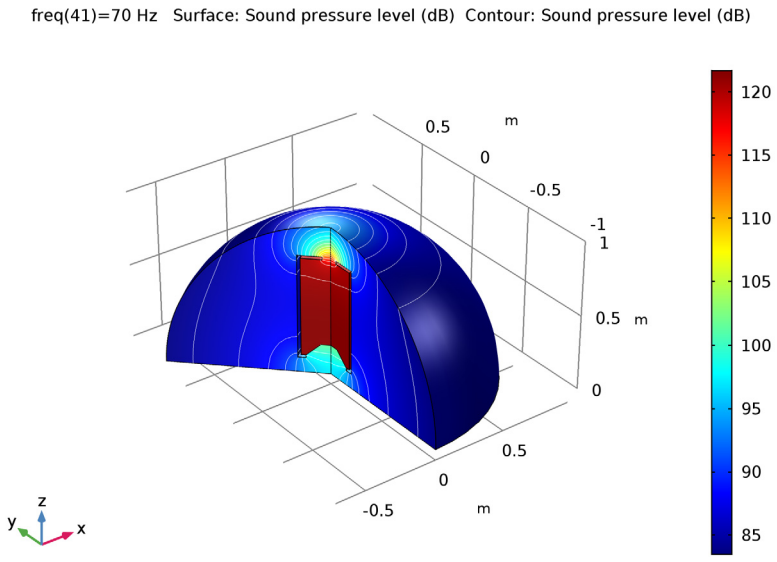
2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **White**.

5 Clear the **Color legend** check box.

6 On the **Sound Pressure Level, 3D (acpr)** toolbar, click **Plot**.



Far-Field Sound Pressure Level (acpr)

The last two default plots represent the far-field sound pressure level and the far-field pressure evaluated at 1 m in the rz-plane (change the evaluation to 2 m, to be outside the computational domain). Look at the first of those to check that the far-field pressure distribution is indeed as uniform as you would expect from a subwoofer.

Initially the spatial response is depicted for all azimuthal angles (0 deg to 360 deg) and for all the calculatd frequencies. The far-field plot has the angle defined such that 0 deg is along the z-axis, the reference direction is per default in the z-axis direction. Plotting the

response of the subwoofer in the positive hemisphere corresponds to angles from -90 deg to 90 deg. Also restrict the evaluation to the 70 Hz frequency (the last frequency).

1 In the **Model Builder** window, under **Results** click **Far-Field Sound Pressure Level (acpr)**.
2 In the **Settings** window for **Polar Plot Group**, locate the **Data** section.
3 From the **Parameter selection (freq)** list, choose **Last**.

This gets you the 70 Hz solution.

Far Field 1

1 In the **Model Builder** window, expand the **Far-Field Sound Pressure Level (acpr)** node, then click **Far Field 1**.
2 In the **Settings** window for **Far Field**, 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 `2`.
7 On the **Far-Field Sound Pressure Level (acpr)** toolbar, click **Plot**.

The plot indicates a maximum in the vicinity of 79 dB along the floor and a minimum of 77 dB straight above the speaker, and is depicted in Figure 3.

Proceed to display the previously defined impedance variable `Z_a` as function of frequency.

1D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Mechanical Impedance` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Mechanical Impedance`.
5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
6 In the associated text field, type `freq (Hz)`.
7 Select the **y-axis label** check box.
8 In the associated text field, type `(Ns/m^3)`.

Global 1

1 Right-click **Mechanical Impedance** and choose **Global**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

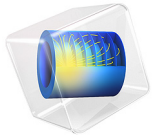
3 In the table, enter the following settings:

Expression	Unit	Description
real(Z_a)	N*s/m	Re(Z_a)
imag(Z_a)	N*s/m	Im(Z_a)

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 On the **Mechanical Impedance** toolbar, click **Plot**.

Your plot should be similar to Figure 2.



Doppler Shift

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

You can notice the Doppler effect when an ambulance or a fire engine passes by with its sirens blaring. The siren's pitch suddenly drops the moment the ambulance starts to move away from you. Another effect you can hear is how the siren's sound suddenly becomes markedly quieter as soon as the ambulance passes.

In this example, the observer and air are at rest while the sound source, the ambulance, moves with the speed V. This gives the same effect as if the sound source were at rest and the observer and air were moving at the same speed but in the opposite direction.

Model Definition

This is an axisymmetric problem with a point source at rest at the origin, $(r, z) = (0, 0)$, emitting spherical sound waves with the frequency $f = 100$ Hz. The surrounding air moves at $V = 50$ m/s (180 km/h or roughly 112 miles/hour) in the negative z direction. With this setup, the rz-plane is the horizontal plane at the level of the source and the observer, and the effects of reflections at the ground are neglected.

Assume, furthermore, that the observer stands 1 m from where the ambulance passes by. In the model geometry, this situation amounts to the observer moving with the flow along the line $r = 1$.

The boundary conditions are absorbing because there is no physical boundary around the source. This is modeled using perfectly matched layers.

Results and Discussion

The acoustic wavelength decreases for a wavefront moving in the opposite direction of the airflow. This situation corresponds to the approach stage of the ambulance, which, for the stationary observer on the ground, implies a perceived frequency that is higher than the nominal source frequency f. Conversely, the wavelength increases and the perceived frequency decreases when the acoustic wave moves with the flow during the departing stage. These two stages correspond, respectively, to the solid and dashed curves in Figure 1.

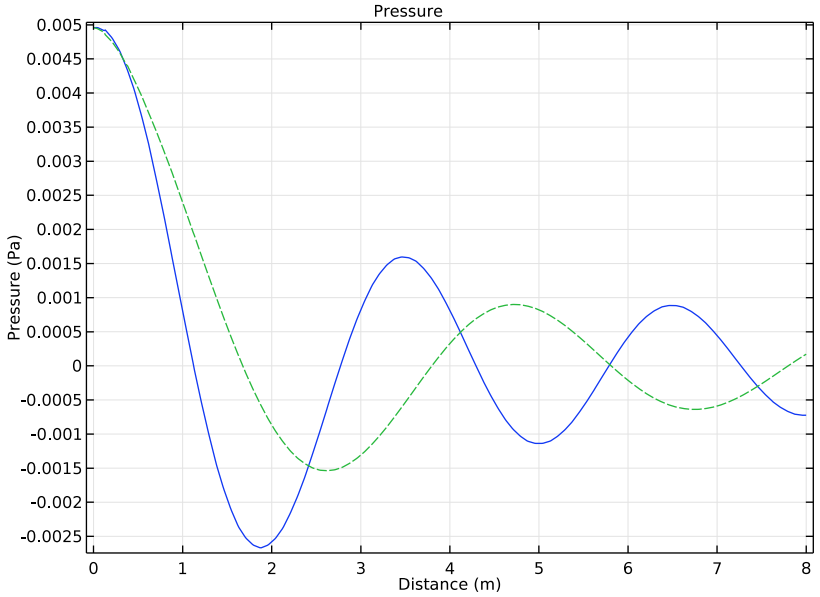


Figure 1: The x-axis represents the distance between the ambulance and the point where it passes the observer. The solid line shows the pressure perceived by the observer as the ambulance is moving towards it. The dashed line shows the pressure as the ambulance is moving away from the observer.

An inspection of Figure 1 also shows that the amplitude drops off at a faster rate when the ambulance is moving away from the observer than when it is moving towards it. This effect is also visible in Figure 2 and Figure 3, which both show the sound pressure level.

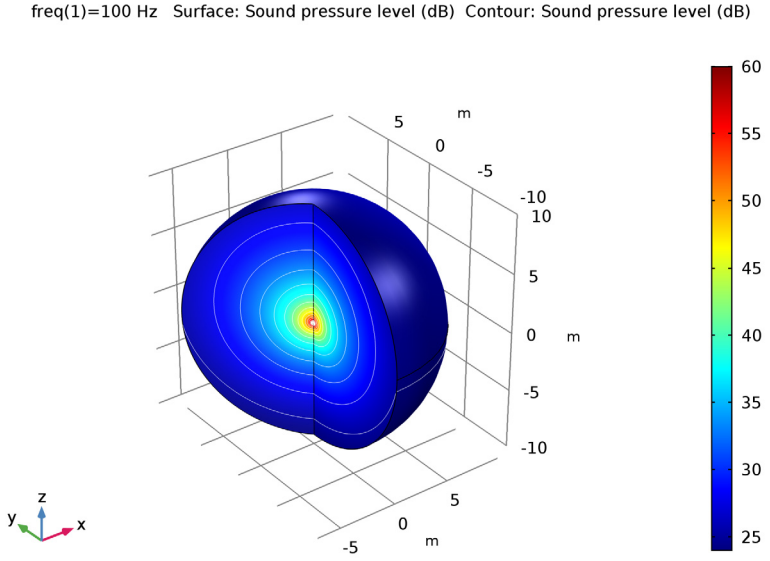


Figure 2: Sound pressure level around the point source, represented by colors and contour lines. Note how the outermost contour runs from well inside the physical domain to the PML, showing that the sound is greater below than above the source.

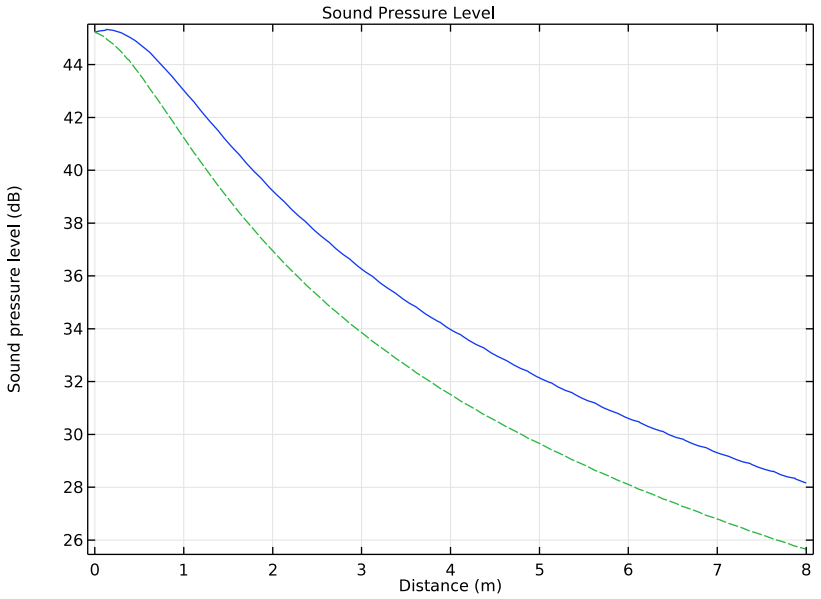


Figure 3: Sound pressure level at the observer's position during the ambulance's approach (solid, blue line) and departing (dashed, green line) versus distance from the position $(r, z) = (1\text{ m}, 0)$ *(where the observer-ambulance distance is the smallest).*

Application Library path: `Acoustics_Module/Tutorials/doppler_shift`

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>Aeroacoustics>Linearized Potential Flow, Frequency Domain (ae)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `12`.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `10`.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `12`.

4 In the **Height** text field, type `24`.

5 Locate the **Position** section. In the **z** text field, type `-12`.

Compose 1 (co1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Compose**.

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

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

4 In the **Set formula** text field, type `r1*(c1+c2)`.

5 Click **Build All Objects**.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

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

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
V	50[m/s]	50 m/s	Ambulance speed

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 1 only.

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN (AE)

Linearized Potential Flow Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Linearized Potential Flow, Frequency Domain (ae)** click **Linearized Potential Flow Model 1**.

2 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.

3 Specify the $\mathbf{V}$ vector as

-V	z

Mass Flow Point Source 1

1 On the **Physics** toolbar, click **Points** and choose **Mass Flow Point Source**.

2 Select Point 3 only.

3 In the **Settings** window for **Mass Flow Point Source**, locate the **Mass Flow Point Source** section.

4 In the *m′* text field, type `1e-4`.

MESH 1

The most important requirement on the mesh in this model is that it needs to properly resolve the waves. To get a good accuracy, you need at least 5-6 elements per wavelength. You could estimate the minimum wavelength by adding the ambulance velocity to the speed of sound in air and divide by 100 Hz. In this model however, you can easily afford a much finer mesh than the minimum requirement.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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 `0.25`.

5 Click **Build All**.

STUDY 1

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type `100[Hz]`.

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

RESULTS

Acoustic Pressure (ae)

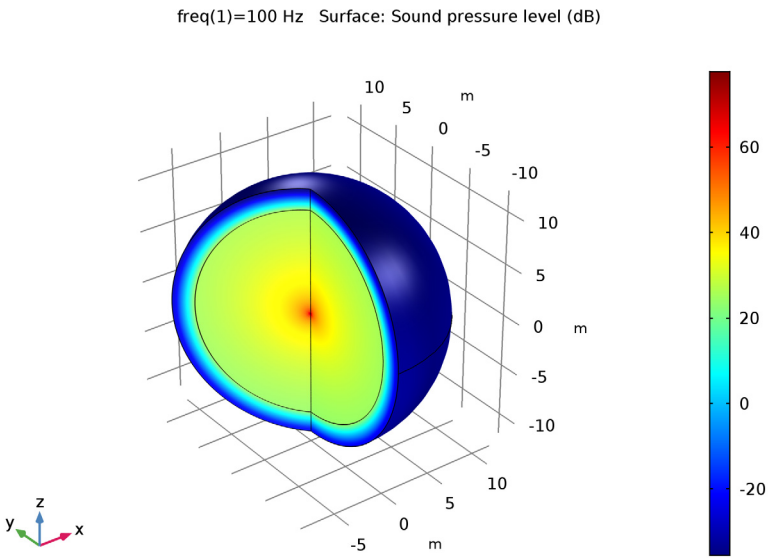
The default plot groups contain plots of the pressure and the sound pressure level, in 2D and 3D.

Sound Pressure Level, 3D (ae)

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

2 In the **Model Builder** window, under **Results** click **Sound Pressure Level, 3D (ae)**.

3 On the **Sound Pressure Level, 3D (ae)** toolbar, click **Plot**.



Contour 1

1 Right-click **Results>Sound Pressure Level, 3D (ae)** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1> Linearized Potential Flow, Frequency Domain>Pressure and sound pressure level>ae.Lp - Sound pressure level**.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **White**.

5 Clear the **Color legend** check box.

6 On the **Sound Pressure Level, 3D (ae)** toolbar, click **Plot**.

The plot is now dominated by the roughly 50 dB losses in the PML. To see only the physical domain, make a selection as follows.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domain 2 only.

Sound Pressure Level, 3D (ae)

Removing the PML from the visualization moves the bias in the color scale towards the value in the source point. Mathematically, the pressure in the vicinity of this point is inversely proportional to the distance from the point. In the point itself, this means an infinitely high pressure. The discretization results in a finite but completely mesh-dependent value (the finer the mesh, the higher the pressure). The flow from the point however is well-defined and results in a converging pressure distribution at any finite distance away from the point. To suppress the highest values and get a better picture of the sound pressure level distribution, you can set the plot range manually.

Surface

1 In the **Model Builder** window, under **Results>Sound Pressure Level, 3D (ae)** click **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Minimum** text field, type `24`.

5 In the **Maximum** text field, type `60`.

6 On the **Sound Pressure Level, 3D (ae)** toolbar, click **Plot**.

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

In order to recreate the line plots in the Results and Discussion section, you first need to create two cut lines, one going up and one going down from the starting point 1 m out from the source.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **R** to `1` and **z** to `0`.

4 In row **Point 2**, set **R** to `1` and **z** to `8`.

5 Click **Plot**.

Cut Line 2D 2

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **R** to `1` and **z** to `0`.

4 In row **Point 2**, set **R** to `1` and **z** to `-8`.

5 Click **Plot**.

With the cut lines defined, create a 1D plot group for the first plot.

1D Plot Group 5

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

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

3 Click to expand the **Title** section. 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 text field, type `Distance (m)`.

Line Graph 1

1 Right-click **Pressure on Cut Lines** and choose **Line Graph**.

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

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Linearized Potential Flow, Frequency Domain> Pressure and sound pressure level>ae.p - Pressure**.

5 Click to expand the **Legends** section.

Line Graph 2

1 Right-click **Results>Pressure on Cut Lines>Line Graph 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Cut Line 2D 2**.

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 On the **Pressure on Cut Lines** toolbar, click **Plot**.

Create another plot group for the second line plot.

1D Plot Group 6

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

2 In the **Settings** window for **1D Plot Group**, type `SPL on Cut Lines` in the **Label** text field.

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

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

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

6 In the associated text field, type `Distance (m)`.

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

8 In the associated text field, type `Sound pressure level (dB)`.

Line Graph 1

1 Right-click **SPL on Cut Lines** and choose **Line Graph**.

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

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Linearized Potential Flow, Frequency Domain> Pressure and sound pressure level>ae.Lp - Sound pressure level**.

Line Graph 2

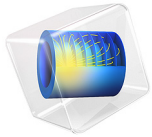
1 Right-click **Results>SPL on Cut Lines>Line Graph 1** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Cut Line 2D 2**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 On the **SPL on Cut Lines** toolbar, click **Plot**.



Eigenmodes in a Muffler

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

In this example, you compute the propagating modes in the chamber of an automotive muffler. The geometry is a cross section of the chamber in the Absorptive Muffler example.

The purpose of the model is to study the shape of the propagating modes and to find their cut-off frequencies. As discussed in the documentation for the Absorptive Muffler example, some of the modes significantly affect the damping of the muffler at frequencies above their cut-off. In this model, you study modes with cut-off frequencies up to 1500 Hz.

Model Definition

The muffler chamber has a race track shaped cross section, as seen in Figure 1. In this model, the chamber is considered to be hollow and field with air at atmospheric pressure.

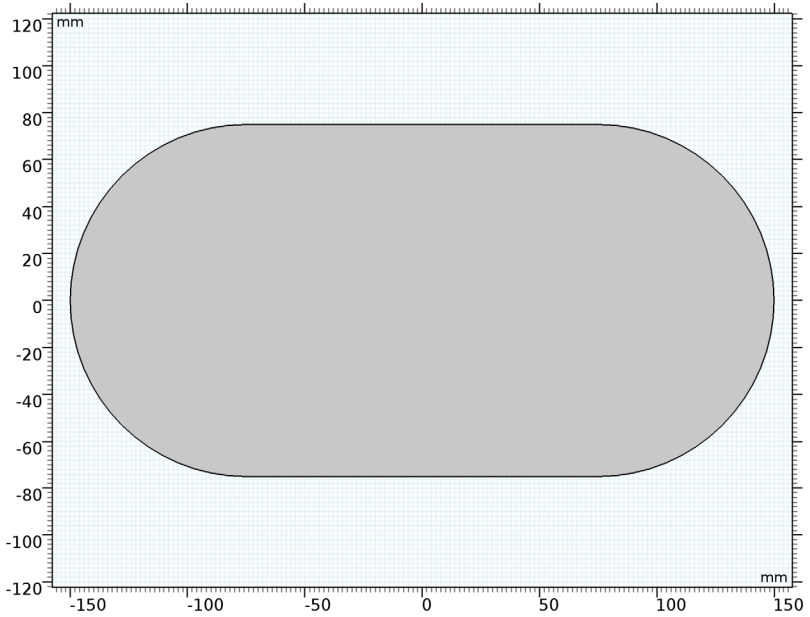


Figure 1: The model geometry.

The wave numbers and mode shapes through a cross section of the chamber are found as the solution of an eigenvalue problem for the acoustic pressure p:

$$\nabla \cdot \left(-\frac{\nabla p(x,y)}{\rho_0}\right) - \left(\frac{\omega^2}{\rho_0 c^2} - \frac{\kappa_z^2}{\rho_0}\right) p(x,y) = 0$$

where ρ_0 is the density, c the speed of sound, κ_z the out-of-plane wave number, and $\omega = 2\pi f$ the angular frequency. For a given angular frequency, only modes such that ${\kappa_z}^2$ is positive can propagate. The cutoff frequency of each mode is calculated as

$$f_j = \frac{\sqrt{\omega^2 - c^2 \kappa_z^2}}{2\pi}$$

Results and Discussion

The model finds five propagating modes, whose characteristics are summed up in the table here below.

Cutoff frequency (Hz)	**Characteristics**
0	Plane wave
635	Antisymmetric with respect to x, symmetric with respect to y
1210	Symmetric with respect to x, antisymmetric with respect to y
1240	Symmetric with respect to x and y
1467	Antisymmetric with respect to x and y

For a muffler with a centered tube leading into the chamber, the first mode that is symmetric with respect to both the x-axis and the y-axis is propagating when the frequency is higher than 1240 Hz. Figure 2 shows this mode, which for an infinitely long chamber occurs at 1240 Hz.

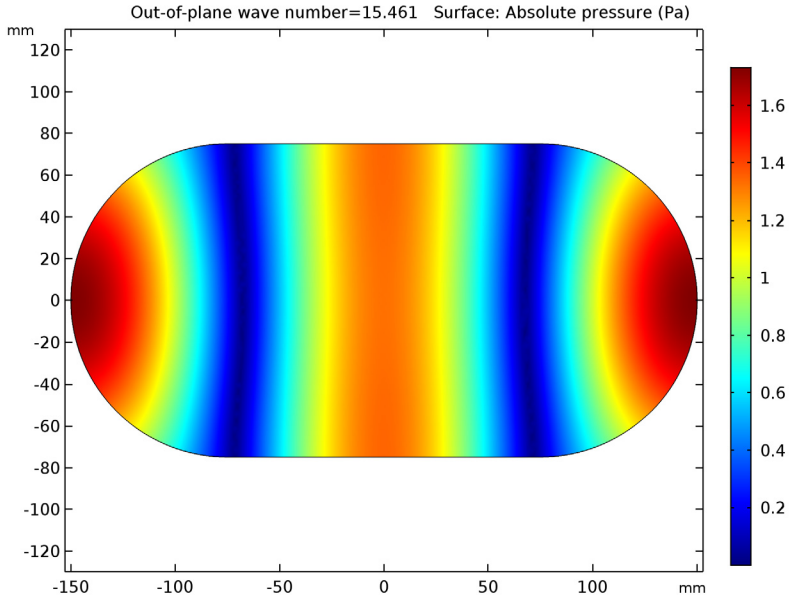


Figure 2: The chamber's first fully symmetric propagation mode. The plot shows the absolute value of the pressure.

Application Library path: `Acoustics_Module/Automotive/eigenmodes_in_muffler`

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**.

2 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Mode Analysis**.

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**.

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

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

3 In the **Side length** text field, type `150`.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `75`.

4 Locate the **Position** section. In the **x** text field, type `-75`.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `75`.

4 Locate the **Position** section. In the **x** text field, type `75`.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

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

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

4 Clear the **Keep interior boundaries** check box.

5 Click **Build All Objects**.

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

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

By default, the boundaries of the geometry will be considered to be sound hard walls. No other physics settings are needed.

MESH 1

The default mesh gives sufficiently accurate results for this analysis. You can therefore skip all mesh settings and proceed to the solver settings.

STUDY 1

Step 1: Mode Analysis

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

2 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.

3 Select the **Desired number of modes** check box.

4 In the associated text field, type `8`.

5 Select the **Search for modes around** check box.

6 In the associated text field, type `20`.

The free-space propagation mode has an out-of-plane wave number equal to `omega/c = 27.5 rad/m`. With these settings, the solver returns the 8 modes with propagation constants closest to 20 rad/m first in the list.

7 In the **Mode analysis frequency** text field, type `1500[Hz]`.

This setting makes the software look for propagating modes with cutoff frequencies up to 1500 Hz.

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

RESULTS

Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **2D Plot Group**, type `Absolute Acoustic Pressure (acpr)` in the **Label** text field.

The solver has found the free-space mode and all other propagating modes. There is a total of 5 different propagating modes. Because the waves can propagate both into and out of the modeling plane, each mode gets reported twice, with positive and negative out-of-plane wave numbers.

For the positive out-of-plane wave numbers, it holds that the higher the mode, the lower the wave number. However, the solver does not stop at zero. Because you asked for more than the 5 existing propagating modes, you get additional modes with imaginary out-of-plane wave numbers. This indicates that they are evanescent. The default plot shows the acoustic pressure distribution for a mode with a wave number of -17.95i rad/m.

3 Locate the **Data** section. From the **Out-of-plane wave number** list, choose **15.461**.

4 On the **Absolute Acoustic Pressure (acpr)** toolbar, click **Plot**.

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

This is the lowest fully symmetric mode. In order to reproduce Figure 2, plot the absolute value of the pressure.

Surface 1

1 In the **Model Builder** window, expand the **Results>Absolute Acoustic Pressure (acpr)** 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 **Model>Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.absp - Absolute pressure**.

3 On the **Absolute Acoustic Pressure (acpr)** toolbar, click **Plot**.

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

You can compute the cut-off frequency of this mode using the expression in the model introduction. In order to refer to the speed of sound in air, use an arbitrary point in the geometry for this evaluation.

Point Evaluation 1

1 On the **Results** toolbar, click **Point Evaluation**.

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

3 From the **Out-of-plane wave number selection** list, choose **From list**.

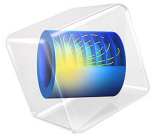
4 In the **Out-of-plane wave number** list, select **15.461**.

5 Select Point 3 only.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`sqrt(acpr.omega^2-acpr.kz^2*acpr.c^2)/(2*pi)`	`rad/s`	

7 Click **Evaluate**.



Eigenmodes in a Muffler with Elastic Walls

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The model below is an extension of the model Eigenmodes in a Muffler. The latter studied the propagation of acoustic waves through the cross section of the Absorptive Muffler chamber, provided that the muffler walls were sound hard. Here the muffler walls are considered to be made of a linear elastic material to account for their influence on the modes propagating through the cross section of the chamber. This is done using the Acoustic-Structure Interaction multiphysics interface.

Model Definition

The muffler chamber has the same shape and dimensions as in example (Eigenmodes in a Muffler). The only difference is that an elastic wall of the thickness W surrounds the inner (acoustic) domain as shown in Figure 1.

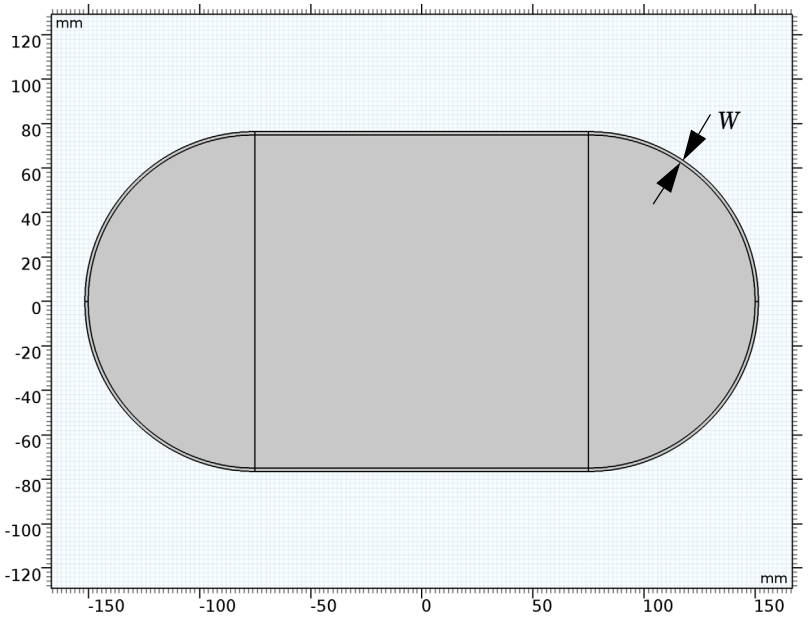


Figure 1: Geometry of model: the chamber of the same cross section as in Eigenmodes in a Muffler *with the elastic wall to the thickness* W.

The chamber is filled with air and the elastic wall is made of steel. Assuming the time harmonic process with the angular frequency $\omega = 2\pi \cdot f$, the Mode Analysis interface solves

the equations for the out-of-plane wave number κ_z as the function of given parameter f. The cutoff frequency for the j-th mode is then calculated as

$$f_j = \frac{\sqrt{\omega^2 - c^2\kappa_z^2}}{2\pi} \tag{1}$$

Results and Discussion

The results of the simulations show that the presence of a thin elastic wall significantly affects the number of modes propagating through the cross section of the chamber. It becomes greater than that for the chamber with the sound hard wall boundary studied in the model Eigenmodes in a Muffler. For example, the first propagating mode with the wave number lower than that of the plane wave mode has its cutoff frequency around 218 Hz. The former mode is shown in Figure 2; the latter, in Figure 3. Both figures shows the absolute value of the acoustic pressure in the chamber and the deformation of the muffler wall.

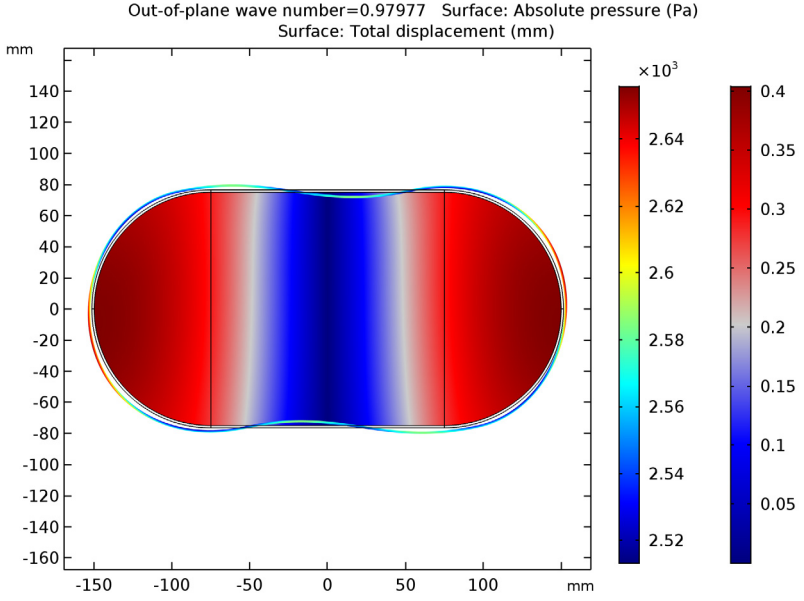


Figure 2: The plane wave propagating through the cross section of the chamber. Notice the color scale for the pressure, it is nearly the constant throughout the cross section.

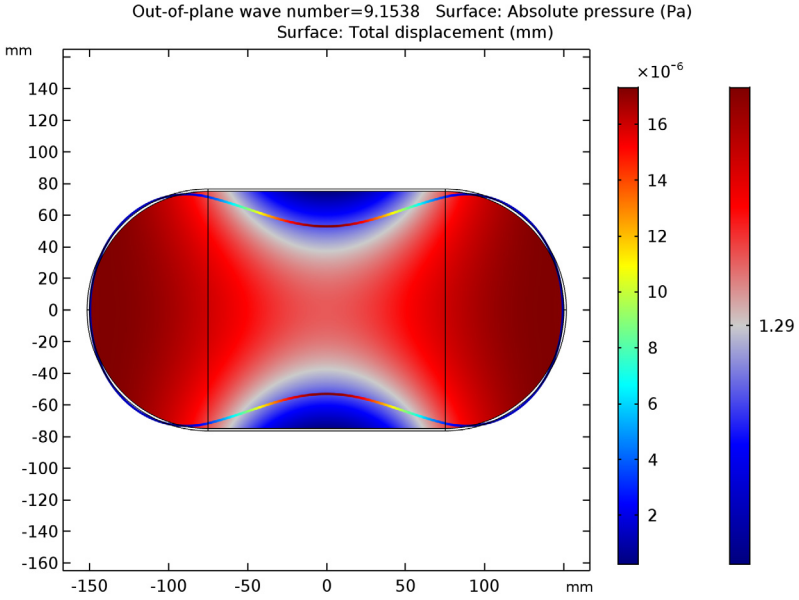


Figure 3: The first propagating mode different from the plane wave.

The model Eigenmodes in a Muffler gives the first least nonzero cutoff frequency equal to 635 Hz. A good understanding of the difference between these two cases is provided by the dispersion diagram shown in Equation 4, where the results corresponding to the presence of the wall are in black, and those obtained with the sound hard wall boundary condition are in red.

Another feature that distinguishes the chamber with the wall is that there can be modes having the wavenumber greater than that for the plane wave mode (see Figure 5), which is due to acoustics-structure interaction. This means that their cutoff frequencies f_j calculated from Equation 1 become pure imaginary. Such modes do not exist in the model Eigenmodes in a Muffler.

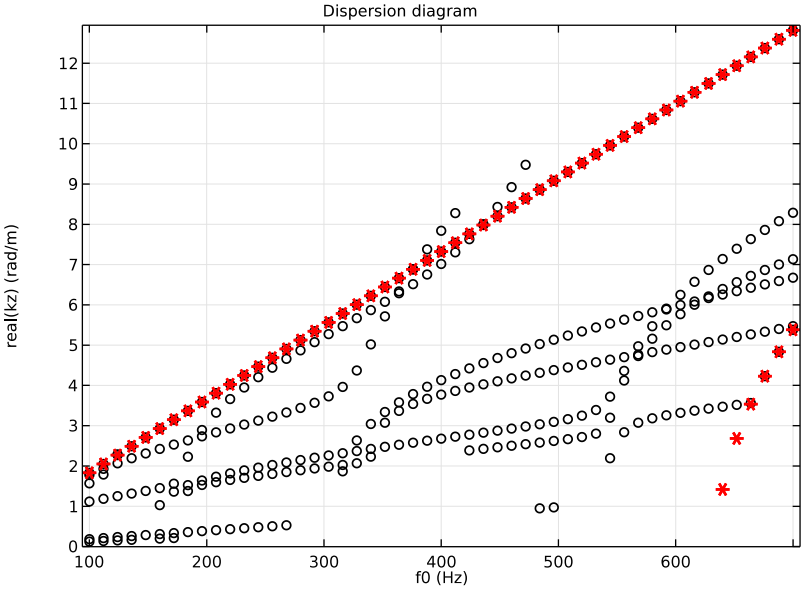


Figure 4: Dispersion diagram: the red asterisks correspond to the sound hard wall boundary; the black circles, to the elastic wall.

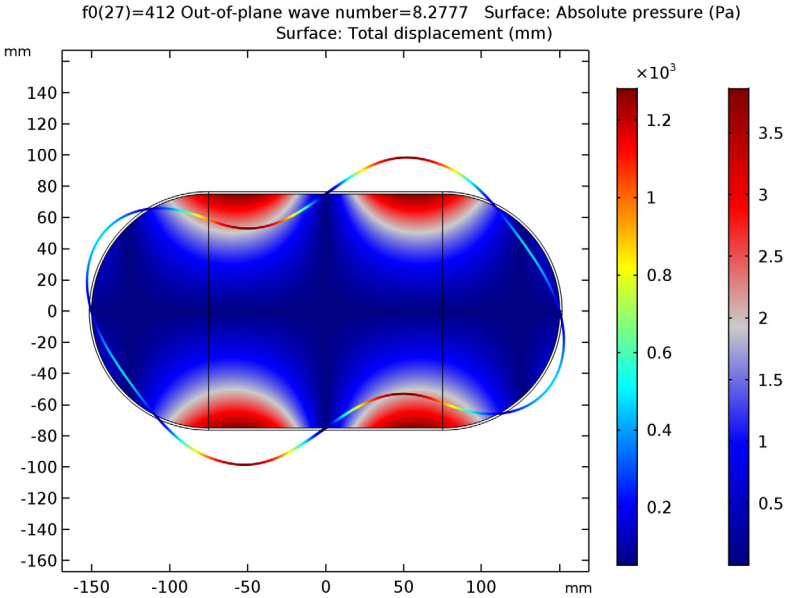


Figure 5: A propagating mode with the wavenumber greater that of the plane wave.

Application Library path: `Acoustics_Module/Automotive/eigenmodes_in_muffler_elastic`

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**.
2. In the **Select Physics** tree, select **Acoustics>Acoustic-Structure Interaction>Acoustic-Solid Interaction, Frequency Domain**.
3. Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Mode Analysis**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
W	1.5[mm]	0.0015 m	Muffler wall thickness
f0	500[Hz]	500 Hz	Cutoff frequncy
c0	343[m/s]	343 m/s	Speed of sound in the air
lambda0	c0/f0	0.686 m	Plane wave wavelength
k0	2*pi/lambda0	9.1592 1/m	Plane wave wave number

The parameter k_0 stands for the wave number of the plane wave generated at the cutoff frequency f_0: $k_0 = 2\pi/\lambda_0 = 2\pi f_0/c_0$.

GEOMETRY 1

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

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

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type 150.

4 In the **Height** text field, type 150 + 2*W.

5 Locate the **Position** section. In the **x** text field, type -75.

6 In the **y** text field, type -75 - W.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	W

8 Select the **Layers on top** check box.

9 Click **Build Selected**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `75 + W`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **x** text field, type `-75`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	W

8 Click **Build Selected**.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type `75 + W`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **x** text field, type `75`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

7 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	W

8 Click **Build Selected**.

9 On the **Geometry** toolbar, click **Build All**.

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

The geometry should look like the one in Figure 1.

ADD MATERIAL

1 On 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>Air**.

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

ADD MATERIAL

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

2 In the tree, select **Built-In>Steel AISI 4340**.

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

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

MATERIALS

Steel AISI 4340 (mat2)
Select Domains 1–3, 6, 7, and 9 only.

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 **Manual**.

4 Select Domains 4, 5, and 8 only.

SOLID MECHANICS (SOLID)

On the **Physics** toolbar, click **Pressure Acoustics, Frequency Domain (acpr)** and choose **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 **Manual**.

4 Select Domains 1–3, 6, 7, and 9 only.

Enable the **Out-of-plane mode extension (time-harmonic)** to perform the **Mode Analysis** for elastic waves in the muffler wall domain.

5 Locate the **2D Approximation** section. Select the **Out-of-plane mode extension (time-harmonic)** check box.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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

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

4 Select Domains 4, 5, and 8 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.

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

3 From the **Predefined** list, choose **Extra fine**.

Create a **Mapped** mesh for the muffler wall domain and choose the **Distribution** depth of 3 elements.

Mapped 1

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

Use the **Zoom Box** functionality to investigate the boundary layer mesh.

Distribution 1

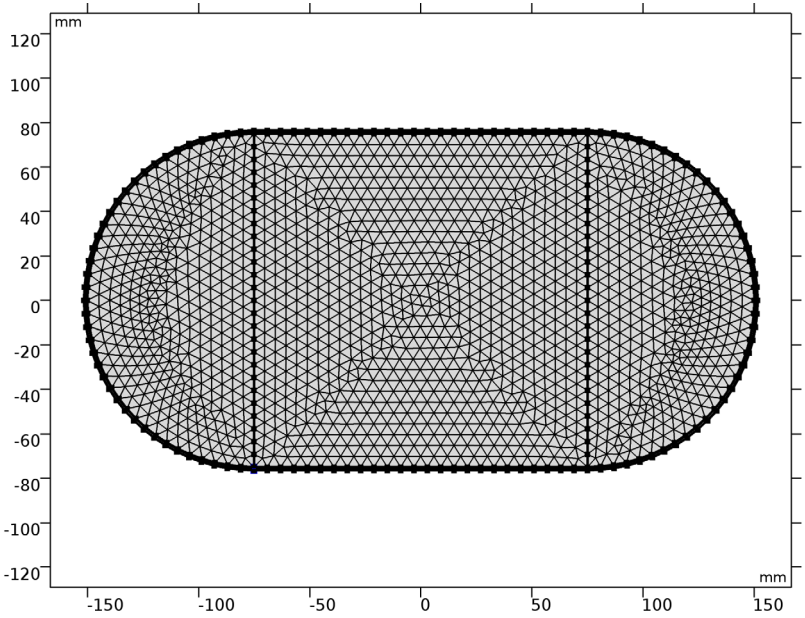
1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type 3.

5 Click **Build All**.



STUDY 1

Step 1: Mode Analysis

1 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.

2 In the **Mode analysis frequency** text field, type `f0`.

Set the solver to search for the first 6 propagating modes with the wave numbers lower that k_0.

3 Select the **Desired number of modes** check box.

4 In the associated text field, type `6`.

Search for the modes below the plane wave mode.

5 Select the **Search for modes around** check box.

6 In the associated text field, type `1.1*k0`.

7 From the **Mode search method around shift** list, choose **Smaller real part**.

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

RESULTS

Acoustic Pressure (acpr)

1. In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.
2. In the **Settings** window for **2D Plot Group**, type `Absolute Acoustic Pressure (acpr)` in the **Label** text field.
3. Locate the **Data** section. From the **Out-of-plane wave number** list, choose **9.1538**.

Surface 1

1. In the **Model Builder** window, expand the **Results>Absolute Acoustic Pressure (acpr)** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `acpr.absp`.
4. Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.
5. On the **Absolute Acoustic Pressure (acpr)** toolbar, click **Plot**.

Surface 2

1. In the **Model Builder** window, under **Results** right-click **Absolute Acoustic Pressure (acpr)** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `solid.disp`.

Deformation 1

1. Right-click **Results>Absolute Acoustic Pressure (acpr)>Surface 2** and choose **Deformation**.

   The geometry should look like the one in Figure 3.

Dispersion Relation Calculation

The next steps are optional and are added to show you how to get the dispersion relations between the frequency and the wave number. The dispersion relations for the chamber with elastic and hard walls are calculated and compared below.

ADD STUDY

1. On 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 **Preset Studies>Mode Analysis**.
4. Click **Add Study** in the window toolbar.

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

STUDY 2

Step 1: Mode Analysis

1 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.

2 In the **Mode analysis frequency** text field, type `f0`.

3 Select the **Desired number of modes** check box.

4 Select the **Search for modes around** check box.

5 In the associated text field, type `1.1*k0`.

6 From the **Mode search method around shift** list, choose **Smaller real part**.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 From the list in the **Parameter name** column, choose **f0 (Cutoff frequnecy)**.

5 Click **Range**.

6 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

7 In the **Start** text field, type `100`.

8 In the **Stop** text field, type `700`.

9 In the **Number of values** text field, type `51`.

10 Click **Replace**.

11 In the **Model Builder** window, click **Study 2**.

12 In the **Settings** window for **Study**, locate the **Study Settings** section.

13 Clear the **Generate default plots** check box.

14 On the **Study** toolbar, click **Compute**.

ADD STUDY

1 On the **Study** 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 **Preset Studies>Mode Analysis**.

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

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

STUDY 3

Step 1: Mode Analysis

1 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.

2 In the **Mode analysis frequency** text field, type `f0`.

3 Select the **Desired number of modes** check box.

4 Select the **Search for modes around** check box.

5 In the associated text field, type `1.1*k0`.

6 From the **Mode search method around shift** list, choose **Smaller real part**.

Now, disable the **Solid Mechanics** physics which results in the outer wall of the acoustic domain to be sound hard.

7 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Discretization
Solid Mechanics (solid)		physics

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 From the list in the **Parameter name** column, choose **f0 (Cutoff frequnecy)**.

5 Click **Range**.

6 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

7 In the **Start** text field, type `100`.

8 In the **Stop** text field, type `700`.

9 In the **Number of values** text field, type `51`.

10 Click **Replace**.

11 In the **Model Builder** window, click **Study 3**.

12 In the **Settings** window for **Study**, locate the **Study Settings** section.

13 Clear the **Generate default plots** check box.

14 On the **Study** toolbar, click **Compute**.

RESULTS

1D Plot Group 4

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.
3. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `Dispersion diagram`.
5. In the **Label** text field, type `Dispersion Relation`.
6. Locate the **Data** section. From the **Data set** list, choose **Study 2/ Parametric Solutions 1 (sol3)**.
7. Locate the **Plot Settings** section. Select the **x-axis label** check box.
8. In the associated text field, type `f0 (Hz)`.
9. Select the **y-axis label** check box.
10. In the associated text field, type `real(kz) (rad/m)`.
11. Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1. Right-click **Dispersion Relation** 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
`if(abs(imag(acpr.kz))/abs(acpr.kz) < 1e-3, abs(real(acpr.kz)), NaN)`	`rad/m`	

4. Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.
5. From the **Parameter** list, choose **Expression**.
6. In the **Expression** text field, type `f0`.
7. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
8. From the **Color** list, choose **Black**.
9. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
10. From the **Positioning** list, choose **In data points**.

Global 2

1. Right-click **Results>Dispersion Relation>Global 1** and choose **Duplicate**.

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

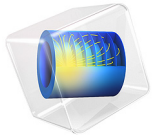
3 From the **Color** list, choose **Red**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

5 Locate the **Data** section. From the **Data set** list, choose **Study 3/ Parametric Solutions 2 (sol56)**.

6 On the **Dispersion Relation** toolbar, click **Plot**.

The dispersion diagrams should look like the ones in Figure 4.



Flow Duct

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The modeling of aircraft-engine noise is a central problem in the field of computational aeroacoustics (CAA). In this example you simulate the harmonically time-varying acoustic field from a turbofan engine under various conditions and calculate the attenuation of the acoustic noise made possible by introducing a layer of lining inside the engine duct.

Model Definition

Assume that the flow in the axisymmetric duct is compressible, inviscid, perfectly isentropic, and irrotational. In this case the acoustic field is well described by the linearized potential flow equations. The Linearized Potential Flow, Frequency Domain interface is used to set up the model.

The flow is in this model described by Euler's equations for an ideal gas (assuming adiabatic processes):

$$\frac{\partial \tilde{\rho}}{\partial t} + \nabla \cdot (\tilde{\rho}\tilde{\mathbf{v}}) = 0$$

$$\tilde{\rho}\left(\frac{\partial \tilde{\mathbf{v}}}{\partial t} + \tilde{\mathbf{v}} \cdot \nabla \tilde{\mathbf{v}}\right) + \nabla \tilde{p} = 0$$

$$\gamma \tilde{p} = \tilde{\rho}^{\gamma} \qquad \tilde{c}^2 = \frac{d\tilde{p}}{d\tilde{\rho}} = \tilde{\rho}^{\gamma - 1}$$

Here $\tilde{\rho}$ is the density, $\tilde{\mathbf{v}}$ equals the velocity, $\tilde{p}$ denotes the pressure, $\tilde{c}$ equals the speed of sound, and γ is the constant ratio of the specific heats at constant pressure and volume. The variables are made dimensionless by division by suitable combinations of a reference duct radius R_∞, a reference speed of sound c_∞, and a reference density ρ_∞.

LINEARIZED POTENTIAL FLOW EQUATIONS

Because the flow is assumed irrotational, you can describe the velocity field, $\tilde{\mathbf{v}} = \tilde{\mathbf{v}}(r, z, t)$, in terms of a potential $\tilde{\phi}$, defined by the equation $\tilde{\mathbf{v}} = \nabla\tilde{\phi}$. The basic time- and space-dependent variables describing the flow are then the velocity potential and the density, $\tilde{\rho}$. These variables (and the velocity field itself) are split into a stationary mean-flow part and a harmonically time-varying acoustic part:

$$\tilde{\phi} = \Phi + \phi e^{i\omega t} \qquad \tilde{\mathbf{v}} = \mathbf{V} + \mathbf{v}e^{i\omega t} \qquad \tilde{\rho} = \rho_0 + \rho e^{i\omega t}$$

where ϕ, $\mathbf{v}$, and ρ are the acoustic variations to the potential, velocity, and density, respectively. Also assume that the amplitudes of the acoustic variables are small compared

to the corresponding mean-flow quantities. This allows for a linearization of the equations of motion and the equation of state. The linearized potential flow equations for the acoustic variables are

$$i\omega\rho + \nabla \cdot (-\rho_0 \nabla\phi + \rho\mathbf{V}) = 0$$
$$\rho_0(i\omega\phi + \mathbf{V} \cdot \nabla\phi) = p$$
$$p = c_0^2\rho$$

For more theory information, see the aeroacoustics theory chapter in the *Acoustics Module User's Guide*.

GEOMETRY AND BOUNDARY CONDITIONS

The duct geometry used in this model, shown in Figure 1, is taken from Ref. 1. It is an approximate model of the inlet section of a turbofan engine in the very common CFM56 series.

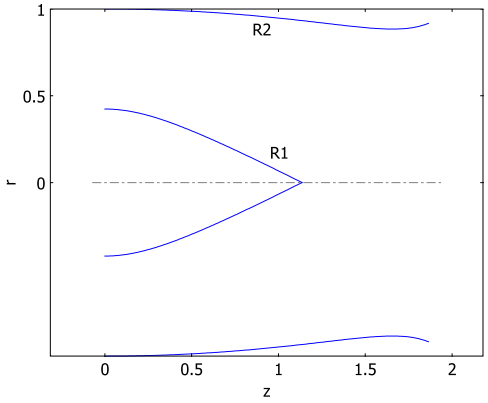


Figure 1: The duct geometry.

The spinner and duct-wall profiles are given, respectively, by the equations

$$R_1(z') = \max[0, 0.64212 - (0.04777 + 0.98234z'^2)^{1/2}]$$
$$R_2(z') = 1 - 0.18453z'^2 + 0.10158\frac{e^{-11(1-z')} - e^{-11}}{1 - e^{-11}}$$

where $0 \le z' = z/L \le 1$, and $L = 1.86393$ is the duct length. A noise source is imposed at $z' = 0$, henceforth referred to as the *source plane*. This is where the fan would be located

in the actual engine geometry. The plane $z = L$ corresponds to the fore end of the engine and is referred to as the *inlet plane*.

For the reference quantities in this model, choose the duct radius, the mean-flow speed of sound, and the mean-flow density at the source plane. Hence, all three of these quantities take the value 1.

To facilitate the COMSOL Multiphysics modeling, add a set of auxiliary domains to the geometry:

- A cylindrical domain—adjoined at the inlet plane and extending to the *terminal plane*, $z = 2.86393$—extends the modeling domain into a region where you can consider the mean flow as being uniform. This allows you to impose the simple boundary condition of a constant velocity potential and a vanishing tangential velocity for the background flow at the terminal plane.
- PML domains, adjoined at the source and terminal planes, allow you to conveniently implement nonreflecting boundary conditions for the aeroacoustic field. At the source plane, the PML domain is split into three annular sections with the innermost and outermost sections damping both in the axial and radial directions, while the central one is damping only in the axial direction. For more information about PMLs, see the *COMSOL Multiphysics Reference Manual*.

The remaining boundary conditions for the mean flow consist of a natural boundary condition specifying the mass-flow rate through the source plane via the normal velocity and the density; slip conditions (vanishing tangential velocity) at the duct wall and at the spinner; and axial symmetry at $r = 0$.

For the aeroacoustic field, the model considers two different boundary conditions at the duct wall:

- *Sound hard*—the normal component of the acoustic particle velocity vanishes at the boundary.
- *Impedance*—the normal component of the acoustic particle velocity is related to the particle displacement through the equation

$$i\omega(\mathbf{v}\cdot\mathbf{n}) = [i\omega + \mathbf{V}\cdot\nabla - (\mathbf{n}\cdot(\mathbf{n}\cdot\nabla\mathbf{V}))]\frac{p}{Z}$$

  where Z is the impedance. This boundary condition, first derived by Myers (Ref. 2), was later recast in a weak form by Eversman (Ref. 3); it is this weak version, which is directly suitable for finite element modeling, that is implemented in the Acoustics Module's Linearized Potential Flow, Frequency Domain interface. The impedance boundary

condition represents a lined duct wall. In this model, following Ref. 1, the impedance is taken to be $Z = 2 - i$.

The spinner, in contrast, is always assumed to be acoustically hard.

This study examines two cases for the mean-flow normal velocity component at the source plane, V_z, which (owing to the choice of reference speed) alternatively can be referred to as the source-plane axial Mach number, M: $M = -0.5$, approximately representative of a passenger aircraft at cruising speed, and $M = 0$.

The dimensionless angular frequency (nondimensionalized through division by R_∞/c_∞) is $\omega = 16$, and the circumferential mode number is $m = 10$. If you want to obtain a deeper understanding of the duct's aeroacoustic characteristics, you can, of course, perform a systematic exploration of parameter space by varying these quantities independently.

Results and Discussion

THE MEAN-FLOW FIELD

For the nontrivial case of a source-plane axial Mach number of $M = -0.5$, the resulting mean-flow field appears in Figure 2. Note that the velocity potential is uniform well beyond the terminal plane, thus justifying the boundary condition imposed there. Furthermore, as could be expected, deviations from the mean density value appear primarily near the nonuniformities of the duct geometry, such as at the tip of the spinner.

As a complement, a more quantitative picture of the variations of the mean-flow velocity and density profiles along the axial direction appear in the cross-section plots in Figure 3.

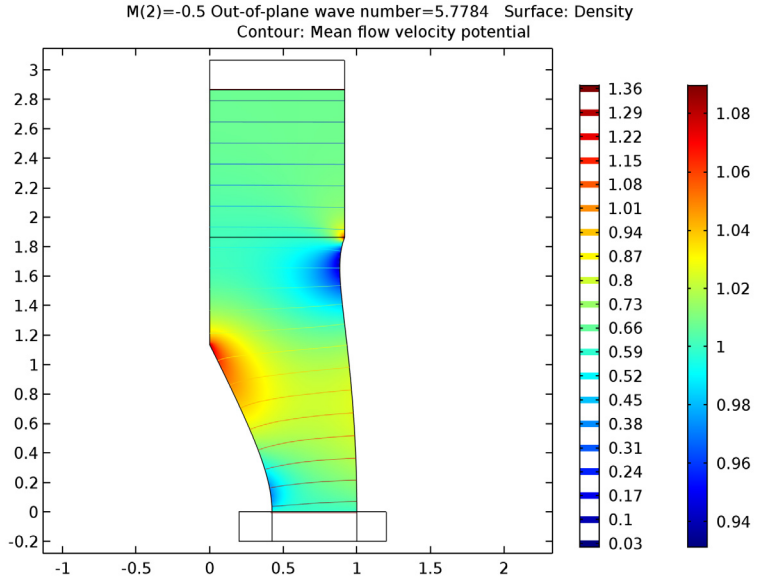


Figure 2: Mean-flow velocity potential and density for source-plane Mach number $M = -0.5$.

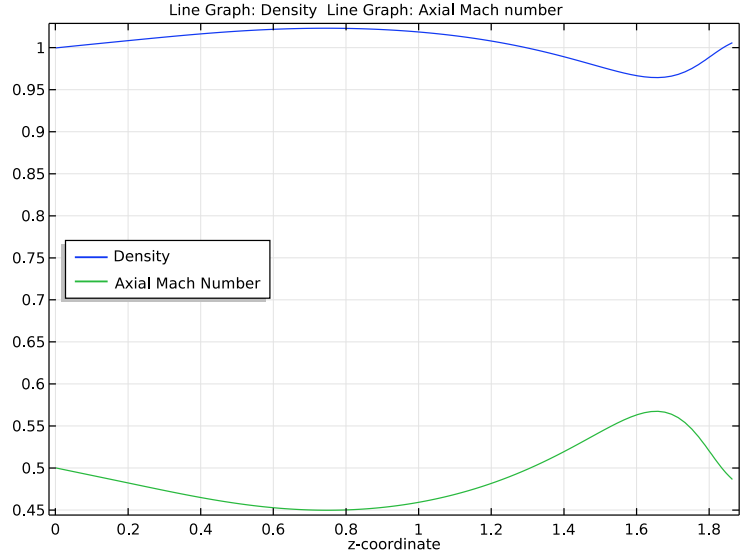


Figure 3: Mean-flow cross-section plot at a sample radius of 0.8.

THE NOISE SOURCE

With the solution for the mean-flow field at hand, it is possible to calculate the corresponding eigenmodes for the acoustic field at the source plane. Figure 4 shows the resulting velocity-potential profile for the lowest mode. This is the boundary mode used as the source of the acoustic noise field in the duct for the case $M = -0.5$.

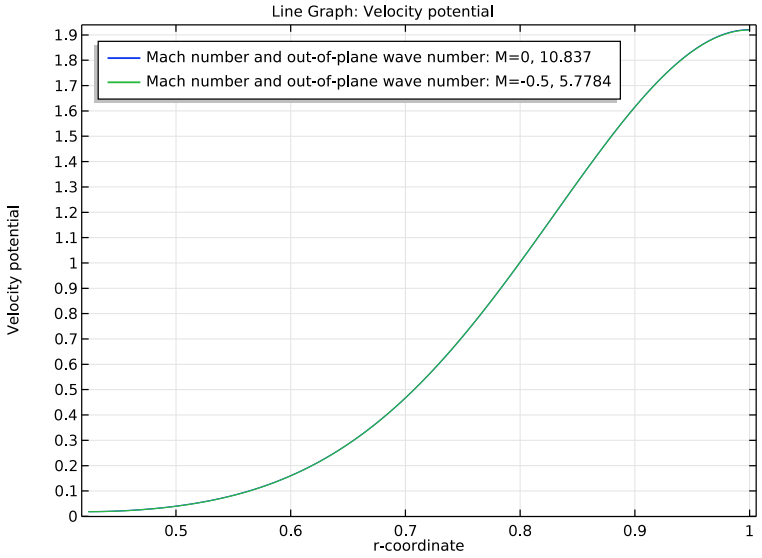


Figure 4: The first axial boundary mode at the source plane ($z = 0$) for the case of a background flow with Mach number $M = -0.5$.

THE AEROACOUSTIC FIELD

The pressure fields for the case without a background mean flow, shown in Figure 5, very closely match those for the corresponding finite element model (FEM) solutions presented in Figure 6 of Ref. 1. Similarly, the results for the attenuation between the source and inlet planes in the lined-wall case are in good agreement: 50.6 dB for the COMSOL Multiphysics solution versus 51.6 dB for the FEM solution in Ref. 1.

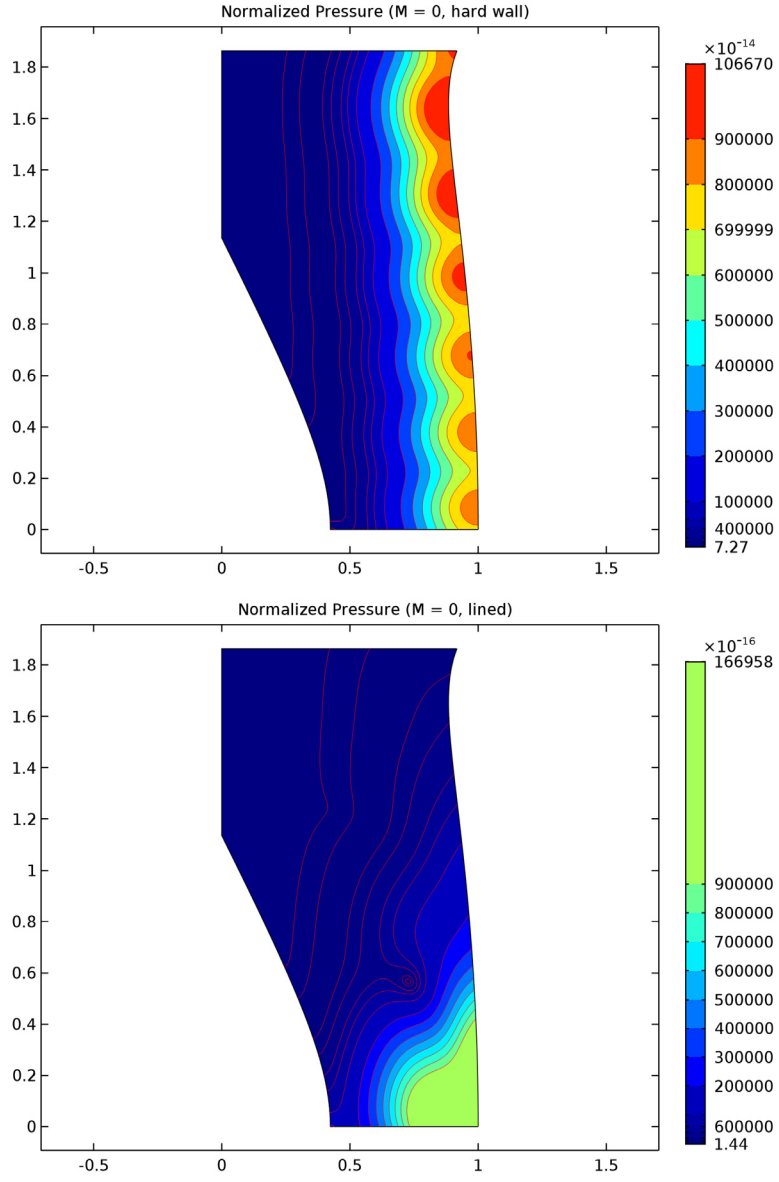


Figure 5: Acoustic pressure field for the cases of hard (top) and lined (bottom) duct wall with no mean flow and at circumferential mode number m = 10 and angular frequency $\omega = 16$.

Turning to the case with a mean flow, the pressure field for the hard-wall case in the upper image of Figure 6 closely resembles the FEM solution obtained by Rienstra and Eversman in Ref. 1. For the lined-wall case in the lower image, although the agreement is still quite good, you can note some differences, especially near the source plane. This observation extends to the attenuation, for which the calculated value of 25.2 dB differs slightly more from the value of 27.2 dB obtained in Ref. 1.

However, these discrepancies have a natural explanation: the source mode in the COMSOL Multiphysics calculation was derived for the case of a hard duct wall, whereas Rienstra and Eversman used a noise source adapted to the acoustic lining. The lowest mode for the lined-wall case is a linear combination of the two forward-propagating hard-wall modes. Thus, the noise source term used to obtain the FEM solution visualized in the lower plot of Figure 6 is not optimally adapted to the duct, and it is consequently not maximally attenuated.

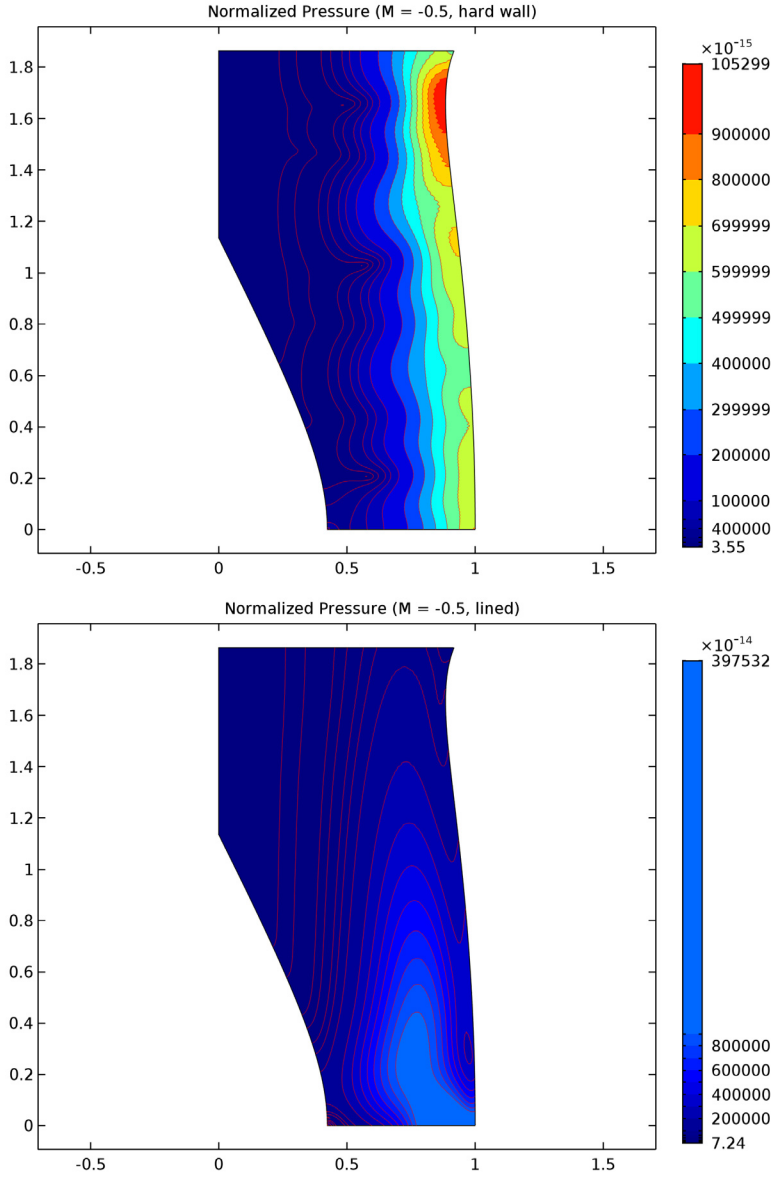


Figure 6: Acoustic pressure distribution for the cases of hard (top) and lined (bottom) duct wall with mean flow ($M = -0.5$) and at circumferential mode number $m = 10$ and angular frequency $\omega = 16$.

Notes About the COMSOL Implementation

The model involves three physics interfaces, the last of which is used twice:

- *Compressible Potential Flow* (cpf)—for modeling the background mean-flow velocity field as a potential flow (a lossless and irrotational flow).
- *Linearized Potential Flow, Boundary Mode* (aebm)—for calculating the boundary eigenmode to be used as the source of the acoustic noise in the background mean-flow.
- *Linearized Potential Flow, Frequency Domain* (ae, ae2)—for modeling the time-harmonic acoustic field above and below the source plane.

After an initial modeling stage—consisting of creating the geometry and the mesh, then defining parameters, variables, and component couplings—you proceed with three consecutive stages corresponding to the items in the list above.

As explained in the Model Definition section, this model uses nondimensional variables obtained by dividing each variable by a suitable reference quantity of the same dimension. The reference length is the duct radius at the source plane (which is why it has the value 1). The mean-flow density and speed of sound at the source plane ($z = 0$) complete the set of reference variables.

References

1. S.W. Rienstra and W. Eversman, "A Numerical Comparison Between the Multiple-Scales and Finite-Element Solution for Sound Propagation in Lined Flow Ducts," *J. Fluid Mech.*, vol. 437, pp. 367–384, 2001.

2. M.K. Myers, "On the Acoustic Boundary Condition in the Presence of Flow," *J. Sound Vib.*, vol. 71, pp. 429–434, 1980.

3. W. Eversman, "The Boundary Condition at an Impedance Wall in a Non-Uniform Duct with Potential Mean Flow," *J. Sound Vib.*, vol. 246, pp. 63–69, 2001. Errata: *ibid*, vol. 258, pp. 791–792, 2002.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/flow_duct`

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>Aeroacoustics> Compressible Potential Flow (cpf)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Acoustics>Aeroacoustics>Linearized Potential Flow, Boundary Mode (aebm)**.
5. Click **Add**.
6. In the **Velocity potential** text field, type `phi_b`.
7. In the **Select Physics** tree, select **Acoustics>Aeroacoustics>Linearized Potential Flow, Frequency Domain (ae)**.
8. Click **Add** twice.
9. Click **Study**.
10. In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Stationary**.
11. Click **Done**.

ROOT

1. In the **Model Builder** window, click the root node.
2. In the root node's **Settings** window, locate the **Unit System** section.
3. From the **Unit system** list, choose **None**.

   This setting turns off all unit support in the model.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
gamma	1.4	1.4	Ratio of specific heats
M	-0.5	-0.5	Mean flow Mach number
m	10	10	Circumferential mode number
omega	16	16	Angular frequency
f	omega/(2*pi)	2.5465	Frequency
rho0	1	1	Reference density
c0	1	1	Reference mean speed of sound
A	0.01	0.01	Acoustic source strength
Zw	2-i	2-i	Duct wall impedance
b	0.01	0.01	Impedance onset rate
zi	1.86393	1.8639	Axial coordinate, inlet plane
zt	2.86393	2.8639	Axial coordinate, terminal plane

Alternatively, you can load the parameters from file. To do this, click the Load from File button, browse to the application's Application Library folder, and then double-click the file `flow_duct_parameters.txt`.

GEOMETRY 1

First import the duct geometry, which is supplied in the form of an MPHBIN-file.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

Next, add the auxiliary cylindrical domain between the inlet plane at z = 1.86393 and the terminal plane at z = 2.86393.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

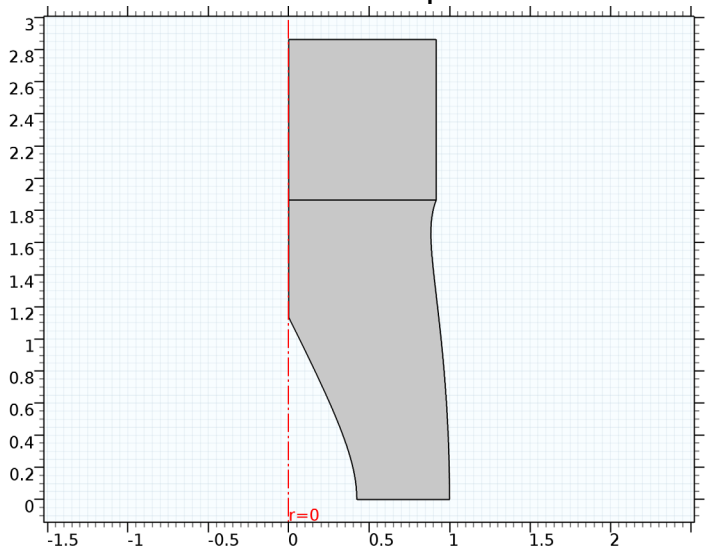
3 In the **Width** text field, type `0.91705`.

4 In the **Height** text field, type `1`.

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

6 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

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



Finally, attach cylindrical PML domains at both ends.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

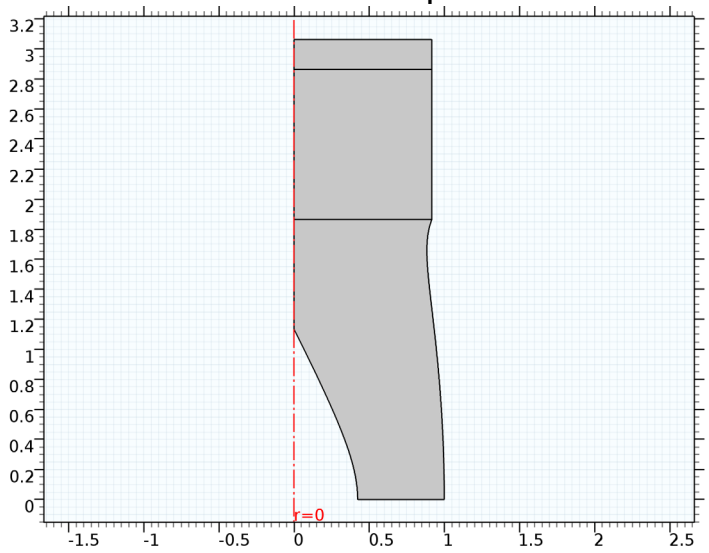
3 In the **Width** text field, type `0.91705`.

4 In the **Height** text field, type `0.2`.

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

6 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

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



Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `1`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **r** text field, type `0.2`.

6 In the **z** text field, type `-0.2`.

7 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

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

Rectangle 4 (r4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.57644`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **r** text field, type `0.42356`.

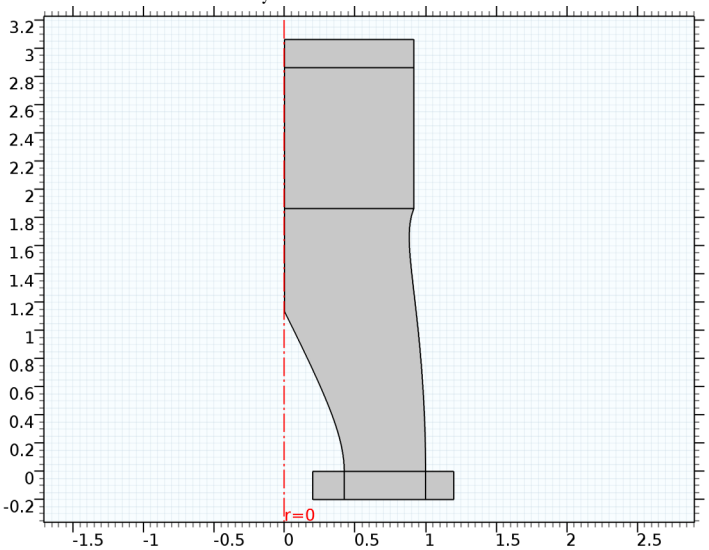
6 In the **z** text field, type `-0.2`.

7 Right-click **Rectangle 4 (r4)** and choose **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

The model geometry is now complete. The axisymmetric model geometry including duct domain and auxiliary domains.



MESH 1

Create a mapped mesh that is sufficiently fine to resolve the small-scale acoustic perturbations by following the instructions below.

Distribution 1

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

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundary 1 only.

This is the duct domain's boundary along the symmetry axis.

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

5 In the **Number of elements** text field, type 39.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 3 only.

This is the symmetry-axis boundary segment for the auxiliary domain above the duct.

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

4 In the **Number of elements** text field, type `60`.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 6 and 43 only.

These are the source-plane and terminal-plane boundaries. Note that you can make the selection by clicking the Paste Selection button and typing the indices in the dialog box that opens.

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

4 In the **Number of elements** text field, type `40`.

Distribution 4

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 5, 19, 20, 96, and 97 only.

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

4 In the **Number of elements** text field, type `18`.

Distribution 5

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type `1`.

Distribution 6

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 8–18, 22–39, and 66–94 only.

You can do this most easily by copying the text `8-18, 22-39, 66-94` and then clicking in the **Selection** box and pressing Ctrl+V or by using the **Paste Selection** dialog box.

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

4 In the **Number of elements** text field, type `3`.

Distribution 7

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 40 and 95 only.

Distribution 8

1 Right-click **Mapped 1** and choose **Distribution**.

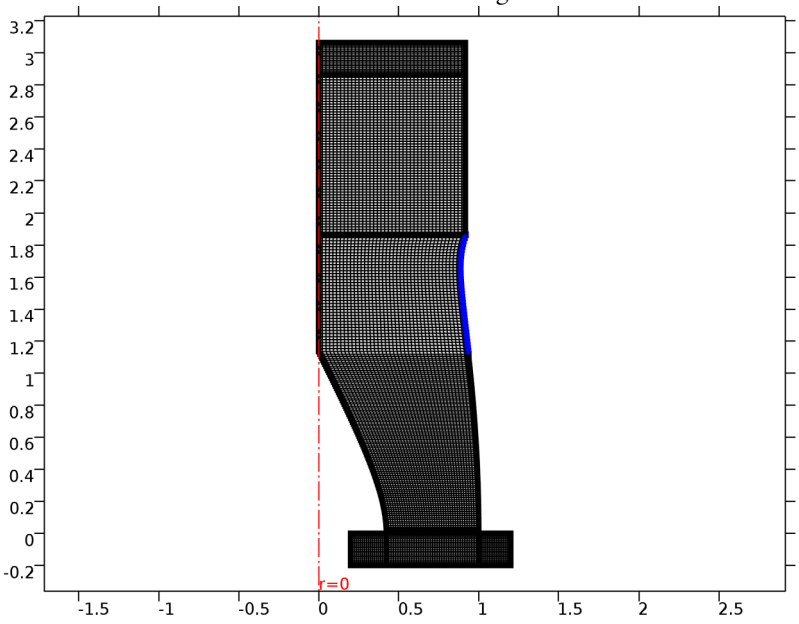
2 Select Boundaries 44–59 and 62–65 only.

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

4 In the **Number of elements** text field, type 2.

5 Click **Build All**.

The finished mesh should look like that in the figure below.



DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

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

4 Select Domains 1 and 2 only.

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

Name	Expression	Description
C	sqrt(gamma*cpf.pref*rho^(gamma-1)/ cpf.rhoref^gamma)	Mean flow speed of sound
Mz	-cpf.Vz	Axial Mach number

Next, define an expression for the source mode's intensity component normal to the source boundary.

Variables 2

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

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

4 Select Boundary 43 only.

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

Name	Expression	Description
Iz_src	A^2*0.5*real((aebm.p/cpf.rhoref+ aebm.Vr*aebm.vr-aebm.Vz*aebm.ikz* phi_b)*conj(rho*aebm.Vz-cpf.rhoref* aebm.ikz*phi_b))	Source mode intensity, normal component

Proceed by defining integration operators for the source and inlet planes for use in computing the attenuation.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type intop_src in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 43 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type intop_inl in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

Using these operators, define variables for the power through the source and inlet planes.

Variables 3

1 On the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Description
W_src	intop_src(Iz_src)	Power through source plane
W_inl	intop_inl(ae.Iz)	Power through inlet plane

The integral is automatically performed in the full azimuthal direction, because of the option selected in the integration operator.

Because the variables you just defined cannot be evaluated for the same solution data sets, it is not possible to define a variable for the attenuation. Instead, create an analytic function that returns the attenuation when supplied with two power values.

Analytic 1 (an1)

1 On the **Definitions** toolbar, click **Analytic**.

2 In the **Settings** window for **Analytic**, type `Attenuation` in the **Label** text field.

3 In the **Function name** text field, type `dw`.

4 Locate the **Definition** section. In the **Expression** text field, type `10*log10(w_src/w_in)`.

5 In the **Arguments** text field, type `w_src, w_in`.

The Background Flow and the Source

In the following modeling steps you derive the stationary background as well as the acoustic source at the duct entrance. Both are used when modeling the time-harmonic acoustic perturbations that you will set up following this.

You calculate the stationary flow field using the Compressible Potential Flow interface defined on the duct geometry (Domain 1) and on the auxiliary region (Domain 2) appended at the inlet plane (z = 1.86393). Subsequently, you extend the flow to the PML domains by continuity. Impose a mass-flow boundary condition at the source plane and a normal-flow condition at the terminal plane (z = 2.86393). The duct wall and the spinner are both impervious to the flow.

As the source generating the acoustic field in the duct, use a single boundary mode imposed at z = 0. More specifically, take this mode to be the lowest propagating axial mode in the duct computed in the background flow field from the previous stage of the modeling process. The subsequent instructions demonstrate how to derive this boundary mode.

Proceed with setting up the background flow physics and couple it to the mode analysis model. Then solve everything coupled for two values of the Mach number (M = 0 and M = -0.5). The solution data sets will include both background flow and source data.

COMPRESSIBLE POTENTIAL FLOW (CPF)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Compressible Potential Flow (cpf)**.
2. Select Domains 1 and 2 only.
3. In the **Settings** window for **Compressible Potential Flow**, locate the **Reference Values** section.
4. In the p_{ref} text field, type `cpf.rhoref^gamma/gamma`.
5. In the ρ_{ref} text field, type `rho0`.
6. In the v_{ref} text field, type `M`.

Compressible Potential Flow Model 1

1. In the **Model Builder** window, expand the **Compressible Potential Flow (cpf)** node, then click **Compressible Potential Flow Model 1**.
2. In the **Settings** window for **Compressible Potential Flow Model**, locate the **Compressible Potential Flow Model** section.
3. From the γ list, choose **User defined**. In the associated text field, type `gamma`.

Normal Flow 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Normal Flow**.
2. Select Boundary 6 only.

Mass Flow 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Mass Flow**.
2. Select Boundary 43 only.
3. In the **Model Builder** window, collapse the **Compressible Potential Flow (cpf)** node.
4. In the **Model Builder** window, collapse the **Compressible Potential Flow (cpf)** node.

STUDY 1

Now set up Linearized Potential Flow, Boundary Mode physics for the boundary mode source calculation.

LINEARIZED POTENTIAL FLOW, BOUNDARY MODE (AEBM)

On the **Physics** toolbar, click **Compressible Potential Flow (cpf)** and choose **Linearized Potential Flow, Boundary Mode (aebm)**.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Boundary Mode (aebm)**.
2. In the **Settings** window for **Linearized Potential Flow, Boundary Mode**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 43 only.
5. Locate the **Linearized Potential Flow Equation Settings** section. In the m text field, type `m`.

Linearized Potential Flow Model 1

The following settings couple the aeroacoustics boundary mode to the background flow:

1. In the **Model Builder** window, expand the **Linearized Potential Flow, Boundary Mode (aebm)** node, then click **Linearized Potential Flow Model 1**.
2. In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.
3. From the ρ_0 list, choose **Density (cpf)**.
4. From the c_0 list, choose **Speed of sound (cpf/cpf1)**.
5. Specify the $\mathbf{V}$ vector as

`cpf.Vr`	`r`
`cpf.Vz`	`z`

6. In the **Model Builder** window, collapse the **Linearized Potential Flow, Boundary Mode (aebm)** node.
7. In the **Model Builder** window, collapse the **Linearized Potential Flow, Boundary Mode (aebm)** node.

Set up the solver. Step 1 solves the stationary background field and Step 2 is the boundary mode analysis. The solution of the stationary step (the background flow) is automatically used in the mode analysis step.

STUDY 1

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

2 In the **Settings** window for **Study**, type `Study 1 - Background and Source` in the **Label** text field.

STUDY 1 - BACKGROUND AND SOURCE

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1 - Background and Source** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the following interfaces:

Physics interface
Linearized Potential Flow, Boundary Mode (aebm) {aebm}
Linearized Potential Flow, Frequency Domain (ae) {ae}
Linearized Potential Flow, Frequency Domain 2 (ae2) {ae2}

Step 2: Mode Analysis

1 On the **Study** toolbar, click **Study Steps** and choose **Other>Mode Analysis**.

2 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.

3 From the **Transform** list, choose **Out-of-plane wave number**.

4 In the **Mode analysis frequency** text field, type `f`.

5 Select the **Desired number of modes** check box.

6 In the associated text field, type `1`.

7 From the **Mode search method around shift** list, choose **Larger real part**.

8 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the following interfaces:

Physics interface
Compressible Potential Flow (cpf) {cpf}
Linearized Potential Flow, Frequency Domain (ae) {ae}
Linearized Potential Flow, Frequency Domain 2 (ae2) {ae2}

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

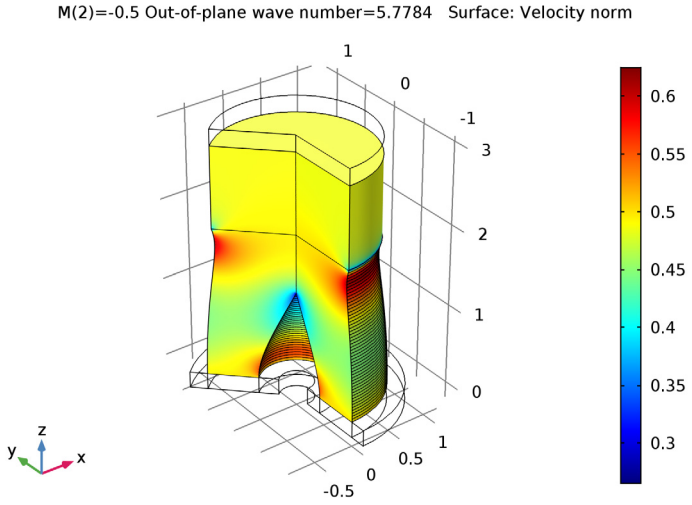
Parameter name	Parameter value list	Parameter unit
M (Mean flow Mach number)	0 -0.5	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Mean Flow Velocity, 3D (cpf)

The second default plot group is a 225 deg. revolution plot of the velocity potential (of the first plot group).



In the plot you can also inspect the value of the out-of-plane wave number. Setting up the solver you only looked for the first propagating mode for each value of the Mach number. For M = 0 you have 10.84 and for M = -0.5 you have 5.778. The strong background flow has shifted the wave number.

NOTE: If you want to investigate more modes you can change the settings in the Mode Analysis study step. Set the desired number of modes, for example 12, and then set the search to Closest in absolute value. Solve again. Now, inspecting the Out-of-plane wave number list (for M = -0.5) you will find four solutions with a purely real wave number, three of them positive and one negative. In other words, there are four propagating waves, three of which propagate in the positive z direction and one in the opposite direction. The

strong background flow has shifted the wave numbers, which in the absence of a mean flow would be symmetrically distributed around zero (if you select M = 0). If you had set these options you would have need to select the correct out-of-plane wave number in subsequent actions. Now you only have one wave number, the forward propagating one, for each M value.

To reproduce the plot shown in Figure 2 create a new 2D plot group.

2D Plot Group 5

1. On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2. In the **Settings** window for **2D Plot Group**, type `Stationary: rho and Phi` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Study 1 - Background and Source/ Parametric Solutions 1 (sol3)**.

Surface 1

1. Right-click **Stationary: rho and Phi** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `rho`.

Contour 1

1. In the **Model Builder** window, under **Results** right-click **Stationary: rho and Phi** and choose **Contour**.
2. On the **Stationary: rho and Phi** toolbar, click **Plot**.

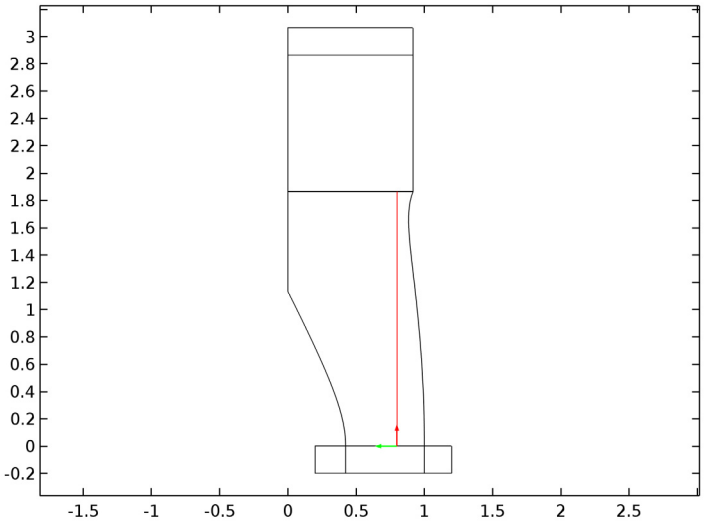
   Compare the result with Figure 2.

   Proceed with creating the figures in Figure 3 and Figure 4. First create a new dataset to get a detailed view of the density and velocity profiles along the length of the duct.

Cut Line 2D 1

1. On the **Results** toolbar, click **Cut Line 2D**.
2. In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.
3. In row **Point 1**, set **R** to `0.8`.
4. In row **Point 2**, set **R** to `0.8` and **z** to `1.86393`.
5. Locate the **Data** section. From the **Data set** list, choose **Study 1 - Background and Source/ Parametric Solutions 1 (sol3)**.
6. Click **Plot**.

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



1D Plot Group 6

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

2 In the **Settings** window for **1D Plot Group**, type `Stationary: rho and Mz` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

4 From the **Parameter selection (M)** list, choose **From list**.

5 In the **Parameter values (M)** list, select **-0.5**.

6 From the **Out-of-plane wave number selection** list, choose **First**.

7 Click to expand the **Legend** section. From the **Position** list, choose **Middle left**.

Line Graph 1

1 Right-click **Stationary: rho and Mz** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `rho`.

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

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

6 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

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

8 In the table, enter the following settings:

Legends
Density

Line Graph 2

1 Right-click **Results>Stationary: rho and Mz>Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `Mz`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Axial Mach Number

5 On the **Stationary: rho and Mz** toolbar, click **Plot**.

The resulting plot should closely resemble that in Figure 3.

1D Plot Group 7

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

2 In the **Settings** window for **1D Plot Group**, type `Boundary Mode Potential: phi_b` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1 - Background and Source/Parametric Solutions 1 (sol3)**.

4 From the **Out-of-plane wave number selection** list, choose **First**.

5 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Boundary Mode Potential: phi_b** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Select the **Active** toggle button.

4 Select Boundary 43 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `phi_b`.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 Find the **Include** subsection. In the **Prefix** text field, type `Mach number and out-of-plane wave number: ` .

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

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

10 On the **Boundary Mode Potential: phi_b** toolbar, click **Plot**.

The resulting plot of the desired boundary modes should closely resemble that in Figure 4. Note that the shapes of the two seem identical. Here the real part is plotted. Change the plot parameter to `imag(phi_p)` and you will see a difference.

The Acoustic Field

Equipped with the solution derived in the previous stage, you can now go on to simulate the acoustic field. Model the noise source through judicious choices of boundary conditions at the source plane (z = 0) for the two Linearized Potential Flow, Frequency Domain interfaces. Furthermore, implement non-reflecting boundary conditions at both ends of the duct geometry by using the auxiliary PML domains that you added to the model earlier in the geometry creation steps.

Here are the detailed instructions for the procedure.

DEFINITIONS

First, add an extrusion coupling that allows you to extend the compressible potential flow quantities rho and C to the PML domain outside the terminal plane (z = 2.86393).

General Extrusion 1 (genext1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.

2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.

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

4 Select Boundary 6 only.

5 Locate the **Destination Map** section. Clear the **z-expression** text field.

6 Locate the **Source** section. Select the **Use source map** check box.

7 Clear the **z?-expression** text field.

Clearing these text fields gives a one-dimensional map that only depends on the radial coordinate, r.

Also, create a maximum coupling operator for the duct domain and use it to define a normalized absolute pressure variable.

Maximum 1 (maxop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Maximum**.

2 Select Domain 1 only.

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Variables 1**.

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

3 In the table, enter the following settings:

Name	Expression	Description
pabsn	abs(ae.p)/comp1.maxop1(ae.p)	Normalized pressure

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 3 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

5 Locate the **Scaling** section. From the **Typical wavelength from** list, choose **User defined**.

6 In the **Typical wavelength** text field, type `ae.c0/f`.

Perfectly Matched Layer 2 (pml2)

1 Right-click **Perfectly Matched Layer 1 (pml1)** and choose **Duplicate**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 4–6 only.

5 Locate the **Scaling** section. In the **Typical wavelength** text field, type `ae2.c0/f`.

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN (AE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Frequency Domain (ae)**.

2 Select Domains 1–3 only.

3 In the **Settings** window for **Linearized Potential Flow, Frequency Domain**, click to expand the **Equation** section.

4 Locate the **Linearized Potential Flow Equation Settings** section. In the m text field, type `m`.

Linearized Potential Flow Model 1

Couple the mean-flow field to the aeroacosutic fields just like you did for the boundary mode.

1 In the **Model Builder** window, expand the **Linearized Potential Flow, Frequency Domain (ae)** node, then click **Linearized Potential Flow Model 1**.

2 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.

3 From the ρ_0 list, choose **Density (cpf)**.

4 From the c_0 list, choose **Speed of sound (cpf/cpfl)**.

5 Specify the **V** vector as

`cpf.Vr`	`r`
`cpf.Vz`	`z`

6 In the **Model Builder** window, click **Linearized Potential Flow, Frequency Domain (ae)**.

Linearized Potential Flow Model 2

1 On the **Physics** toolbar, click **Domains** and choose **Linearized Potential Flow Model**.

2 Select Domain 3 only.

3 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.

4 From the ρ_0 list, choose **User defined**. In the associated text field, type `genext1(rho)`.

5 From the c_0 list, choose **User defined**. In the associated text field, type `genext1(C)`.

6 Specify the **V** vector as

`0`	`r`
`genext1(cpf.Vz)`	`z`

Normal Mass Flow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Mass Flow**.

2 Select Boundary 43 only.

3 In the **Settings** window for **Normal Mass Flow**, locate the **Normal Mass Flow** section.

4 In the m_n text field, type `rho*(-aebm.ikz*A*phi_b)`.

Impedance 1

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

2 Select Boundaries 44–59 and 62–95 only.

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

4 In the Z_i text field, type `Zw/flc2hs(z/zi,b)`.

The reason behind using the smoothed Heaviside function `flc2hs` is to make the impedance a continuous (albeit abruptly changing) function across the interfaces between regions with and without an acoustic lining. This is a condition required for

the equivalence of Myers's original impedance boundary condition and its weak reformulation due to Eversman used here to hold (see Ref. 3).

Impedance 2

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

2 Select Boundary 60 only.

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

4 In the Z_i text field, type `Zw/flc2hs((zt-z)/(zt-zi),b)`.

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN 2 (AE2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Frequency Domain 2 (ae2)**.

2 Select Domains 4–6 only.

3 In the **Settings** window for **Linearized Potential Flow, Frequency Domain**, click to expand the **Equation** section.

4 Locate the **Linearized Potential Flow Equation Settings** section. In the m text field, type `m`.

Linearized Potential Flow Model 1

1 In the **Model Builder** window, expand the **Linearized Potential Flow, Frequency Domain 2 (ae2)** node, then click **Linearized Potential Flow Model 1**.

2 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.

3 From the ρ_0 list, choose **User defined**. In the associated text field, type `rho0`.

4 From the c_0 list, choose **User defined**. In the associated text field, type `c0`.

5 Specify the $\mathbf{V}$ vector as

0	r
M	z

Linearized Potential Flow Model 2

1 Right-click **Component 1 (comp1)>Linearized Potential Flow, Frequency Domain 2 (ae2)> Linearized Potential Flow Model 1** and choose **Duplicate**.

2 Select Domains 4 and 6 only.

3 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.

4 Specify the **V** vector as

0	r
0	z

Velocity Potential 1

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

2 Select Boundary 43 only.

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

4 In the ϕ_0 text field, type `phi-A*phi_b`.

5 In the **Model Builder** window, collapse the **Linearized Potential Flow, Frequency Domain 2 (ae2)** node.

6 In the **Model Builder** window, collapse the **Linearized Potential Flow, Frequency Domain 2 (ae2)** node.

Next compute the acoustic fields for the case with (M = -0.5) and without (M = 0) background flow as well as with and without the wall lining. Do this by adding four frequency domain studies, select the desired background flow solution, and enable or disable the impedance boundary condition using the **Modify physics tree and variables for study step** option in the solver step.

ADD STUDY

1 On 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 **Custom Studies**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Compressible Potential Flow (cpf)** and **Linearized Potential Flow, Boundary Mode (aebm)**.

5 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

6 Click **Add Study** in the window toolbar.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Frequency Domain**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Compressible Potential Flow (cpf)** and **Linearized Potential Flow, Boundary Mode (aebm)**.

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

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Frequency Domain**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Compressible Potential Flow (cpf)** and **Linearized Potential Flow, Boundary Mode (aebm)**.

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

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Frequency Domain**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Compressible Potential Flow (cpf)** and **Linearized Potential Flow, Boundary Mode (aebm)**.

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

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

STUDY 2

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

2 Clear the **Generate default plots** check box.

3 In the **Label** text field, type `Study 2 - Frequency Domain (M = 0, lined)`.

STUDY 2 - FREQUENCY DOMAIN (M = 0, LINED)

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - Frequency Domain (M = 0, lined)** 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 `f`.

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - Background and Source, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

8 From the **Use** list, choose **M=0 (sol4)**.

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

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 3 - Frequency Domain (M = -0.5, lined)`.

STUDY 3 - FREQUENCY DOMAIN (M = -0.5, LINED)

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3 - Frequency Domain (M = -0.5, lined)** 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 `f`.

4 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - Background and Source, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

8 From the **Use** list, choose **M=-0.5 (sol5)**.

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

STUDY 4

1 In the **Model Builder** window, click **Study 4**.

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 4 - Frequency Domain (M = 0, hard wall)`.

STUDY 4 - FREQUENCY DOMAIN (M = 0, HARD WALL)

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 4 - Frequency Domain (M = 0, hard wall)** 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 `f`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Linearized Potential Flow, Frequency Domain (ae)>Impedance 1**.

6 Click **Disable**.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Linearized Potential Flow, Frequency Domain (ae)>Impedance 2**.

8 Click **Disable**.

9 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

10 From the **Method** list, choose **Solution**.

11 From the **Study** list, choose **Study 1 - Background and Source, Mode Analysis**.

12 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

13 From the **Use** list, choose **M=0 (sol4)**.

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

STUDY 5

1 In the **Model Builder** window, click **Study 5**.

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

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 5 - Frequency Domain (M = -0.5, hard wall)`.

STUDY 5 - FREQUENCY DOMAIN (M = -0.5, HARD WALL)

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 5 - Frequency Domain (M = -0.5, hard wall)** 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 `f`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Linearized Potential Flow, Frequency Domain (ae)>Impedance 1**.

6 Click **Disable**.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Linearized Potential Flow, Frequency Domain (ae)>Impedance 2**.

8 Click **Disable**.

9 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

10 From the **Method** list, choose **Solution**.

11 From the **Study** list, choose **Study 1 - Background and Source, Mode Analysis**.

12 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

13 From the **Use** list, choose **M=-0.5 (sol5)**.

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

You now have all the necessary data sets to create the results depicted in Figure 5 and in Figure 6. Start by creating one plot and set it up, then simply duplicate the plot and select the correct data set. After generating the plots proceed to calculating the attenuation of the system under **Derived Values** in the **Results** node.

RESULTS

2D Plot Group 8

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Pressure: M = 0, lined` in the **Label** text field.

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

4 In the **Title** text area, type `Normalized Pressure (M = 0, lined)`.

5 Locate the **Data** section. From the **Data set** list, choose **Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)**.

Contour 1

1 Right-click **Pressure: M = 0, lined** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Expression** section.

3 In the **Expression** text field, type `pabsn`.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.0001 0.001 0.01 0.02 0.04 0.06 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9`.

6 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

Contour 2

1 In the **Model Builder** window, under **Results** right-click **Pressure: M = 0, lined** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Expression** section.

3 In the **Expression** text field, type `pabsn`.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.0001 0.001 0.01 0.02 0.04 0.06 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9`.

6 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

7 Clear the **Color legend** check box.

8 On the **Pressure: M = 0, lined** toolbar, click **Plot**.

The plot does not look exactly like the one at the top of Figure 5, this is because only the solution inside the flow duct is of interest. Add selections to the data sets to restrict plotting to this domain.

Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)

In the **Model Builder** window, under **Results>Data Sets** click **Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domain 1 only.

Study 3 - Frequency Domain (M = -0.5, lined)/Solution 7 (sol7)

In the **Model Builder** window, under **Results>Data Sets** click **Study 3 - Frequency Domain (M = -0.5, lined)/Solution 7 (sol7)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domain 1 only.

Study 4 - Frequency Domain (M = 0, hard wall)/Solution 8 (sol8)

In the **Model Builder** window, under **Results>Data Sets** click **Study 4 - Frequency Domain (M = 0, hard wall)/Solution 8 (sol8)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domain 1 only.

Study 5 - Frequency Domain (M = -0.5, hard wall)/Solution 9 (sol9)

In the **Model Builder** window, under **Results>Data Sets** click **Study 5 - Frequency Domain (M = -0.5, hard wall)/Solution 9 (sol9)**.

Selection

1 On the **Results** toolbar, click **Selection**.

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

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

4 Select Domain 1 only.

Pressure: M = 0, lined

1 In the **Model Builder** window, under **Results** click **Pressure: M = 0, lined**.

2 On the **Pressure: M = 0, lined** toolbar, click **Plot**.

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

This should look like the lower plot in Figure 5.

Pressure: M = 0, lined 1

1 Right-click **Results>Pressure: M = 0, lined** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Pressure: M = -0.5, lined` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Normalized Pressure (M = -0.5, lined)`.

4 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Frequency Domain (M = -0.5, lined)/Solution 7 (sol7)**.

5 On the **Pressure: M = -0.5, lined** toolbar, click **Plot**.

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

This should look like the lower plot in Figure 6.

Pressure: M = -0.5, lined 1

1 Right-click **Results>Pressure: M = -0.5, lined** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Pressure: M = 0, hard wall` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Normalized Pressure (M = 0, hard wall)`.

4 Locate the **Data** section. From the **Data set** list, choose **Study 4 - Frequency Domain (M = 0, hard wall)/Solution 8 (sol8)**.

5 On the **Pressure: M = 0, hard wall** toolbar, click **Plot**.

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

This should look like the upper plot in Figure 5.

Pressure: M = 0, hard wall 1

1 Right-click **Results>Pressure: M = 0, hard wall** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Pressure: M = -0.5, hard wall` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Normalized Pressure (M = -0.5, hard wall)`.

4 Locate the **Data** section. From the **Data set** list, choose **Study 5 - Frequency Domain (M = -0.5, hard wall)/Solution 9 (sol9)**.

5 On the **Pressure: M = -0.5, hard wall** toolbar, click **Plot**.

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

This should look like the upper plot in Figure 6.

Finally, calculate the attenuation. But first, determine the power at the source and the duct inlet. These numerical values are used to evaluate the attenuation using the attenuation function `dw` you have created.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: W_src (M = 0)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1 - Background and Source/ Parametric Solutions 1 (sol3)**.

4 From the **Parameter selection (M)** list, choose **First**.

5 From the **Out-of-plane wave number selection** list, choose **First**.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`W_src`		`Power through source plane`

7 Click **Evaluate**.

Global Evaluation: W_src (M = 0) 1

1 Right-click **Global Evaluation: W_src (M = 0)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: W_src (M = -0.5)` in the **Label** text field.

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

4 Click the small triangle in the **Settings** window toolbar and choose **New Table** from the menu.

Global Evaluation 3

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: W_inl (M = 0, lined)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`W_inl`		`Power through inlet plane`

5 Click the small triangle in the **Settings** window toolbar and choose **New Table** from the menu.

Global Evaluation: W_inl (M = 0, lined) 1

1 Right-click **Global Evaluation: W_inl (M = 0, lined)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: W_inl (M = -0.5, lined)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Frequency Domain (M = -0.5, lined)/Solution 7 (sol7)**.

4 Click the small triangle in the **Settings** window toolbar and choose **New Table** from the menu.

Global Evaluation 5

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: Attenuation (M = 0, lined)` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 - Frequency Domain (M = 0, lined)/Solution 6 (sol6)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`comp1.dw(0.02796, 2.43e-7)`		

5 Click **Evaluate**.

The result should be approximately 50.6 dB, which is quite close to that in Ref. 1 (51.6 dB). The reason is that both use the same source mode as a result of the fact that there is a single forward-propagating mode in the flow-free case, thus making the two calculations directly comparable.

Global Evaluation: Attenuation (M = 0, lined) 1

1 Right-click **Global Evaluation: Attenuation (M = 0, lined)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `Global Evaluation: Attenuation (M = -0.5, lined)` in the **Label** text field.

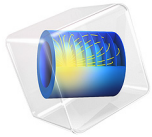
3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`comp1.dw(9.5194e-3, 2.8650e-5)`		

4 Click the small triangle in the **Settings** window toolbar and choose **New Table** from the menu.

The result should now be roughly to 25.2 dB. Compare this result to the value of approximately 27 dB obtained for the corresponding quantity in Ref. 1. In contrast to that paper, the source mode used in these calculations was derived for the case of a hard

duct wall, whereas the lowest mode for the lined-wall case would be a linear combination of the two forward-propagating hard-wall modes. For this reason, the noise source is not an eigenmode and is, consequently, not maximally attenuated.



Ultrasonic Flow Meter with Piezoelectric Transducers: Coupling between FEM and DG

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Ultrasonic flow meters are used to determine the velocity of the fluid flowing through a pipe. The principle is to send an ultrasonic signal across the flow at a skew angle. In case of no flow, the transmitting time between the transmitter and the receiver is the same for the signals sent in the upstream and the downstream directions. Otherwise, the downstream traveling wave moves faster than the one traveling upstream. In many cases piezoelectric transducers are used to send and receive the ultrasonic wave.

This tutorial example shows how to simulate an ultrasonic flow meter with piezoelectric transducers in the simplified no-flow case. The finite element method (FEM) is used to model the piezoelectric transducers, whereas the modeling of the ultrasonic wave propagation is based on the discontinuous Galerkin (DG) method. The whole model is split into three sub-models. A FEM to DG one-way coupling is used to send the wave from the transmitter, and a DG to FEM one-way coupling is used for the receiver.

Model Definition

The flow meter geometry used in this model is the same as given in the model Ultrasound Flow Meter with Generic Time-of-Flight Configuration. It consists of a main pipe and a signal pipe of a smaller diameter. The signal tube is tilted to the main pipe at the angle $\alpha = 45^{o}$. Two piezoelectric transducers placed at either end of the signal pipe operate as the transmitter and the receiver. An input voltage signal applied to the transmitter results in the mechanical deformation of the piezoelectric transducer, which is due to the inverse piezoelectric effect. The mechanical deformation in its turn generates an acoustic wave in the fluid. When the acoustic wave reaches the receiver, the inverse process takes place: the mechanical load is being converted into an electric signal because of the direct piezoelectric effect.

As previously mentioned, this model studies the propagation of the acoustic wave is the simplified no-flow condition. Thus the main focus is the interaction between the fluid and the solid and the conversion *input electric signal - acoustic wave - output electric signal*. The background flow can be added if necessary, as it is done in the model Ultrasound Flow Meter with Generic Time-of-Flight Configuration.

This model solves the full three-dimensional problem with the symmetry condition imposed on the sagittal plane of the structure. The model involves the following physical phenomena.

PIEZOELECTRICITY: SOLID MECHANICS AND ELECTROSTATICS

The transmitter and the receiver can be used interchangeably; therefore, the principles of their construction are the same. The main part is a disk made of a piezoelectric material (for example, PZT) that is used for conversion of between the electric and the mechanical waves.

A direct propagation of the mechanical wave in the piezoelectic transducer in form of the acoustic wave in the fluid will result in significant reflections from the solid/fluid interface and, consequently, losses. This is due to the impedance mismatch: for water, the characteristic acoustic impedance $Z_{\text{water}} \approx 1.5 \times 10^6$ kg/m^2·s, and for PZT-5H, $Z_{\text{PZT}} \approx 30 \times 10^6$ kg/m^2·s. Therefore, a matching layer is required to minimize the losses. Its size is taken to be 1/4 of the wavelength, and its impedance is calculated as follows Ref. 1

$$Z_{\text{match}} = \sqrt{Z_1 Z_2}\,. \tag{1}$$

The impedance matching itself is a problem that requires a though investigation and is out of scope of this model. Here, the acrylic plastic is used as the matching layer material.

The input signal applied to the transmitter is a harmonic voltage pulse of the amplitude $V_0 = 50$ V, the frequency $f_0 = 2.5$ MHz, and duration of 2 µs. The voltage profile is depicted in Figure 1.

ACOUSTICS: CONVECTED WAVE EQUATION, TIME EXPLICIT

The vibrations of the transmitter generate an ultrasonic acoustic wave traveling from the transmitter to the receiver. The propagation of the acoustic wave is defined by the normal velocity boundary condition prescribed on the interface between the matching layer of the transmitter and the fluid domain. The velocity components are derived from the submodel that simulates the transmitter.

When the acoustic signal reaches the other end of the signal tube, it will interact with the receiver. The acoustic pressure imposed on the interface between the fluid domain and the matching layer of the receiver will be used as the boundary condition in the receiver submodel.

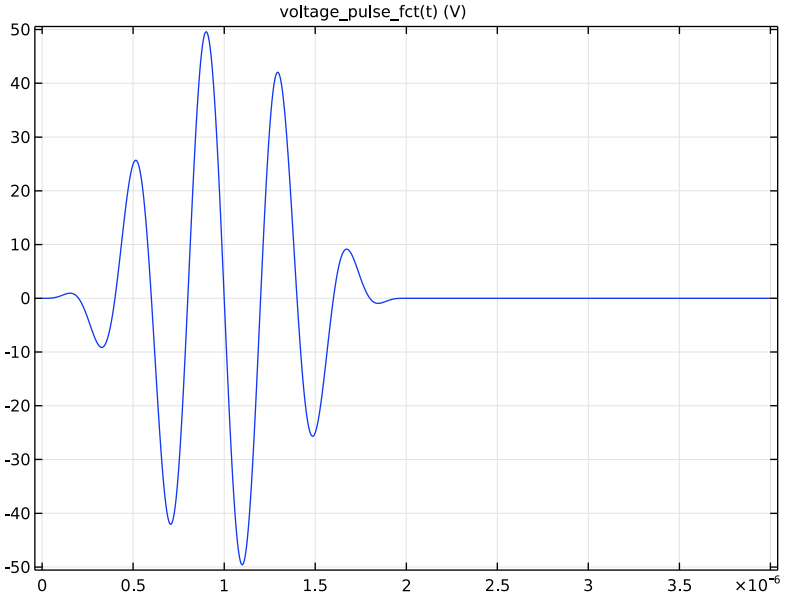


Figure 1: Input voltage applied to the transmitter.

Results and Discussion

The propagation of the ultrasonic acoustic wave near the transmitter at time $t = 2.4$ µs is shown in Figure 2. The driving voltage applied to the transmitter is that depicted in Figure 1.

The propagation of the ultrasonic pressure wave over the symmetry plane of the flow meter is depicted in Figure 3. Note that the upper left plot presents the full profile of the signal shown in Figure 2.

By the time $t = 10$ µs the pressure signal has reached the receiver. Figure 4 shows the mechanical deformation profile of the receiver. In Figure 5 you can see the profiles of the input voltage applied to the transducer and the output electric signal read on the receiver. As previously mentioned, the time of flight will be different for the signal propagating downstream and the one propagating upstream in the presence of a background flow.

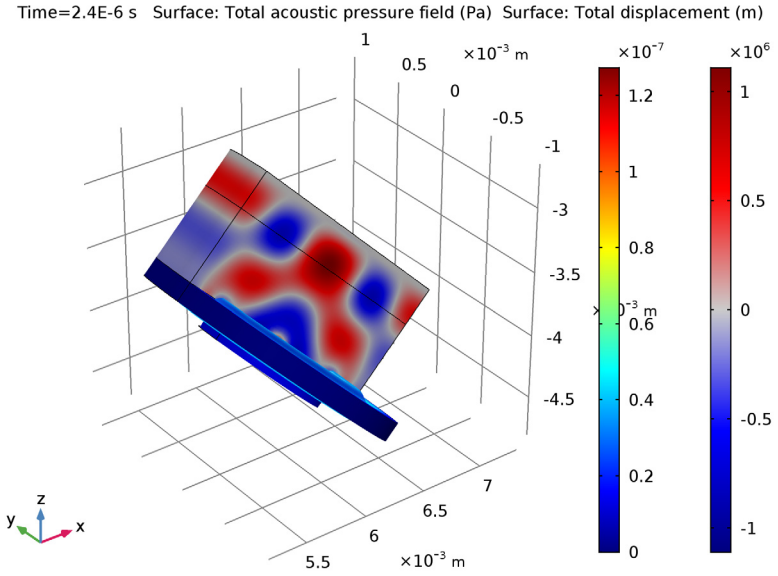


Figure 2: Propagation of the ultrasonic acoustic wave from the transmitter.

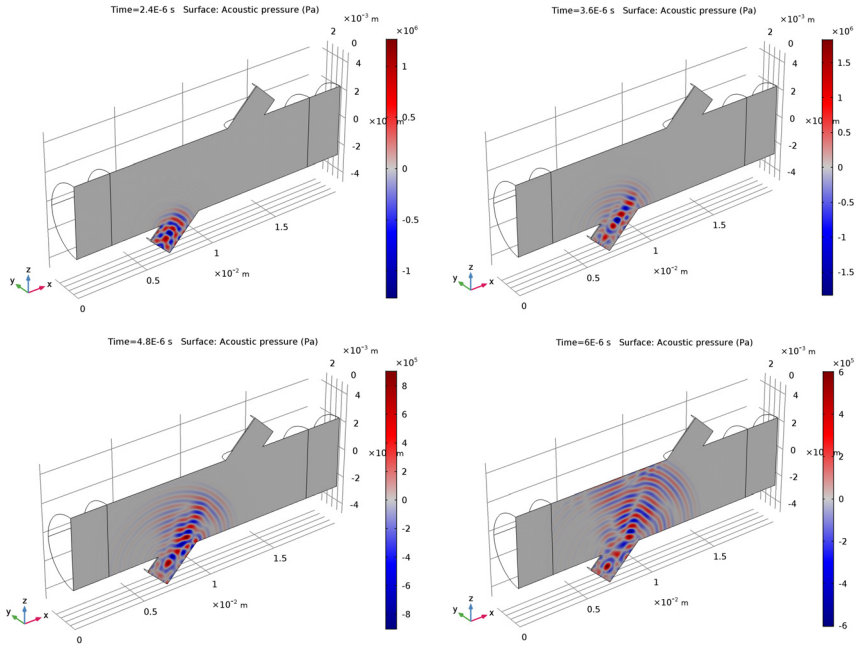


Figure 3: Propagation of the acoustic pressure signal at 4 time steps.

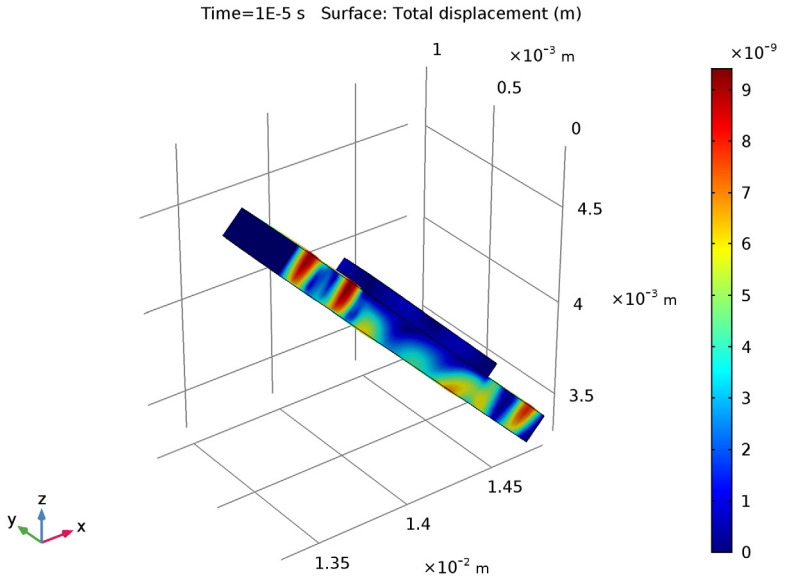


Figure 4: Mechanical deformation of the receiver.

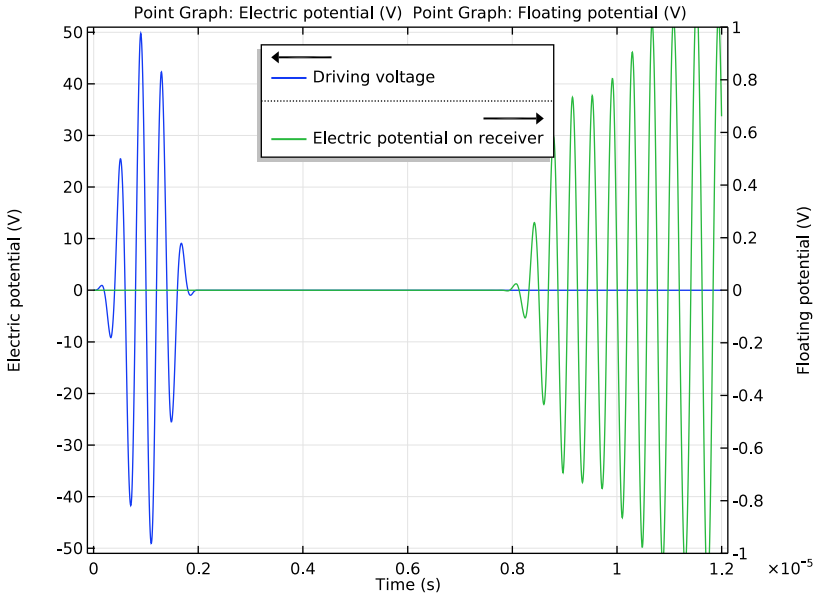


Figure 5: Input and output electric signals as functions of time.

Notes About the COMSOL Implementation

The described split of the entire model into three submodels has a certain reason. The parts that model the piezoelectric transducers use the conventional FEM, whereas the ultrasonic wave propagation is modeled by the DG method. The time-dependent solvers for both approaches are different, which makes the direct FEM to DG or DG to FEM couplings impossible.

The workaround discussed in this tutorial is based on the indirect one-way coupling between the models that use different numerical schemes. This is done through the identity mapping coupling operators `idmap1` and `idmap2`. These are used on the transmitter-fluid and the fluid-receiver interfaces, respectively. Thus the operators map the results between the submodels and provide the required couplings.

The following points of the submodels' setups must be taken into account:

- The transmitter submodel includes the domains for the piezoelectric transducer, the matching layer, and the fluid layer. The last is located on top of the matching layer and is used to account the attached fluid mass providing the fluid-structure interaction. The

Pressure Acoustics, Transient interface solves for the pressure wave in the fluid. The fluid domain is truncated by a perfectly matched layer. Note that this approach remains appropriate even in the case of a background flow, because the considered fluid domain is occupies only a small part of the sensing tube adjacent to the transmitter. Therefore, the interaction between the fluid attached to the matching layer and the background flow in the main pipe is negligible.

- In order to account for the interaction between the ultrasonic pressure wave and the receiver, an impedance boundary condition is imposed on the fluid-receiver interface in a Convected Wave Equation, Time Explicit interface.

Reference

1. V. A. Gavrilova, M. G. Fazlyyyakhmatov, and N. F. Kashapov, "Protective matching polymer powder coating of piezoelectric element," *J. Phys.: Conf. Ser.*, vol. 479, issue 1, 2013.

Application Library path: `Acoustics_Module/Ultrasound/flow_meter_piezoelectric_transducers`

Modeling Instructions

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

NEW

In the **New** window, click **Blank Model**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `flow_meter_piezoelectric_transducers_parameters.txt`.

Specify the driving voltage applied to the transmitter.

Rectangle 1 (rect1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Rectangle**.

2 In the **Settings** window for **Rectangle**, type `rect_fct` in the **Function name** text field.

3 Locate the **Parameters** section. In the **Lower limit** text field, type `0.5e-6`.

4 In the **Upper limit** text field, type `1.5e-6`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1e-6`.

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `voltage_pulse_fct` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `V0*sin(omega0*t)*rect_fct(t)`.

4 In the **Arguments** text field, type `t`.

5 Locate the **Units** section. In the **Arguments** text field, type `s`.

6 In the **Function** text field, type `V`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
t	0	10*T0

8 Click **Plot**.

The input electric signal should look like the one in Figure 1.

The entire model is split into submodels. First, create a model that simulates the propagation of the acoustic signal induced by the input signal applied to the transmitter.

ROOT

1 On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

The geometry will contain the domains corresponding to the piezoelectric transducer, the matching layer, and a layer of the fluid in the signal tube. The last is used to account for the attached fluid mass and model the fluid-structure interaction.

GEOMETRY 1

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `D_transducer/2`.

4 In the **Height** text field, type `lambda0+L_pml`.

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `-L_transducer/2`.

7 In the **Settings** window for **Cylinder**, click to expand the **Layers** section.

8 In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`L_pml`

9 Clear the **Layers on side** check box.

10 Select the **Layers on top** check box.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `D_transducer/2`.

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

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `-L_transducer/2-L_matching`.

Cylinder 3 (cyl3)

1 On the **Geometry** toolbar, click **Cylinder**.

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

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

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

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `-L_transducer/2-L_matching-L_piezo`.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.

3 In the **Rotation** text field, type `alpha`.

4 Locate the **Point on Axis of Rotation** section. In the **x** text field, type `L/2`.

5 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **y-axis**.

6 Click in the **Graphics** window and then press Ctrl+A to select all objects.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Partition Objects 1 (par1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

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

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

Delete Entities 1 (del1)

1 Right-click **Partition Objects 1 (par1)** and choose **Delete Entities**.

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

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

4 On the object **par1(2)**, select Domain 1 only.

5 On the object **par1(3)**, select Domain 1 only.

6 On the object **par1(1)**, select Domains 1 and 3 only.

7 On the **Geometry** toolbar, click **Build All**.

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

ADD MATERIAL

1 On 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>Water, liquid**.

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

ADD MATERIAL

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

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

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

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat2)
Select Domain 3 only.

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **Built-In>Acrylic plastic**.
3. Click **Add to Component** in the window toolbar.
4. On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Acrylic plastic (mat3)
Select Domain 1 only.

ADD PHYSICS

1. On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.
4. Click **Add to Component** in the window toolbar.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

Select Domains 2 and 4 only.

ADD PHYSICS

1. Go to the **Add Physics** window.
2. In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3. Click **Add to Component** in the window toolbar.

SOLID MECHANICS (SOLID)

Select Domains 1 and 3 only.

ADD PHYSICS

1. Go to the **Add Physics** window.
2. In the tree, select **AC/DC>Electrostatics (es)**.
3. Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ELECTROSTATICS (ES)

Select Domain 3 only.

ADD STUDY

1 On 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 **Preset Studies> Time Dependent**.

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

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

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundary 6 only.

5 In the **Label** text field, type `Transmitter`.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 Select Domains 1 and 3 only.

3 In the **Settings** window for **Explicit**, type `Solid` in the **Label** text field.

The fluid domain is truncated by introducing a perfectly matched layer (PML) which absorbs the propagating acoustic waves in the fluid.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 4 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

Define a coordinate system that corresponds to the piezoelectric material orientation: the z-axis of the piezoelectric crystal points along the signal tube axis.

Base Vector System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.

2 In the **Settings** window for **Base Vector System**, type `sys_PZT` in the **Label** text field.

3 Locate the **Settings** section. Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

4 Find the **Base vectors** subsection. In the table, enter the following settings:

	x	y	z
x1	`cos(pi/4)`	`0`	`-sin(pi/4)`
x3	`sin(pi/4)`	`0`	`cos(pi/4)`

Create a mapping operator that will be used to map the results between the submodels.

Identity Mapping 1 (idmap1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Identity Mapping**.

2 In the **Settings** window for **Identity Mapping**, locate the **Source Selection** section.

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

4 Select Boundary 6 only.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

Modify the **Typical Wave Speed for Perfectly Matched Layers** and **Transient Solver Settings** according to the fluid material and the frequency of the input signal. These settings are used to adjust the PML and the time-dependent solver settings.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient (actd)**.

2 In the **Settings** window for **Pressure Acoustics, Transient**, locate the **Typical Wave Speed for Perfectly Matched Layers** section.

3 In the c_{ref} text field, type `c0`.

4 Locate the **Transient Solver Settings** section. In the **Maximum frequency to resolve** field enter `f0`.

5 In the **Model Builder** window, click **Pressure Acoustics, Transient (actd)**.

Symmetry 1

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

2 Select Boundaries 5 and 12 only.

SOLID MECHANICS (SOLID)

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

Piezoelectric Material 1

1. On the **Physics** toolbar, click **Domains** and choose **Piezoelectric Material**.
2. Select Domain 3 only.
3. In the **Settings** window for **Piezoelectric Material**, locate the **Coordinate System Selection** section.
4. From the **Coordinate system** list, choose **sys_PZT (sys2)**.

Mechanical Damping 1

1. On the **Physics** toolbar, click **Attributes** and choose **Mechanical Damping**.
2. In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.
3. From the **Damping type** list, choose **Rayleigh damping**.
4. In the α_{dM} text field, type `alpha_dmp`.
5. In the β_{dK} text field, type `beta_dmp`.

Fixed Constraint 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2. Select Boundaries 2 and 15 only.

Symmetry 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundaries 1 and 8 only.

Roller 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Roller**.
2. Select Boundary 9 only.

ELECTROSTATICS (ES)

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

Charge Conservation, Piezoelectric 1

1. On the **Physics** toolbar, click **Domains** and choose **Charge Conservation, Piezoelectric**.
2. Select Domain 3 only.

Ground 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Ground**.
2. Select Boundary 9 only.

Electric Potential 1

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

2 Select Boundary 10 only.

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

4 In the V_0 text field, type `voltage_pulse_fct(t)`.

COMPONENT 1 (COMP1)

On the **Home** toolbar, click **Windows** and choose **Add Multiphysics**.

ADD MULTIPHYSICS

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

2 In the tree, select **Acoustics>Acoustic-Structure Interaction>Acoustic-Piezoelectric Interaction, Transient**.

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

4 On the **Home** toolbar, click **Add Multiphysics**.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 3 and 10 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 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. Select the **Maximum element size** check box.

5 In the associated text field, type `lambda0/6`.

6 Click **Build Selected**.

Swept 1

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

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

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

4 Select Domain 3 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `2`.

Swept 2

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

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

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

4 Select Domain 1 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 2** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `4`.

Swept 3

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

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

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

4 Select Domain 2 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 3** 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. Select the **Maximum element size** check box.

5 In the associated text field, type `lambda0/6`.

Distribution 1

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

2 Right-click **Swept 4** and choose **Distribution**.

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

4 In the **Number of elements** text field, type `6`.

5 Click **Build All**.

COMPONENT 1 (COMP1)

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

ROOT

1 On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

Create the geometry that contains the main pipe, the signal tube, and the receiver. As the transmitter, the receiver contains a piezoelectric transducer and a matched layer.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `D/2`.

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

5 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`0.5*D`

7 Clear the **Layers on side** check box.

8 Select the **Layers on bottom** check box.

9 Select the **Layers on top** check box.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `D_transducer/2`.

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

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `-L_transducer/2`.

Cylinder 3 (cyl3)

1 On the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `D_transducer/2`.

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

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `L_transducer/2`.

Cylinder 4 (cyl4)

1 On the **Geometry** toolbar, click **Cylinder**.

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

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

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

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `L_transducer/2+L_matching`.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **cyl2**, **cyl4**, and **cyl3** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.

4 In the **Rotation** text field, type `alpha`.

5 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **y-axis**.

6 Locate the **Point on Axis of Rotation** section. In the **x** text field, type `L/2`.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

Partition Objects 1 (par1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

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

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 From the **Work plane** list, choose **Work Plane 1 (wp1)**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 2** and choose **Delete Entities**.

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

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

4 On the object **par1(1)**, select Domains 1, 3, and 5 only.

5 On the object **par1(2)**, select Domain 1 only.

6 On the object **par1(3)**, select Domain 1 only.

7 On the object **par1(4)**, select Domain 1 only.

Form Composite Domains 1 (cmd1)

1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 On the object **fin**, select Domains 2–5 only.

3 On the **Geometry** toolbar, click **Build All**.

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

ADD MATERIAL

1 On 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>Water, liquid**.

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

ADD MATERIAL

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

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

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

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat5)

Select Domain 4 only.

ADD MATERIAL

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

2 In the tree, select **Built-In>Acrylic plastic**.

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

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

MATERIALS

Acrylic plastic (mat6)
Select Domain 3 only.

COMPONENT 2 (COMP2)

On the **Home** toolbar, click **Windows** and choose **Add Physics**.

ADD PHYSICS

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

2 In the tree, select **Acoustics>Ultrasound>Convected Wave Equation, Time Explicit (cwe)**.

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1	

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

CONVECTED WAVE EQUATION, TIME EXPLICIT (CWE)

Select Domains 1, 2, and 5 only.

ADD PHYSICS

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

2 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1	

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

SOLID MECHANICS 2 (SOLID2)

Select Domains 3 and 4 only.

ADD PHYSICS

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

2 In the tree, select **AC/DC>Electrostatics (es)**.

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1	

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

ELECTROSTATICS 2 (ES2)

1 Select Domain 4 only.

2 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ROOT

On the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Time Dependent**.

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Pressure Acoustics, Transient (actd)	
Solid Mechanics (solid)	
Electrostatics (es)	
Solid Mechanics 2 (solid2)	
Electrostatics 2 (es2)	

4 Find the **Multiphysics couplings in study** subsection. In the table, enter the following settings:

Multiphysics couplings	Solve
Acoustic-Structure Boundary 1 (asb1)	
Piezoelectric Effect 1 (pze1)	

5 Click **Add Study** in the window toolbar.

ADD STUDY

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

2 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Pressure Acoustics, Transient (actd)	
Solid Mechanics (solid)	
Electrostatics (es)	
Convected Wave Equation, Time Explicit (cwe)	

3 Find the **Multiphysics couplings in study** subsection. In the table, enter the following settings:

Multiphysics couplings	Solve
Acoustic-Structure Boundary 1 (asb1)	
Piezoelectric Effect 1 (pze1)	

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

STUDY 3

Step 1: Time Dependent

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

DEFINITIONS

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundaries 2, 6, and 25 only.

5 In the **Label** text field, type `Symmetry`.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundary 10 only.

5 In the **Label** text field, type `Source`.

Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundary 16 only.

5 In the **Label** text field, type `Receiver`.

Explicit 6

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundaries 3, 4, 7–9, 11–13, 26, and 27 only.

5 In the **Label** text field, type `Walls`.

Absorbing Layer 1 (ab1)

1 On the **Definitions** toolbar, click **Absorbing Layer**.

Add absorbing layers (sponge layers) to truncate the computational domain.

2 Select Domains 1 and 5 only.

Identity Mapping 2 (idmap2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Identity Mapping**.

2 In the **Settings** window for **Identity Mapping**, locate the **Source Selection** section.

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

4 Select Boundary 16 only.

Again, create a coordinate system specifying the piezoelectric material orientation.

Base Vector System 4 (sys4)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.

2 In the **Settings** window for **Base Vector System**, type `sys_PZT` in the **Label** text field.

3 Locate the **Settings** section. Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

4 Find the **Base vectors** subsection. In the table, enter the following settings:

	x	y	z
x1	`cos(pi/4)`	0	`-sin(pi/4)`
x3	`sin(pi/4)`	0	`cos(pi/4)`

CONVECTED WAVE EQUATION, TIME EXPLICIT (CWE)

In the **Model Builder** window, under **Component 2 (comp2)** click **Convected Wave Equation, Time Explicit (cwe)**.

Symmetry 1

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

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

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

Acoustic Impedance 1

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

2 Select Boundaries 1 and 28 only.

Acoustic Impedance 2

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

On the receiver-fluid interface, specify the characteristic acoustic impedance of the matching layer.

2 In the **Settings** window for **Acoustic Impedance**, locate the **Boundary Selection** section.

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

4 Locate the **Acoustic Impedance** section. In the Z text field, type `Z_matching`.

Normal Velocity 1

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

On the transmitter-fluid interface, specify the velocity obtained from the transmitter submodel.

2 In the **Settings** window for **Normal Velocity**, locate the **Boundary Selection** section.

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

4 Locate the **Normal Velocity** section. From the **Type** list, choose **Velocity**.

5 Specify the $\mathbf{v}_b(t)$ vector as

`comp1.idmap1(withsol('sol1', ut, setval(t, t)))`	x
`comp1.idmap1(withsol('sol1', vt, setval(t, t)))`	y
`comp1.idmap1(withsol('sol1', wt, setval(t, t)))`	z

SOLID MECHANICS 2 (SOLID2)

In the **Model Builder** window, under **Component 2 (comp2)** click **Solid Mechanics 2 (solid2)**.

Fixed Constraint 1

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

2 Select Boundaries 14 and 22 only.

Symmetry 1

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

2 Select Boundaries 15 and 19 only.

Roller 1

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

2 Select Boundary 21 only.

Boundary Load 1

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

2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.

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

4 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

5 In the p text field, type `idmap2(withsol('sol2', p2, setval(t, t)))`.

Piezoelectric Material 1

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

2 Select Domain 4 only.

3 In the **Settings** window for **Piezoelectric Material**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **sys_PZT (sys4)**.

Mechanical Damping 1

1 On the **Physics** toolbar, click **Attributes** and choose **Mechanical Damping**.

2 In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.

3 From the **Damping type** list, choose **Rayleigh damping**.

4 In the α_{dM} text field, type `alpha_dmp`.

5 In the β_{dK} text field, type `beta_dmp`.

ELECTROSTATICS 2 (ES2)

In the **Model Builder** window, under **Component 2 (comp2)** click **Electrostatics 2 (es2)**.

Ground 1

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

2 Select Boundary 21 only.

Floating Potential 1

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

2 Select Boundary 20 only.

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

4 Select the **Floating potential group** check box.

Charge Conservation, Piezoelectric 1

1 On the **Physics** toolbar, click **Domains** and choose **Charge Conservation, Piezoelectric**.

2 Select Domain 4 only.

MULTIPHYSICS

Piezoelectric Effect 2 (pze2)

On the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Piezoelectric Effect**.

MESH 2

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Mesh 2** and choose **Free Tetrahedral**.

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

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

4 Select Domains 1, 2, and 5 only.

When modeling using physics interfaces that are based on the DG method, it is important to avoid small mesh elemnts, since they control the time steps taken by the solver. In order to avoid this use the **Element Quality Optimization** functionality available for the tetrahedral mesh. This step is very important.

5 Click to expand the **Element quality optimization** section. Locate the **Element Quality Optimization** section. Select the **Avoid too small elements** check box.

6 From the **Optimization level** list, choose **High**.

Size

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Size**.

2 In the **Settings** window for **Size**, click to expand the **Element size parameters** section.

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

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambda0/1.5`.

5 In the **Minimum element size** text field, type `lambda0/3`.

6 In the **Model Builder** window, click **Mesh 2**.

7 In the **Settings** window for **Mesh**, click **Build All**.

Mesh 3

On the **Mesh** toolbar, click **Add Mesh**.

MESH 3

Free Triangular 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Meshes** right-click **Mesh 3** and choose **More Operations>Free Triangular**.

2 Select Boundaries 17 and 20 only.

Size 1

1 Right-click **Component 2 (comp2)>Meshes>Mesh 3>Free Triangular 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. Select the **Maximum element size** check box.

5 In the associated text field, type `lambda0/6`.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 3** and choose **Swept**.

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

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

4 Select Domain 4 only.

Distribution 1

1 Right-click **Component 2 (comp2)>Meshes>Mesh 3>Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `2`.

Swept 2

1 In the **Model Builder** window, right-click **Mesh 3** and choose **Swept**.

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

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

4 Select Domain 3 only.

Distribution 1

1 Right-click **Component 2 (comp2)>Meshes>Mesh 3>Swept 2** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `4`.

4 In the **Model Builder** window, click **Mesh 3**.

5 In the **Settings** window for **Mesh**, click **Build All**.

COMPONENT 2 (COMP2)

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

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.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, type `Transmitter` in the **Label** text field.

3 Locate the **Study Settings** section. In the **Times** text field, type `range(0,T0/5,30*T0)`.

4 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Multiphysics couplings	Solve for
Piezoelectric Effect 2 (pze2) {pze2}	

5 Click to expand the **Mesh selection** section.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

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

3 Clear the **Generate default plots** check box.

4 In the **Model Builder** window, click **Step 1: Time Dependent**.

5 In the **Settings** window for **Time Dependent**, type `Propagation of Signal` in the **Label** text field.

6 Locate the **Study Settings** section. In the **Times** text field, type `range(0,T0/5,30*T0)`.

7 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Multiphysics couplings	Solve for
Piezoelectric Effect 2 (pze2) {pze2}	

8 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

9 Under **Selections**, click **Add**.

Store the results on the structure-fluid interfaces and the symmetry plane only.

10 In the **Add** dialog box, in the **Selections** list, choose **Symmetry**, **Source**, and **Receiver**.

11 Click **OK**.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

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

3 Clear the **Generate default plots** check box.

4 In the **Model Builder** window, click **Step 1: Time Dependent**.

5 In the **Settings** window for **Time Dependent**, type `Receiver` in the **Label** text field.

6 Locate the **Study Settings** section. In the **Times** text field, type `range(0,T0/5,30*T0)`.

7 From the **Tolerance** list, choose **User controlled**.

8 In the **Relative tolerance** text field, type `1e-4`.

9 Locate the **Mesh Selection** section. In the table, enter the following settings:

Geometry	Mesh
Geometry 1 {geom1}	No mesh
Geometry 2 {geom2}	Mesh 3

Since this multiphysics problem involves a transient acoustics interface, the **Transient Solver Settings** defined for the acoustics interface will be automatically used when solving the coupled problem. This ensures optimal solver configuration for this piezoacoustic problem solved in the time domain.

STUDY 1

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

STUDY 2

1 Click **Compute**.

The default solver settings generated for Study 3 will be different from those in Study 1. This is because the Pressure Acoustics, Transient interface is absent here. It is required to change the default solver settings to **Manual** and specify the time step explicitly.

STUDY 3

Solution 3 (sol3)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `1/(60*f0)`.

This expression is the same as the one generated in Study 1.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Mirror 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

Create a plot that shows the propagating acoustic signal induced by the driving voltage applied to the transmitter. The result should look as in Figure 2.

3D Plot Group 1

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

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

3 Locate the **Data** section. From the **Time (s)** list, choose **2.4E-6**.

Surface 1

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

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

3 From the **Color table** list, choose **Wave**.

4 Select the **Symmetrize color range** check box.

Surface 2

1 In the **Model Builder** window, under **Results** right-click **Transmitter** and choose **Surface**.

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

3 From the **Data set** list, choose **Mirror 3D 1**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Solid Mechanics>Displacement>solid.disp - Total displacement**.

5 Locate the **Data** section. From the **Time (s)** list, choose **2.4E-6**.

Deformation 1

1 Right-click **Results>Transmitter>Surface 2** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `1e3`.

Transmitter

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

2 In the **Model Builder** window, under **Results** click **Transmitter**.

3 On the **Transmitter** toolbar, click **Plot**.

Next, create a plot showing the propagation of the acoustic signal through the main pipe. For selected times, the acoustic signal profile can be seen in Figure 3.

3D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (4) (sol2)**.

4 From the **Time (s)** list, choose **2.4E-6**.

5 In the **Label** text field, type `Signal Propagation`.

Surface 1

1 Right-click **Signal Propagation** and choose **Surface**.

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

3 From the **Color table** list, choose **Wave**.

4 Select the **Symmetrize color range** check box.

Modify the **Resolution** and increase the **Element refinement** to represent the solution properly. This is because the shape functions used for the convected wave equation interface are 4th order per default.

5 Locate the **Quality** section. From the **Resolution** list, choose **Custom**.

6 In the **Element refinement** text field, type `6`.

Selection 1

1 Right-click **Results>Signal Propagation>Surface 1** and choose **Selection**.

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

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

4 On the **Signal Propagation** toolbar, click **Plot**.

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

Plot the mechanical deformation of the receiver caused by the incoming acoustic signal. The deformation is depicted in Figure 4.

3D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 3/Solution 3 (6) (sol3)**.

4 From the **Time (s)** list, choose **1E-5**.

Surface 1

1 Right-click **Receiver** 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 2>Solid Mechanics 2> Displacement>solid2.disp - Total displacement**.

Deformation 1

1 Right-click **Results>Receiver>Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `1e3`.

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

6 On the **Receiver** toolbar, click **Plot**.

The last plot will show the input voltage applied to the transmitter and the output voltage signal read on the receiver (see Figure 5).

1D Plot Group 7

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

2 In the **Settings** window for **1D Plot Group**, type `Sent and Received Signals` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

Point Graph 1

1 Right-click **Sent and Received Signals** and choose **Point Graph**.

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

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

4 Select Point 4 only.

5 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Electrostatics>Electric>V - Electric potential**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Driving voltage`

9 Locate the **Data** section. From the **Time selection** list, choose **Interpolated**.

10 Click **Range**.

11 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

12 In the **Start** text field, type `0`.

13 In the **Stop** text field, type `30*T0`.

14 In the **Number of values** text field, type `2001`.

15 Click **Replace**.

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Sent and Received Signals** and choose **Point Graph**.

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

3 From the **Data set** list, choose **Study 3/Solution 3 (6) (sol3)**.

4 Select Point 17 only.

5 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 2>Electrostatics 2>Floating potentials>es2.fp1.V - Floating potential**.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
Electric potential on receiver

9 Locate the **Data** section. From the **Time selection** list, choose **Interpolated**.

10 Click **Range**.

11 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

12 In the **Start** text field, type `0`.

13 In the **Stop** text field, type `30*T0`.

14 In the **Number of values** text field, type `2001`.

15 Click **Replace**.

Sent and Received Signals

1 In the **Model Builder** window, under **Results** click **Sent and Received Signals**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **Two y-axes** check box.

4 In the table, enter the following settings:

Plot	Plot on secondary y-axis
Point Graph 2	√

5 Locate the **Legend** section. From the **Position** list, choose **Upper middle**.

6 Locate the **Axis** section. Select the **Manual axis limits** check box.

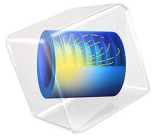
7 In the **y minimum** text field, type `-51`.

8 In the **y maximum** text field, type `51`.

9 In the **Secondary y minimum** text field, type `-1`.

10 In the **Secondary y maximum** text field, type `1`.

11 On the **Sent and Received Signals** toolbar, click **Plot**.



Transient Gaussian Explosion

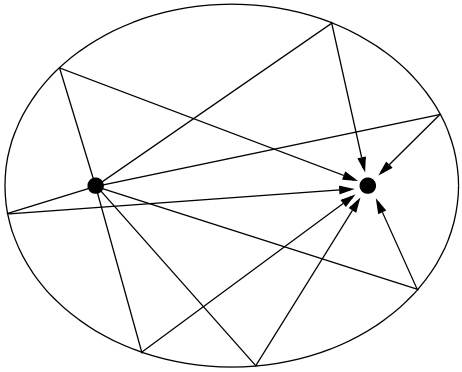
This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example introduces some important concepts to have in mind when solving transient problems. In particular, it examines the relationship between the frequency content in the sources driving the model, the mesh resolution, and the time step.

Model Definition

An ellipse with sound-hard walls has the interesting property that an acoustic signal emanating from one of the foci refocuses at the other focal point b/c seconds later, where b (SI unit: m) is the major axis length and c (SI unit: m/s) is the speed of sound.



Inspired by Ref. 1 and Ref. 2, this model involves a Gaussian explosion at one focus of an ellipse to illustrate some properties of time-dependent acoustic problems. The major and minor axis lengths are 10 m and 8 m, respectively. The major axis coincides with the x-axis and the foci are located at $x = -3$ m and $x = 3$ m. Because of symmetry the model can be limited to the upper half-plane.

Denoting the fluid density by ρ and the speed of sound by c, the acoustic pressure field $p(\mathbf{x}, t)$, inside the elliptical chamber is governed by the scalar wave equation

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}\nabla p\right) = S(\mathbf{x}, t)$$

where the point-source term on the right-hand side is given by

$$S(\mathbf{x}, t) = \frac{dg}{dt}(t)\,\delta^{(2)}(\mathbf{x} - \mathbf{x}_0)$$

where $\delta^{(2)}$ is the two dimensional Dirac delta function. The time dependence of the explosion is determined by the cutoff Gaussian pulse

$$g(t) = \begin{cases} A e^{-\pi^2 f_0^2 (t-\tau)^2} & 0 < t < 2\tau \\ 0 & \text{otherwise} \end{cases}$$

describing the rate of airflow (SI unit: m^2/s) away from the source, located at $\mathbf{x} = \mathbf{x}_0$. The parameter f_0, which is proportional to the pulse bandwidth, is chosen as $f_0 = 380$ Hz. As the following plots show, by taking $\tau = 1/f_0$ the pulse very closely approximates a full Gaussian, the effect of the cutoff tails being numerically insignificant.

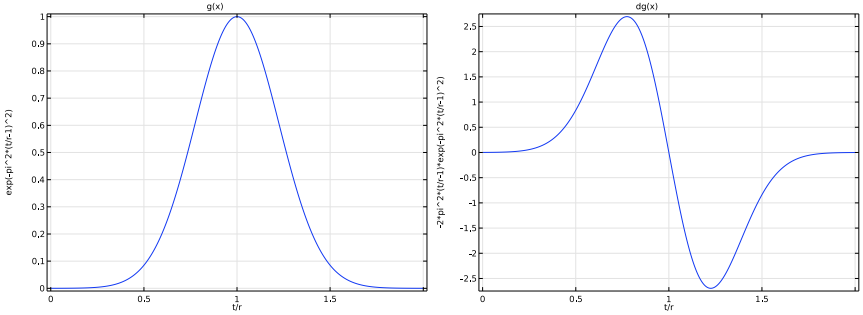


Figure 1: Normalized Gaussian pulse (left) and its derivative (right).

A particularly interesting property of the Gaussian function is that its Fourier transform, depicted in Figure 3, is equally simple (neglect cutoff effects):

$$G(\omega) \equiv \int_{-\infty}^{\infty} g(t) e^{-i\omega t} dt = \frac{2A\sqrt{\pi}}{\omega_0} e^{-\frac{\omega^2}{\omega_0^2} - i\omega t_0}$$

where $\omega_0 = 2\pi f_0$. The magnitude of the Fourier transform falls off quickly for increasing angular frequencies ω. Practically all the energy in the signal is contained in the frequency band $-2\omega_0 < \omega < 2\omega_0$ with most of it concentrated between $-\omega_0$ and ω_0.

Therefore, when using a forcing function of this type, it is enough to resolve wavelengths corresponding to the angular frequency ω_0, which in turn corresponds to the frequency f_0. In this model the parameter that defined the maximal frequency to resolve is $f_{\max}$, it is set equal to 600 Hz which is slightly higher than f_0 (and will resolve the pulse fully).

In order to well resolve the propagating pulse in space the rule of thumb is to use a minimum of 5 to 6 elements per wavelength (at the maximal frequency), when using quadratic Lagrange elements. The typical mesh size is given by

$$h_{\max} = \frac{\lambda_{\min}}{N} = \frac{c}{f_{\max} N}$$

Where $h_{\max}$ is a typical maximal mesh-element size, and N is the number of elements per wavelength required to resolve a harmonic wave with some accuracy. The following discussion uses $N = 5$. The important point is that there is little to gain in prescribing a forcing function that contains frequencies that the mesh cannot resolve.

In addition to controlling the pulse shape the mesh resolution imposes a restriction on the internal time steps size taken by the solver. In the transient interfaces of the Acoustics Module the solver is controlled from the **Transient Solver Settings** section found at the top physics level. The user should enter the **Maximum frequency to resolve** in the model. It is also possible to choose a **Time stepping** (method) here it is recommended to use the default **Fixed (preferred)** option.

TIME STEPPING EXPLANATION

The logic for the automatic choice made is as follows. The relationship between mesh size and time-step size is closely related to the CFL number (Ref. 3), which is defined as

$$\mathrm{CFL} = \frac{c\Delta t}{h}$$

This nondimensional number can be interpreted as the fraction of an element the wave travels in a single time step. A CFL number around 1 would correspond to the same resolution in space and time if the discretization errors were of the same size; however, that is normally not the case.

By default, COMSOL Multiphysics uses the implicit second-order accurate method generalized-α to solve transient acoustics problems. In space, the default is 2nd-order Lagrange elements. Generalized-α introduces some numerical damping of high frequencies but much less than the BDF method.

The temporal discretization errors for generalized-α are larger than the spatial discretization errors when 2nd-order elements are used in space. The limiting step size, where the errors are of roughly the same size, can be found somewhere at $\mathrm{CFL} < 0.2$. You can get away with a longer time step if the forcing does not make full use of the mesh resolution; that is, if high frequencies are absent from the outset.

When the excitation contains all the frequencies the mesh can resolve, there is no point in using an automatic time-step control which can be provided by the time-dependent solver. The tolerances in the automatic error control are difficult to tune when there is weak but important high-frequency content. Instead, you can use your knowledge of the typical mesh size, speed of sound, and CFL number to calculate and prescribe a fixed time step. This is exactly the default behavior when the **Fixed (preferred)** method is chosen in the **Transient Solver Settings** section. The **Free** option corresponds to the automatic time-step control but with some tighter controls of the allowed time-steps. This latter option is still not recommended as the manual option typically yields much better results (and is faster).

The internal time step generated by the **Fixed (preferred)** option and the entered **Maximal frequency to resolve** is set by assuming that the user has generated a mesh that properly resolves the same maximal frequency (minimal wavelength). The following step is generated

$$\Delta t = \frac{h_{\max}\mathrm{CFL}}{c} = \frac{\mathrm{CFL}}{f_{\max}N} \approx \frac{1}{60f_{\max}}$$

Assuming that N is between 5 and 6 and the CFL number is roughly 0.1. These values give a good margin of safety.

Using the properties of air for the medium and selecting a mesh density based on the parameters The model runs for 0.035 s so that you can study the refocusing at the right-hand focus point at roughly 0.0315 s.

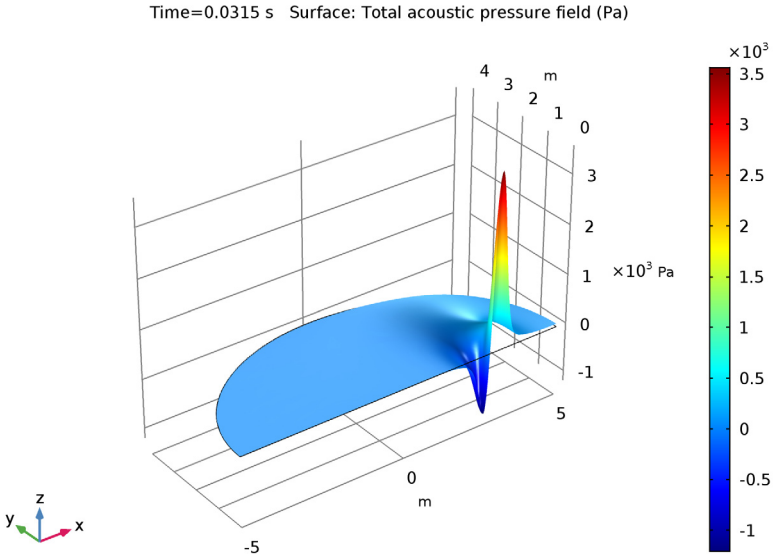


Figure 2: The refocusing occurs at roughly 0.0315 s. An animation gives a better feeling for the process.

For the selected combination of mesh size, pulse shape, and time step, the solution can be shown to be both smooth and accurate. Selecting a smaller value for N leads to oscillations if the CFL number is small enough (choosing a too low frequency to resolve), while selecting a higher CFL number (and consequentially a larger time step) leads to an inaccurate solution.

Figure 3 shows the Fourier transform of the source signal. Plotting the frequency content of the source signal is a good way for selecting the maximal frequency to resolve in the model.

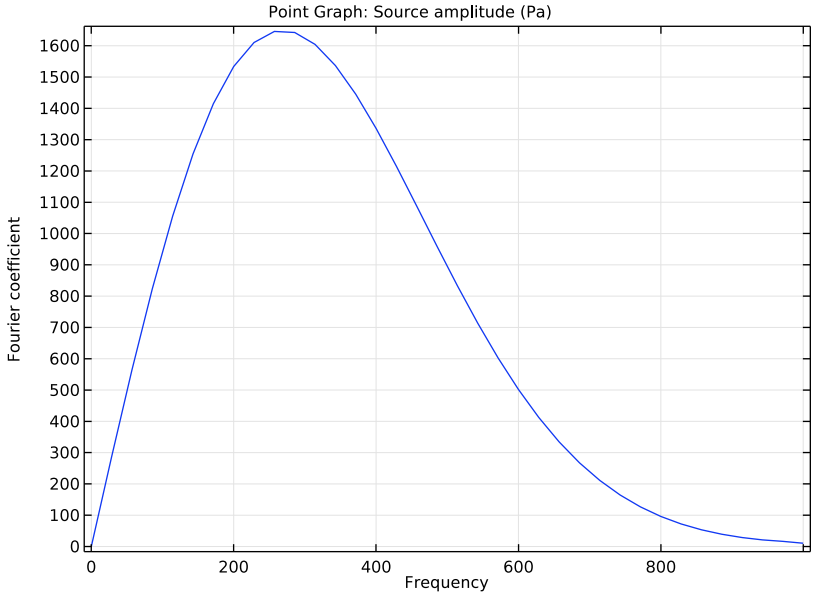


Figure 3: Fourier transform of the source signal.

References

1. B. Yue and M.N. Guddati, "Dispersion-reducing Finite Elements for Transient Acoustics", *J. Acoust. Soc. Am.*, vol. 118, no. 4, pp. 2132–2141, 2005.

2. H.-O. Kreiss, N.A. Peterson, and J. Yström, "Difference Approximations for the for the second order wave equation", *SIAM J. of Num. Analys.*, vol. 40, 1940–1967, 2002.

3. R. Courant, K.O. Friedrichs, and H. Lewy, "On the Partial Difference Equations of Mathematical Physics", *IBM Journal*, vol. 11, pp. 215–234, 1956.

Application Library path: `Acoustics_Module/Tutorials/gaussian_explosion`

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**.

2 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

6 Click **Done**.

GEOMETRY 1

Ellipse 1 (e1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Ellipse**.

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

3 In the **a-semiaxis** text field, type `5`.

4 In the **b-semiaxis** text field, type `4`.

5 In the **Sector angle** text field, type `180`.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Point**.

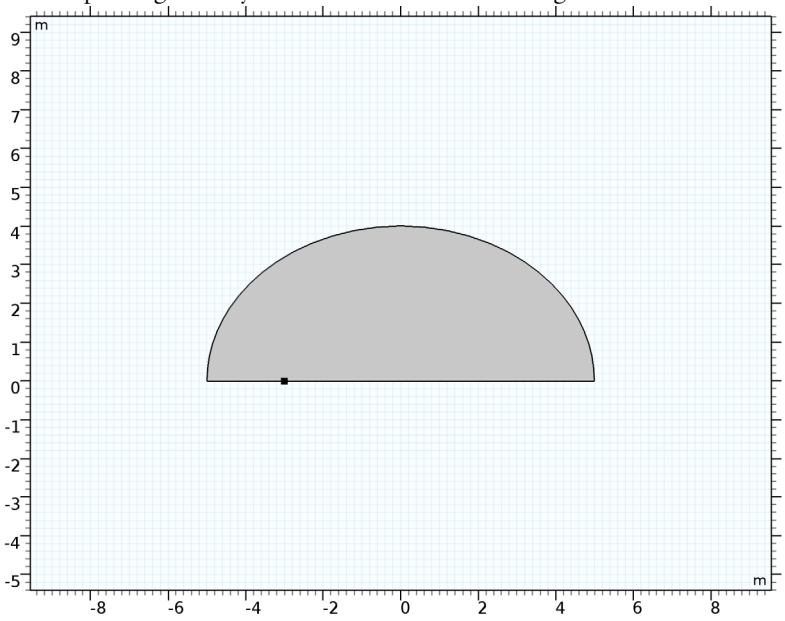
2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `-3`.

4 Click **Build All Objects**.

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

The completed geometry should look like that in the figure below.



GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
c_air	343[m/s]	343 m/s	Speed of sound in air
f_max	600[Hz]	600 Hz	Maximum frequency to resolve
N	5	5	Elements per wavelength
h_max	c_air/f_max/N	0.11433 m	Typical element size
A	4[m^2/s]	4 m²/s	Point-source amplitude
f0	380[Hz]	380 Hz	Source frequency bandwidth
t0	1/f0	0.0026316 s	Source pulse half width

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

Enter the maximal frequency to be resolved in the model. This value can be assessed from a Fourier analysis of the sources as depicted in Figure 3.

Under the **Transient Solver Settings** section enter the **Maximum frequency to resolve** and set it to `f_max`.

Point Source 1

1 On the **Physics** toolbar, click **Points** and choose **Point Source**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Source**, locate the **Point Source** section.

4 From the **Type** list, choose **Gaussian pulse**.

5 In the A text field, type `A`.

6 In the f_0 text field, type `f0`.

7 In the t_p text field, type `t0`.

MESH 1

With the choice of a typical mesh size set to h_max, the mesh must be made as isotropic as possible. You can accomplish this by setting the maximum mesh size explicitly while keeping the other mesh parameters relaxed.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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 `h_max`.

5 Click **Build All**.

STUDY 1

Step 1: Time Dependent

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

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,0.5e-3,35e-3)`.

This setting gives you a solution output at every 0.5 ms from t = 0 to t = 35 ms. This should not be confused with the time steps actually taken by the solver. These are set automatically based on the settings entered above. To see where they end up, do the following:

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

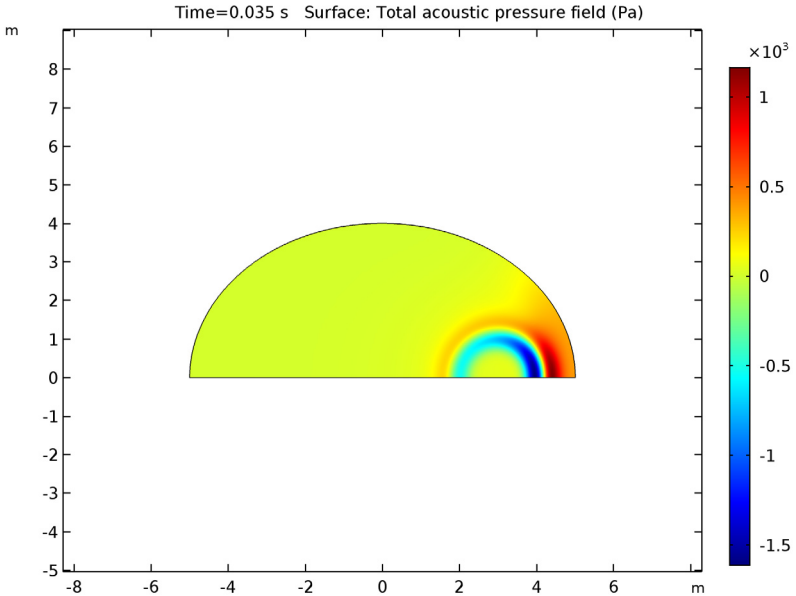
The solver automatically picks up the settings entered at the physics level (in the Transient Solver Settings section) when the default solver is generated or when solve is clicked the first time. It is in general not necessary to edit these settings.

Experienced users maybe find it useful to edit these settings.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (actd)



The default plot shows the pressure at the final time. To get a more attractive plot, you can add a height.

Surface

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

Height Expression 1

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

You can select different times to look at the wave using the parent Acoustic Pressure (`actd`) node. At t = 0.0315 s, you are close to the moment when the waves refocus.

Acoustic Pressure (actd)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (actd)**.

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

3 From the **Time (s)** list, choose **0.0315**.

4 On the **Acoustic Pressure (actd)** toolbar, click **Plot**.

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

The resulting plot is found in Figure 2. It is illustrative to animate transient problems in general and wave propagation in particular; you can do this by right-clicking the Export node and adding an Animation feature.

It is also possible to plot the field on all sides of the point source by defining a mirror data set.

Mirror 2D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 2D**.

The symmetry line is the x-axis.

2 In the **Settings** window for **Mirror 2D**, locate the **Axis Data** section.

3 From the **Axis entry method** list, choose **Point and direction**.

4 Find the **Direction** subsection. In the **X** text field, type 1.

5 In the **Y** text field, type 0.

Acoustic Pressure (actd)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (actd)**.

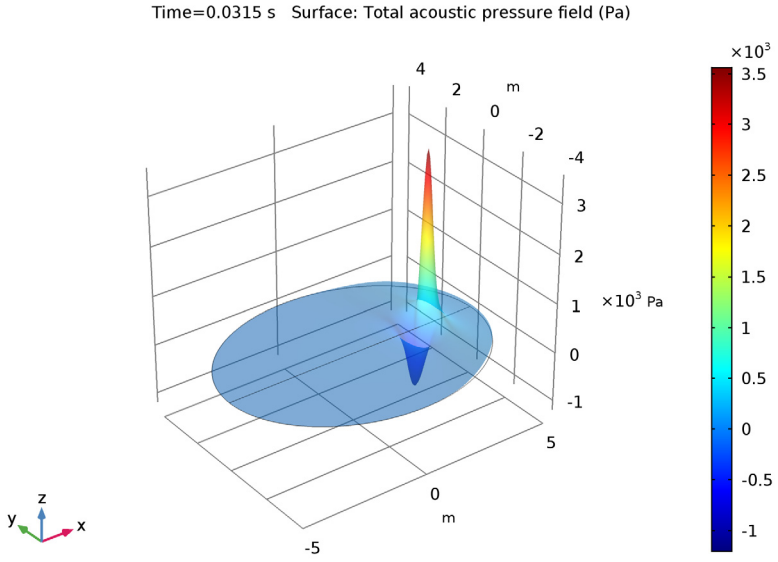
2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Mirror 2D 1**.

4 On the **Acoustic Pressure (actd)** toolbar, click **Plot**.

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

6 Click the **Transparency** button on the **Graphics** toolbar.



7 Click the **Transparency** button on the **Graphics** toolbar to restore the default transparency setting.

The following instructions show how to create Figure 1. These instructions are optional.

Plot the normalized Gaussian and its derivative by plotting two analytic functions.

GLOBAL DEFINITIONS

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `g` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `exp(-pi^2*(x-1)^2)`.

4 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
x	0	2

5 Click **Create Plot**.

RESULTS

1D Plot Group 2

1 In the **Model Builder** window, under **Results** click **1D Plot Group 2**.

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

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `t/r`.

4 In the **y-axis label** text field, type `exp(-pi^2*(t/r-1)^2)`.

5 On the **Normalized Gaussian** toolbar, click **Plot**.

GLOBAL DEFINITIONS

Analytic 2 (an2)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `dg` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `-2*pi^2*(x-1)*exp(-pi^2*(x-1)^2)`.

4 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
x	0	2

5 Click **Create Plot**.

RESULTS

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, type `Derivative of norm. Gaussian` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `t/r`.

4 In the **y-axis label** text field, type `-2*pi^2*(t/r-1)*exp(-pi^2*(t/r-1)^2)`.

5 On the **Derivative of norm. Gaussian** toolbar, click **Plot**.

Finally, create the plot shown in Figure 3 where the frequency content of the source is analyzed.

1D Plot Group 4

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

2 In the **Settings** window for **1D Plot Group**, type `Fourier Transform of Source` in the **Label** text field.

Point Graph 1

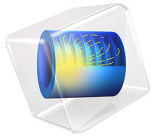
1 Right-click **Fourier Transform of Source** and choose **Point Graph**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1> Pressure Acoustics, Transient>Sources>actd.mls1.S - Source amplitude**.

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

5 On the **Fourier Transform of Source** toolbar, click **Plot**.



Gaussian Pulse in 2D Uniform Flow: Convected Wave Equation and Absorbing Layers

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This small tutorial is a standard test and benchmark model for nonreflecting conditions and sponge layers for linearized Euler like systems, see Ref. 1 and Ref. 2. The Convected Wave Equation, Time Explicit interface solves the linearized Euler equations with an adiabatic equation of state and the interface has the Absorbing Layers to model infinite domains.

An acoustic pulse is generated by an initial Gaussian distribution at the center of the computational domain. The pulse propagates in a high Mach number uniform flow. An analytical solution exists to the problem and is used to validate the solution. The model shows how to set up and use the absorbing layers.

Model Definition

The computational domain with absorbing layers is depicted in Figure 1. The model is set up in a dimensionless system where the speed of sound $c_0 = 1$ and the density $\rho_0 = 1$. The Gaussian pulse is emitted at the origin $\mathbf{x} = \mathbf{0}$ with initial values

$$\begin{bmatrix} p \\ u \\ v \end{bmatrix} = \begin{bmatrix} 1 \\ \beta x \\ \beta y \end{bmatrix} e^{-\alpha(x^2+y^2)} \qquad \text{for} \qquad t = 0 \tag{1}$$

where $\alpha = \ln(2)/9$ and $\beta = 0.04$. The parameters and the expressions for the initial values are defined as parameters and variables and loaded from files during the model setup. The pulse propagates in a uniform background flow $\mathbf{u}_0 = (\mathrm{Ma},0)$, where $\mathrm{Ma} = 0.5$. The analytical solution to Equation 1 is given by (see Ref. 2)

$$p(\mathbf{x},t) = \frac{1}{2\alpha}\int_0^\infty \left[\cos(\lambda t) - \frac{\beta}{2\alpha}\lambda\sin(\lambda t)\right]\lambda J_0(\lambda r) e^{-\frac{\lambda^2}{4\alpha}} d\lambda \tag{2}$$

$$r = \sqrt{(x-\mathrm{Ma}t)^2 + y^2}$$

In the model the `integrate()` operator is used to express the analytical solution. The integration is performed on a finite interval.

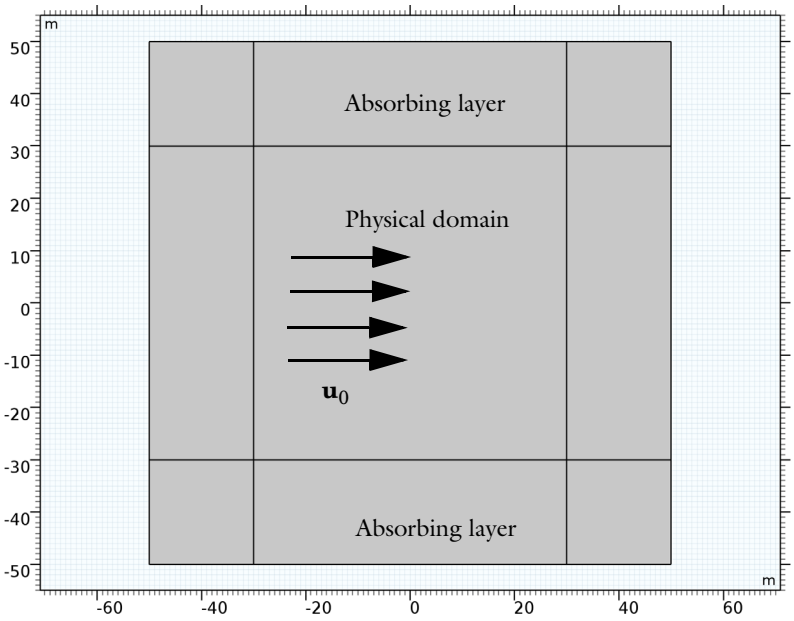


Figure 1: Geometry, physical domain and absorbing layers.

Results and Discussion

The propagation of the acoustic pulse is depicted in Figure 2 and Figure 3. The two figures show the acoustic pressure and the acoustic particle velocity, respectively, at four consecutive time instances. In Figure 4 the pressure and velocity are depicted at the final simulated time $t = 120$. Here the pulse is inside the absorbing layer. It is evident how the scaling in the layer slows the pulse (it moves slower and slower towards the outer edges) and makes it propagate more normal to the outer boundary. This shows one of the principles of the absorbing layer. The other two mechanisms at work is filtering and a simple impedance condition at the outer boundary.

In the final two figures the simulated results are compared to the analytical solution. In Figure 5 the pressure at point $(20, 30)$ is depicted as function of time. In Figure 6 the pressure is depicted along the x-axis at $t = 50$. Both show very good agreement with the analytical solution. The absorbing layers can under optimal condition reduce the spuriously reflected waves to $1/1000$ of the incident field amplitude.

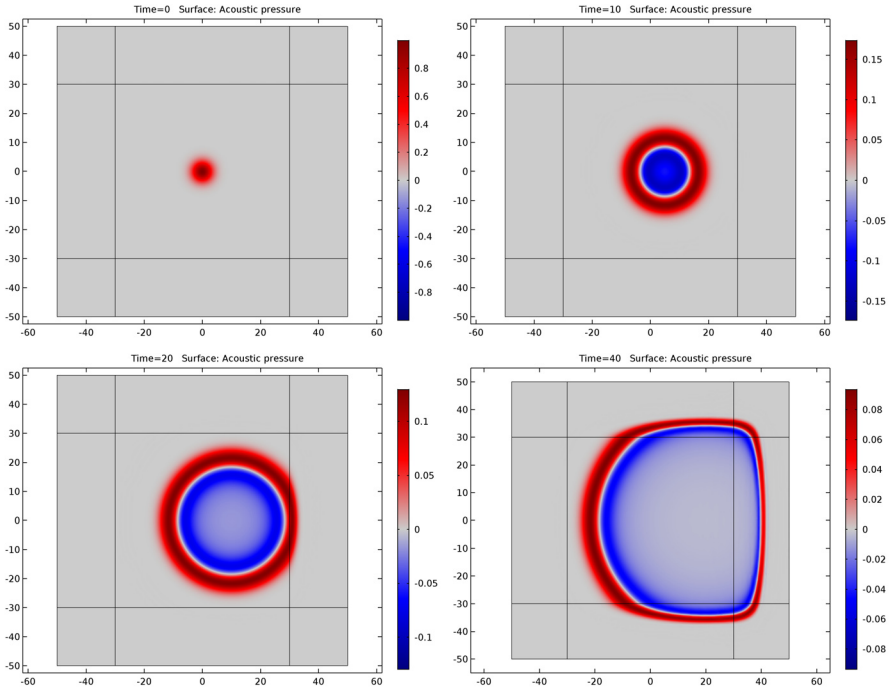


Figure 2: The pressure profile at different time steps.

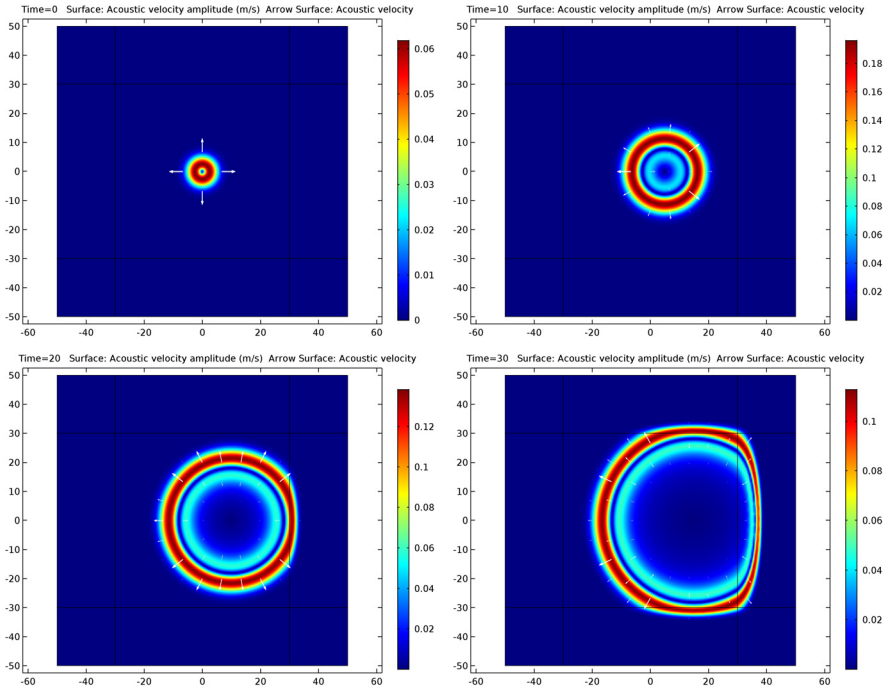


Figure 3: The acoustic velocity profiles at different time steps.

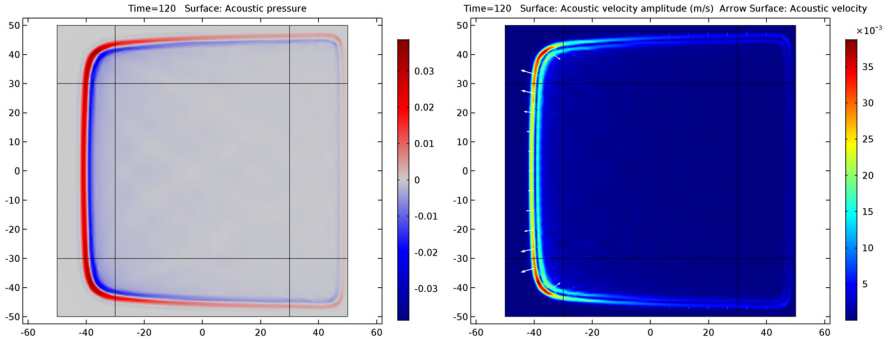


Figure 4: The pressure profile (left) and acoustic velocity profile (right) at the final simulated time t = 120.

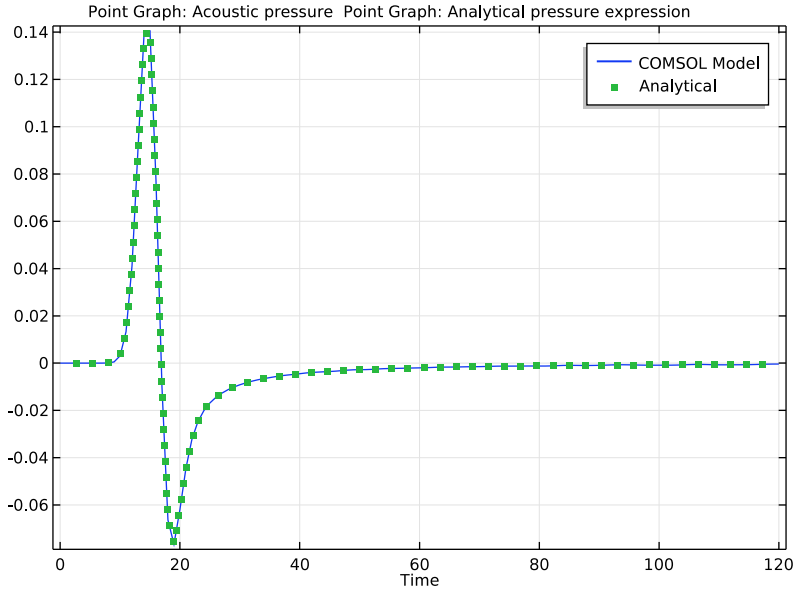


Figure 5: The pressure as function of time in point (x,y) = (20,30). The model solution compared to the analytical solution.

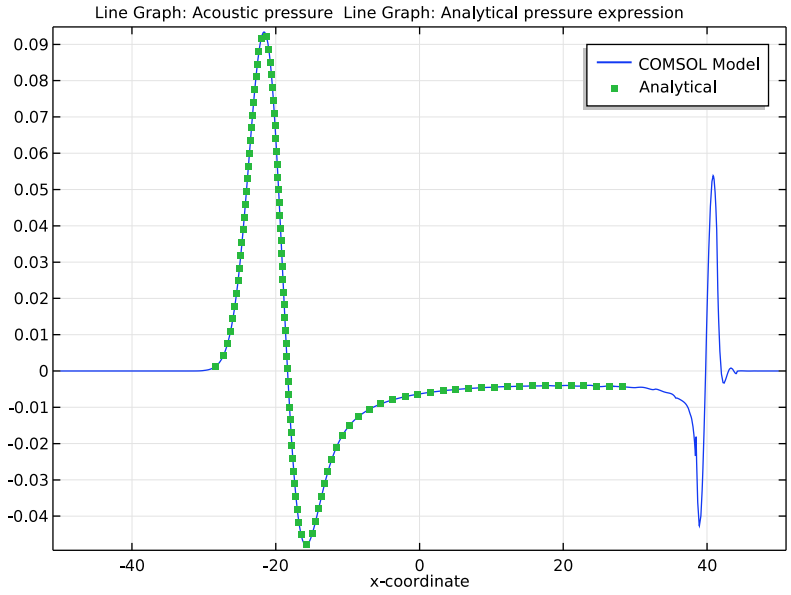


Figure 6: Pressure profile along the x-axis at t = 50 comparing the analytical solution to the COMSOL solution.

References

1. H. L. Atkins, "Application of Essentially Nonoscillatory Methods to Aeroacoustic Flow Problems," Proceedings of ICASE/LaRCWorkshop on Benchmark Problems in Computational Aeroacoustics, Edited by J. C. Hardin, J. R. Ristorcelli and C. K. W. Tam, NASA CP-3300, pp. 15-26, 1995.

2. H. L. Atkins and C. W. Shu, "Quadrature-Free Implementation of Discontinuous Galerkin Method for Hyperbolic Equations," AIAA Journal, Vol. 36, pp. 775-782, 1998.

Application Library path: `Acoustics_Module/Tutorials/gaussian_pulse_absorbing_layers`

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

NEW

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

MODEL WIZARD

1. In the **Model Wizard** window, click **2D**.
2. In the **Select Physics** tree, select **Acoustics>Ultrasound>Convected Wave Equation, Time Explicit (cwe)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Time Dependent**.
6. Click **Done**.

GLOBAL DEFINITIONS

Load the parameters used to define the geometry, the Gaussian pulse, and the background flow properties.

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `gaussian_pulse_absorbing_layers_parameters.txt`.

GEOMETRY 1

Square 1 (sq1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Square**.
2. In the **Settings** window for **Square**, locate the **Size** section.
3. In the **Side length** text field, type `W`.
4. Locate the **Position** section. From the **Base** list, choose **Center**.
5. Click to expand the **Layers** section. Select the **Layers to the left** check box.
6. Select the **Layers to the right** check box.
7. Select the **Layers on top** check box.

8 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	dW

9 Click **Build All Objects**.

The geometry consists of a physical domain surrounded by the absorbing layers, see Figure 1.

DEFINITIONS

Next, load the variables that define the initial Gaussian shape (acoustic pressure and velocity components) and then set up the absorbing layers.

1 In the **Model Builder** window, expand the **Definitions** node.

Variables 1

1 Right-click **Component 1 (comp1)>Definitions** and choose **Variables**.

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

3 Click **Load from File**.

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

Absorbing Layer 1 (ab1)

1 On the **Definitions** toolbar, click **Absorbing Layer**.

2 Select Domains 1–4 and 6–9 only.

Before setting up the physics, change the unit system to be dimensionless.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Unit System** section.

3 From the **Unit system** list, choose **None**.

CONVECTED WAVE EQUATION, TIME EXPLICIT (CWE)

At the physics interface level there are a number of interesting settings. Some are hidden per default. Start by enabling the **Advanced Physics** options to be able to see the **Filter Parameters for Absorbing Layers** section.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

In the same way you can also access the **Discretization** section by enabling it. Notice that the default discretization for both the pressure and the velocity is **Quartic**, that is, fourth order. Further details about this choice and the time explicit method used in the Convected Wave Equation interface can be found in the Acoustics Module User's Guide.

Convected Wave Equation Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Convected Wave Equation, Time Explicit (cwe)** click **Convected Wave Equation Model 1**.

2 In the **Settings** window for **Convected Wave Equation Model**, locate the **Model Input** section.

3 In the p_0 text field, type `p0`.

4 Specify the $\mathbf{u}_0$ vector as

`Ma*c0`	`x`
`0`	`y`

5 Locate the **Fluid Properties** section. From the ρ_0 list, choose **User defined**. In the associated text field, type `rho0`.

6 From the c_0 list, choose **User defined**. In the associated text field, type `c0`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Convected Wave Equation, Time Explicit (cwe)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the p text field, type `p_i`.

4 Specify the $\mathbf{u}$ vector as

`u_i`	`x`
`v_i`	`y`
`0`	`z`

Finally, set up the impedance condition and apply it to all the outer boundaries. This simple nonreflecting condition is the final piece (combined with filtering and coordinate stretching) that makes the absorbing layers work.

Acoustic Impedance 1

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

2 In the **Settings** window for **Acoustic Impedance**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

MESH 1

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

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

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

For the mesh we have selected the default **Finer** setting. A more thorough approach would be to define the mesh size according to the highest frequency component in the model (the smallest wavelength to resolve). For the time explicit discontinuous Galerkin method (with default fourth order discretization) use two mesh elements per wavelength. Guidelines for meshing can be found in the documentation for the interface.

STUDY 1

Step 1: Time Dependent

Solve the model from time t = 0 to 120 in steps of 1 (dimensionless time units). Simply modify the default expression.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,1,120)`.

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

The first default plot shows the acoustic pressure in the computational domain. You can change the times to plot the distribution at various times, see Figure 2 and Figure 4.

RESULTS

Acoustic Pressure (cwe)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (cwe)**.

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

3 From the **Time (s)** list, choose **0**.

4 On the **Acoustic Pressure (cwe)** toolbar, click **Plot**.

The second default plot shows the acoustic velocity. The plots, for selected times, can be seen in Figure 3 and Figure 4.

Acoustic Velocity (cwe)

1 In the **Model Builder** window, under **Results** click **Acoustic Velocity (cwe)**.

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

3 From the **Time (s)** list, choose **0**.

4 On the **Acoustic Velocity (cwe)** toolbar, click **Plot**.

Create some extra data sets to use in the following plots. Create a cut point to plot the pressure as function of time in a desired coordinate set. Create a cut line to plot the pressure along a line for a given time. Finally, create a data set with a selection restricting the solution to the physical domain (not the absorbing layers).

Cut Point 2D 1

1 On the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **x** text field, type 20.

4 In the **y** text field, type 10.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **x** to -50.

4 In row **Point 2**, set **x** to 50.

Cut Line 2D 2

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **x** to -30.

4 In row **Point 2**, set **x** to 30.

The pressure as function of time is depicted in Figure 5 where it is compared to the analytical solution.

1D Plot Group 4

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.

Point Graph 1

1 Right-click **Pressure in Point** and choose **Point Graph**.

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

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

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
COMSOL Model

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Pressure in Point** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `p_a`.

4 Locate the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
Analytical

7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

9 In the **Number** text field, type `100`.

10 On the **Pressure in Point** toolbar, click **Plot**.

The pressure depicred on the x-axis for a given time is depicted in Figure 6 where it is compared to the analytical solution.

1D Plot Group 5

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

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

3 Locate the **Data** section. From the **Data set** list, choose **None**.

Line Graph 1

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

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

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, select **40**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `p`.

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

8 In the **Expression** text field, type `x`.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
COMSOL Model

12 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

13 On the **Pressure over Cut Line** toolbar, click **Plot**.

14 Click **Plot**.

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **Pressure over Cut Line** and choose **Line Graph**.

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

3 From the **Data set** list, choose **Cut Line 2D 2**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, select **40**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `p_a`.

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

8 In the **Expression** text field, type `x`.

9 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

10 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

11 In the **Number** text field, type `100`.

12 Locate the **Legends** section. Select the **Show legends** check box.

13 From the **Legends** list, choose **Manual**.

14 In the table, enter the following settings:

Legends
Analytical

15 Locate the **Quality** section. From the **Resolution** list, choose **Extra fine**.

Note that when plotting the solution of the simulation, as function of spatial variables (line plot or a surface), you need to increase the resolution. This is because of the default quadratic elements used. The default plots generated already have a higher default resolution.

16 On the **Pressure over Cut Line** toolbar, click **Plot**.

Finally, plot the pressure only in the physical domain. This plot can, for example, be used to create a nice animation. Under the **Export >** node select **Animation** and then select the last plot.

Another interesting observation is that the remaining (spuriously) reflected waves have an amplitude which is a factor 1000 smaller than the original signal (from the solution at time t = 0).

Acoustic Pressure (cwe) 1

1 In the **Model Builder** window, under **Results** right-click **Acoustic Pressure (cwe)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Acoustic Pressure (cwe) Selection` in the **Label** text field.

Acoustic Pressure

In the **Model Builder** window, expand the **Results>Acoustic Pressure (cwe) Selection** node.

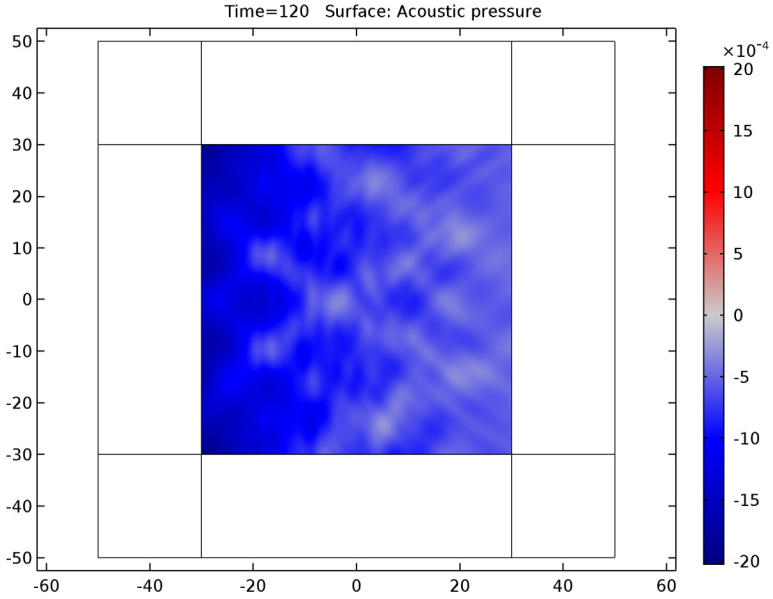
Selection 1

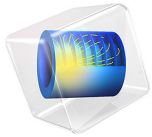
1 Right-click **Acoustic Pressure** and choose **Selection**.

2 Select Domain 5 only.

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

4 On the **Acoustic Pressure (cwe) Selection** toolbar, click **Plot**.





Gaussian Pulse Absorption by Perfectly Matched Layers: Pressure Acoustics, Transient

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This tutorial model is a test and benchmark model for perfectly matched layers (PML) in time domain. PMLs are widely used for the problems of wave propagation in open domains. A perfectly matched layer is supposed to meet two main requirements. First, a wave must propagate without spurious reflections from the interface between the physical domain and the PML. Second, a PML must ensure the long time stability of the numerical solution.

The model solves the same problem as in Gaussian Pulse in 2D Uniform Flow: Convected Wave Equation and Absorbing Layers in the absence of the background flow. The Pressure Acoustics, Transient interface solves the wave equation in a squared computational domain, the Perfectly Matched Layers surround the computational domain suppressing the reflections from the outer boundary. An acoustic pulse is generated by an initial Gaussian distribution at the center of the computational domain. An analytical solution to the problem exists and is used to validate the solution. The model shows how to set up and use the PMLs.

Model Definition

The computational domain with perfectly matched layers is depicted in Figure 1. The model is set up in a dimensionless system where the speed of sound $c_0 = 1$ and the density $\rho_0 = 1$. The Gaussian pulse is emitted at the origin $\mathbf{x} = \mathbf{0}$ with initial values

$$\begin{bmatrix} p \\ u \\ v \end{bmatrix} = \begin{bmatrix} 1 \\ \beta x \\ \beta y \end{bmatrix} e^{-\alpha(x^2+y^2)} \qquad \text{for} \qquad t = 0 \tag{1}$$

where $\alpha = \ln(2)/9$ and $\beta = 0.04$. Since the velocity $\mathbf{u} = (u, v)$ does not enter the wave equation explicitly, the two last initial values are reformulated in terms of the first time derivative of the pressure. The use of the mass conservation law yields

$$\frac{\partial p}{\partial t} = -\rho_0 c_0^2 \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) \qquad \text{for} \qquad t = 0.$$

The Gaussian pulse parameters and the expressions for the initial values are included to the model as parameters and variables.

The analytical solution to Equation 1 is given by (see Ref. 1)

$$p(\mathbf{x},t)=\frac{1}{2\alpha}\int_0^{\infty}\left[\cos(\lambda t)-\frac{\beta}{2\alpha}\lambda\sin(\lambda t)\right]\lambda J_0(\lambda r)e^{-\frac{\lambda^2}{4\alpha}}d\lambda,\ r=\sqrt{x^2+y^2}$$

In the model the `integrate()` operator is used to express the analytical solution. The integration is performed on a finite interval.

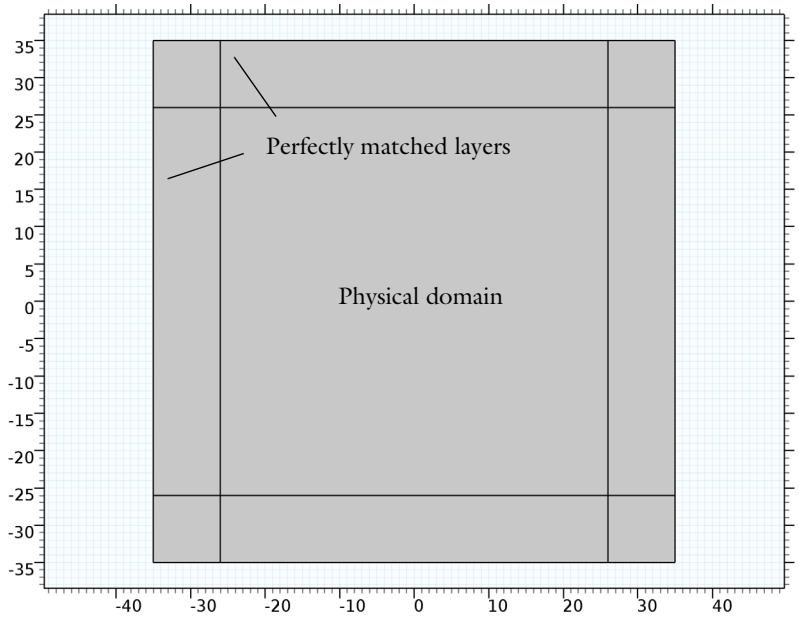


Figure 1: Geometry: physical domain and perfectly matched layers.

Results and Discussion

The propagation of the acoustic pulse is depicted in Figure 2. It shows the distribution of the acoustic pressure across the computational domain at four time steps. By the time $t = 30$, the pulse has entered the PML domain without any reflections from the interface between the physical domain and the PMLs. The pressure at the final simulated time $t = 120$ is depicted in Figure 3. By this time, the pulse has left the computational domain. There are no visible signals propagating back to the physical domain, which asserts the long-time stability of the perfectly matched layers.

In the next two figures the simulated results are compared to the analytical solution. In Figure 4 the pressure at point (20, 10) is depicted as function of time. In Figure 5 the

pressure is depicted along the x-axis at $t = 40$. Both show very good agreement with the analytical solution.

Figure 6 shows the frequency spectrum of the signal at point (20, 10).

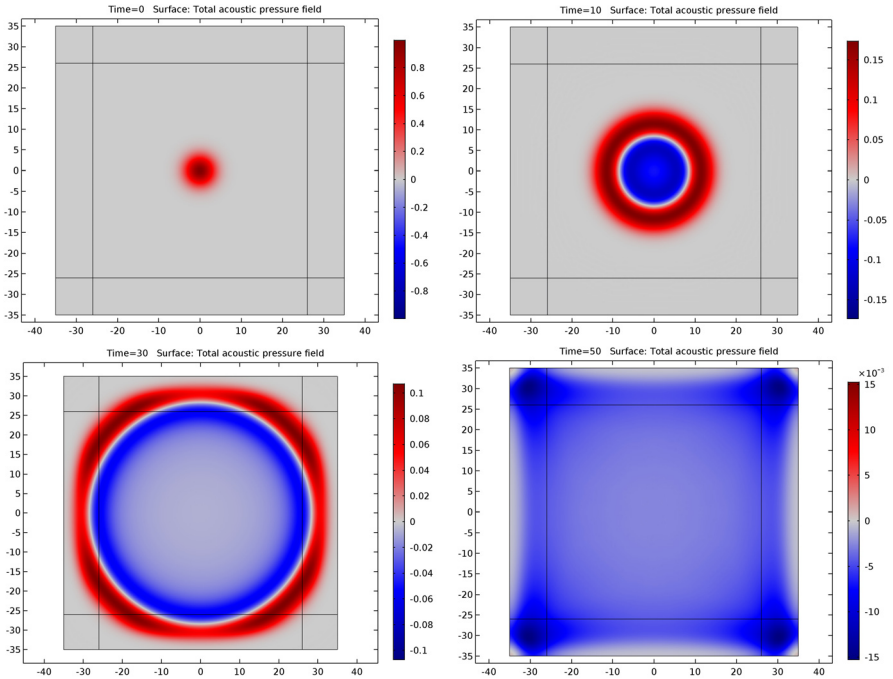


Figure 2: The pressure profile at different time steps.

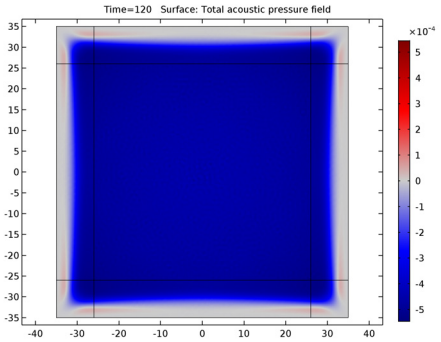


Figure 3: The pressure profile at the final simulated time t = 120.

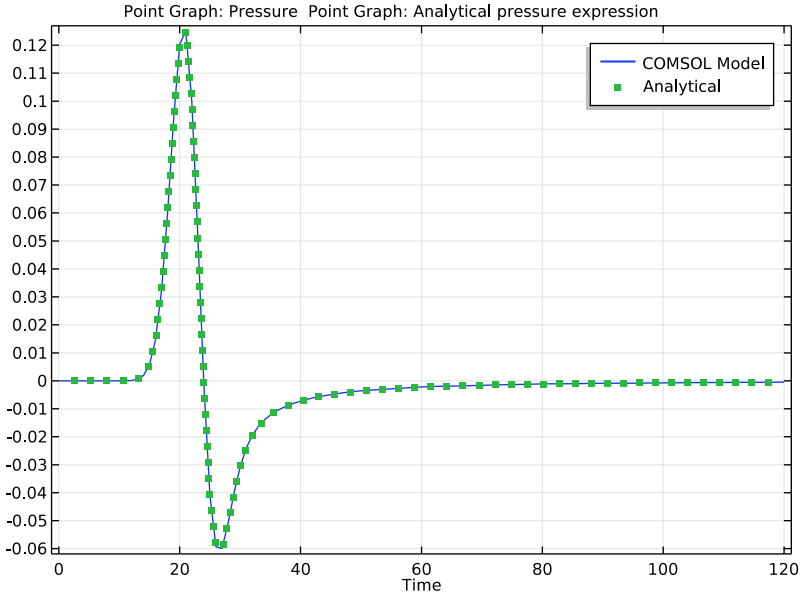


Figure 4: The pressure as function of time at point (x, y) = (20, 10). The model solution compared to the analytical solution.

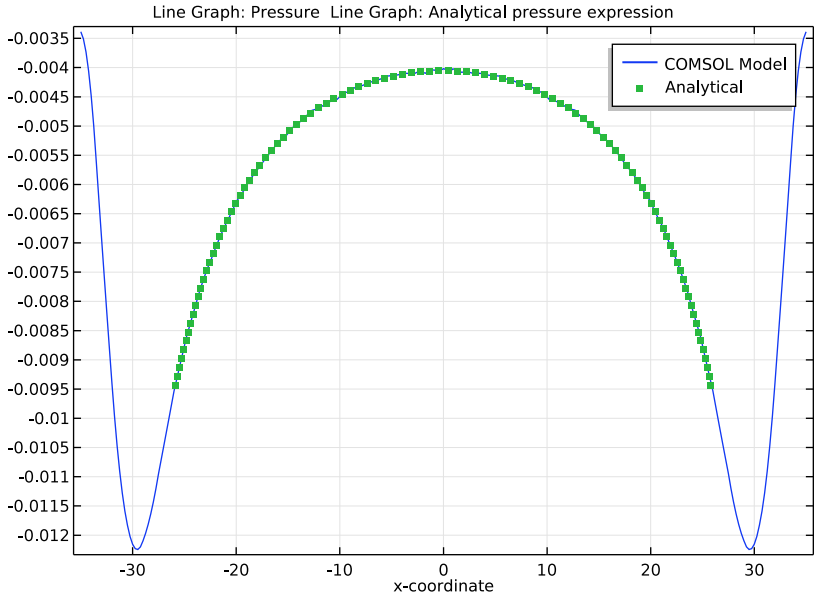


Figure 5: Pressure profile along the x-axis at t = 40 comparing the analytical solution to the COMSOL solution.

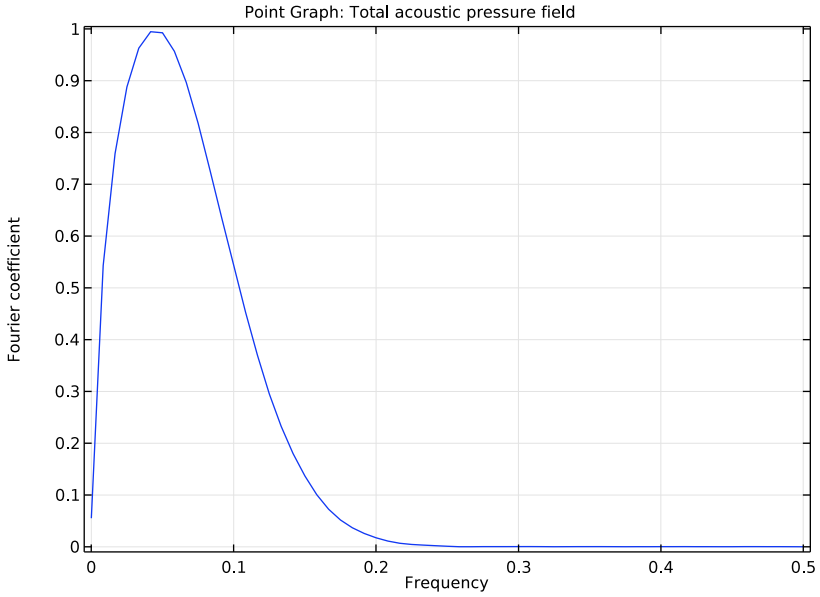


Figure 6: Frequency spectrum of the signal at point (x, y) = (20, 10).

References

1. H. L. Atkins and C. W. Shu, "Quadrature-Free Implementation of Discontinuous Galerkin Method for Hyperbolic Equations," AIAA Journal, Vol. 36, pp. 775-782, 1998.

Application Library path: `Acoustics_Module/Tutorials/gaussian_pulse_perfectly_matched_layers`

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**.

2 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

6 Click **Done**.

GEOMETRY 1

Before setting up the physics, change the unit system to be dimensionless.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Unit System** section.

3 From the **Unit system** list, choose **None**.

GLOBAL DEFINITIONS

Load the parameters that define the geometry and the Gaussian pulse properties.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `gaussian_pulse_perfectly_matched_layers_parameters.txt`.

DEFINITIONS

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

Next, load the variables that define the initial Gaussian shape. These variables will be used as the initial conditions for the acoustic pressure and its first time derivative.

Variables 1

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

2 Click **Load from File**.

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

The geometry contains a physical domain surrounded by perfectly matched layers (PML), see Figure 1. The last are used to truncate the physical domain without introducing spurious numerical reflections from the outer boundary.

GEOMETRY 1

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for **Square**, locate the **Position** section.

3 From the **Base** list, choose **Center**.

4 Locate the **Size** section. In the **Side length** text field, type `W`.

5 Click to expand the **Layers** section. Select the **Layers to the left** check box.

6 Select the **Layers to the right** check box.

7 Select the **Layers on top** check box.

Specify the width of the PMLs.

8 In the table, enter the following settings:

Layer name	Thickness
`Layer 1`	`dW`

9 On the **Geometry** toolbar, click **Build All**.

DEFINITIONS

Set up a PML surrounding the physical domain.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1–4 and 6–9 only.

Use cubic polynomial coordinate stretching in the PML domain.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 In the **PML scaling curvature parameter** text field, type `3`.

Modify the **Typical Wave Speed for Perfectly Matched Layers** and **Transient Solver Settings** according to the problem formulation.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient (actd)**.

2 In the **Settings** window for **Pressure Acoustics, Transient**, locate the **Typical Wave Speed for Perfectly Matched Layers** section.

3 In the c_{ref} text field, type `c0`.

4 Locate the **Transient Solver Settings** section. Find the **Changes made to these settings only take effect when the default solver is generated.** subsection. In the $f_{\mathrm{max,\,sol}}$ text field, type `0.3`.

The frequency content of sources is often known or can be analyzed with an FFT. In this model the FFT of the source signal is depicted in Figure 6 created when analyzing the results. A maximal frequency of 0.3 (dimensionless) will capture the full signal information.

Transient Pressure Acoustics Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Transient (actd)** click **Transient Pressure Acoustics Model 1**.

2 In the **Settings** window for **Transient Pressure Acoustics Model**, locate the **Transient Pressure Acoustics Model** section.

3 From the c list, choose **User defined**. In the associated text field, type `c0`.

4 From the ρ list, choose **User defined**. In the associated text field, type `rho0`.

Specify the initial values for the pressure and its first time derivative.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Transient (actd)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the p text field, type `p_i`.

4 In the dp/dt text field, type `dp_i`.

Note that it is not required to impose any special boundary conditions on the outer boundary.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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

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

4 Select Domain 5 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 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. Select the **Maximum element size** check box.

5 In the associated text field, type `1.5`.

Create a mapped mesh in the PML domains.

Distribution 1

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

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundaries 4, 12, 15, and 20 only.

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

5 In the **Number of elements** text field, type `6`.

6 Click **Build All**.

STUDY 1

Step 1: Time Dependent

Solve the model from time t = 0 to 120 in steps of 1 (dimensionless time units).

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,1,120)`.

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

RESULTS

Surface

1 In the **Model Builder** window, expand the **Acoustic Pressure (actd)** node, then click **Surface**.

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

3 From the **Color table** list, choose **Wave**.

4 Select the **Symmetrize color range** check box.

The first default plot shows the acoustic pressure in the computational domain. You can see the pressure distribution at various times in Figure 2 and Figure 3.

Acoustic Pressure (actd)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (actd)**.

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

3 From the **Time (s)** list, choose **0**.

4 On the **Acoustic Pressure (actd)** toolbar, click **Plot**.

Create a **Cut Point 2D** data set to plot the pressure as a function of time.

Cut Point 2D 1

1 On the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **X** text field, type `20`.

4 In the **Y** text field, type `10`.

The plot in Figure 4 shows the comparison of the pressure to the analytical solution at the cut point.

1D Plot Group 2

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.

Point Graph 1

1 Right-click **Pressure at Point** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `p`.

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

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`COMSOL Model`

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Pressure at Point** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `p_a`.

4 Locate the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
Analytical

7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

9 In the **Number** text field, type `100`.

10 On the **Pressure at Point** toolbar, click **Plot**.

Create **Cut Line 2D** data sets to plot the pressure along the line at a given time.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **X** to `-W/2`.

4 In row **Point 2**, set **X** to `W/2`.

Cut Line 2D 2

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **X** to `-W/2+dW`.

4 In row **Point 2**, set **X** to `W/2-dW`.

The comparison of the pressure to the analytical solution along the x-axis for a given time is depicted in Figure 5.

1D Plot Group 3

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

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

3 Locate the **Data** section. From the **Data set** list, choose **None**.

Line Graph 1

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

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

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, select **40**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `p`.

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

8 In the **Expression** text field, type `x`.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
`COMSOL Model`

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **Pressure along Cut Line** and choose **Line Graph**.

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

3 From the **Data set** list, choose **Cut Line 2D 2**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, select **40**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `p_a`.

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

8 In the **Expression** text field, type `x`.

9 Locate the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
Analytical

12 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

13 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

14 In the **Number** text field, type 100.

15 On the **Pressure along Cut Line** toolbar, click **Plot**.

Pressure along Cut Line

Figure 6 shows the frequncy spectrum of the pressure wave at the cut point.

1D Plot Group 4

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Cut Point 2D 1**.

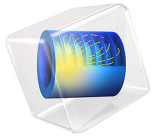
Point Graph 1

1 Right-click **Pressure at Point, FFT** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Frequency spectrum**.

4 On the **Pressure at Point, FFT** toolbar, click **Plot**.



Noise Radiation by a Compound Gear Train

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example illustrates the modeling of noise radiation from the housing of a gear train. First, a multibody analysis is performed in the time domain to compute the housing vibrations at the specified driver shaft speed. Then, an acoustic analysis is performed at a selected frequency to compute the sound pressure levels in the near, far, and exterior fields using the housing normal acceleration as a noise source.

Predicting the noise radiation from a dynamic system gives designers an insight into the behavior of moving mechanisms early in the design process. For example, consider a gearbox in which the change in the gear mesh stiffness causes sustained vibrations. These vibrations are transmitted to the gearbox housing through shafts and joints. The vibrating housing further transmits energy to the surrounding fluid, resulting in acoustic wave radiation.

Note: This model requires the Acoustics Module and the Multibody Dynamics Module.

Model Definition

The gear train enclosed in an acoustic domain is shown in Figure 1.The details of the compound gear train is shown in Figure 2.

This model is solved in two steps:

1. *Multibody analysis*: In the first part of the model, a transient analysis is performed to compute the dynamics of the gears as well as the acceleration levels on the gear train housing.
2. *Acoustic analysis*: In the second part of the model, a spherical domain enclosing the gear train housing is created. The computed accelerations on the housing are used as noise source for the acoustic domain. A frequency domain analysis is performed in order to compute the sound pressure levels outside the gear train.

Note: For the details of the multibody analysis of the gear train, see the model Vibrations in a Compound Gear Train in the Multibody Dynamics Module Application Library.

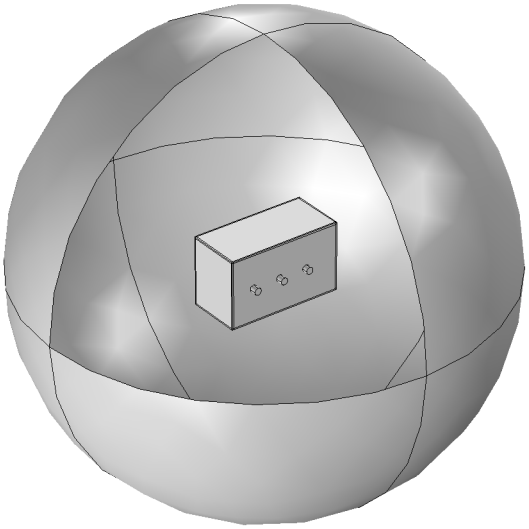


Figure 1: Model geometry of the gear train enclosed in a spherical acoustic domain.

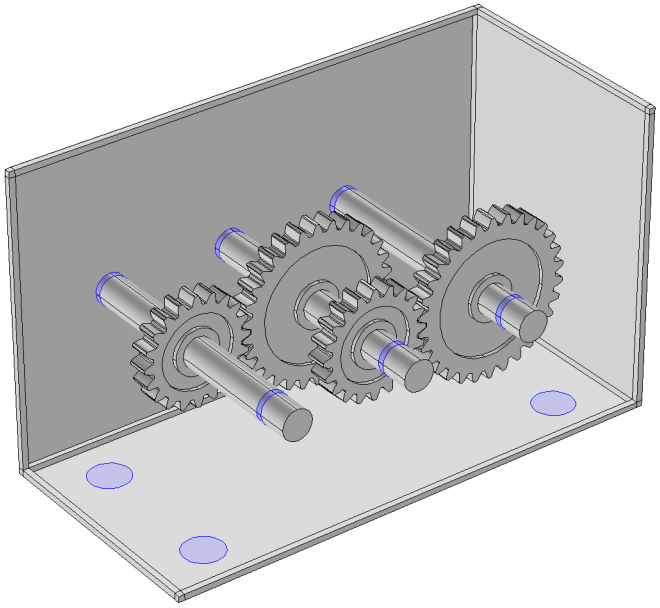


Figure 2: Details of the compound gear train.

MULTIBODY-ACOUSTICS COUPLING

One-way Coupling

When coupling multibody/structural and acoustics physics, one-way coupling can be assumed if the exterior fluid is air (or any other lighter fluid). This implies that the vibrations from the gear-train housing impacts the surrounding fluid, whereas the feedback from the acoustic waves to the structure is neglected. This model uses such an approach.

FFT Solver

The multibody dynamics is solved in the time domain, whereas the acoustics is solved in the frequency domain. Therefore the FFT solver is used to convert the housing accelerations from the time domain to the frequency domain.

DOMAIN EQUATIONS

This model solves the problem in the frequency domain using the Pressure Acoustics, Frequency Domain interface. The model equation is a slightly modified version of the Helmholtz equation for the acoustic pressure p:

$$\nabla \cdot \left(-\frac{\nabla p}{\rho}\right) - \frac{\omega^2 p}{c^2 \rho} = 0$$

where ρ is the density, c is the speed of sound, and ω is the angular frequency.

BOUNDARY CONDITIONS

The following boundary conditions are applied in the acoustic domain:

- Normal acceleration of the gear train housing is applied on the interior boundaries of the acoustics domain.

$$-\boldsymbol{n} \cdot \left\langle -\frac{\nabla p}{\rho} \right\rangle = a_n$$

  where a_n is the normal acceleration.
- A spherical wave radiation condition is applied on the exterior boundaries of the acoustic domain.
- A far-field calculation is added on the exterior boundaries of the acoustic domain to compute the sound pressure levels in the far field.

MODEL PARAMETERS

The domain material is chosen as air. The following parameters are used in the model:

- The excitation frequency is taken as 2000 Hz.
- The size of enclosing sphere is taken as 0.5 m.
- Far field results are evaluated at a distance of 2 m from the center.

Results and Discussion

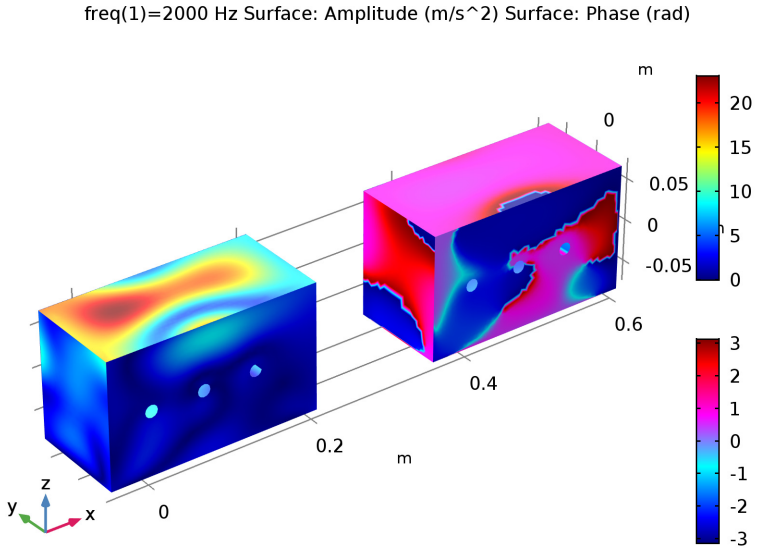


Figure 3: Amplitude and phase of normal acceleration of housing

Figure 3 shows the amplitude (left) and phase (right) of the normal acceleration of the housing at 2000 Hz. The top part vibrates with higher velocity as compared to the rest of the housing. This part also has constant phase as indicated by the color distribution in the second plot.

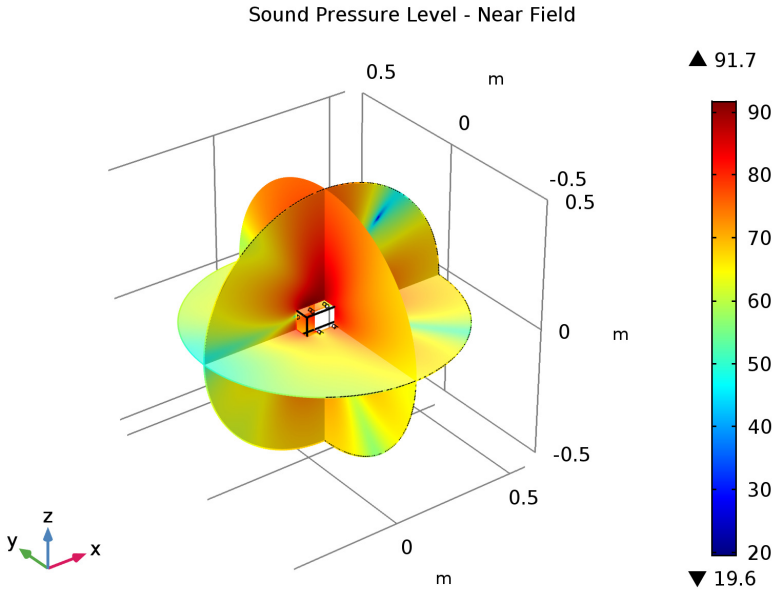


Figure 4: Sound Pressure Level (dB) in the near-field region at 2000 Hz.

Figure 4 shows the sound pressure level (SPL) in the acoustic domain. Different SPL values can be seen at different locations in space, with a maximum at the top of the gear train. The SPL field on the housing/casing surface is shown in Figure 5.

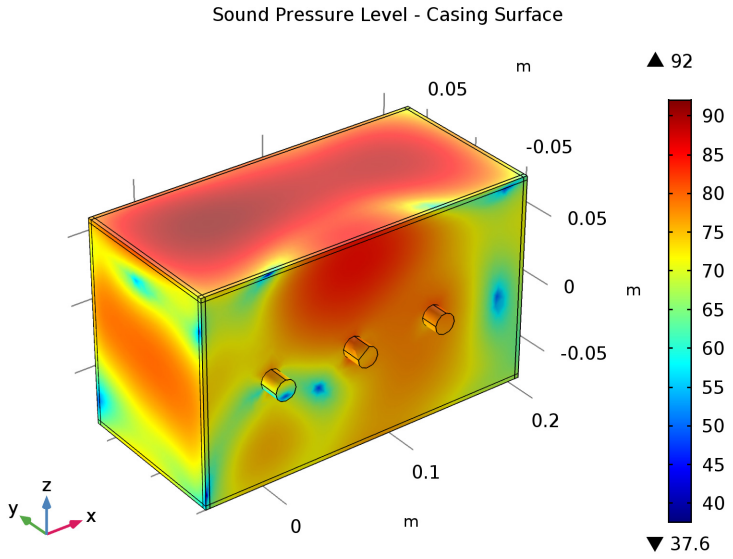


Figure 5: Sound Pressure Level (dB) on the casing surface at 2000 Hz.

The sound pressure levels in the far field at a distance of 2 m are shown in Figure 6, Figure 7, and Figure 8. All these plots show the sound pressure levels in different planes.

Figure 9 shows a three-dimensional plot of the sound pressure level, giving an idea of the dominant directions of noise radiation.

It is also possible to plot the acoustic wave propagation in a domain exterior to the modeled spherical domain. Figure 10 shows a plot of the acoustics waves propagating in the exterior domain.

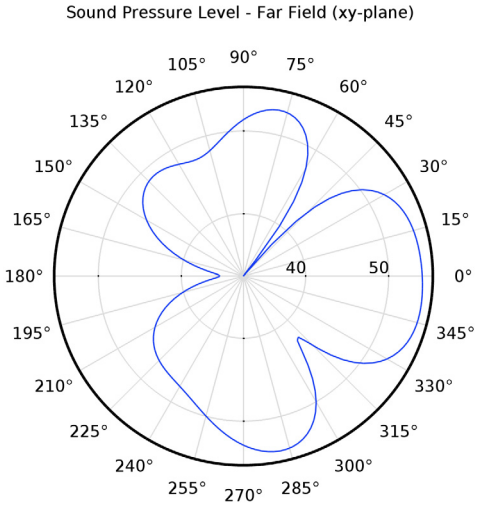


Figure 6: Far-field sound pressure level in the xy-plane at a distance of 2 m.

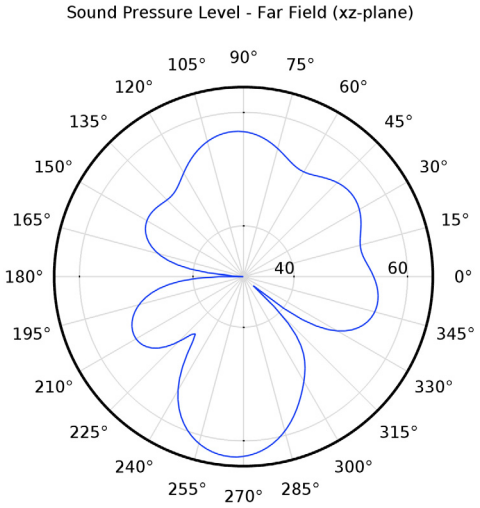


Figure 7: Far-field sound pressure level in the xz-plane at a distance of 2 m.

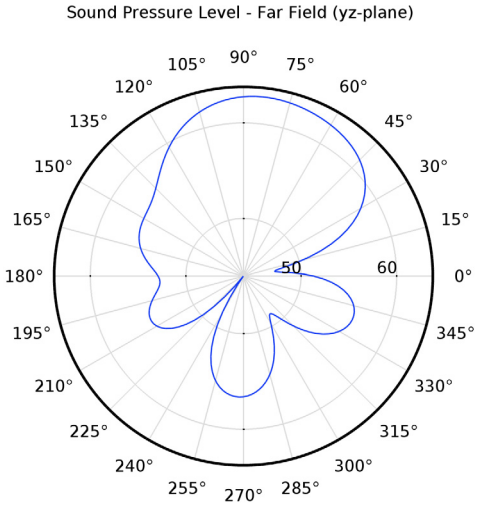


Figure 8: Far-field sound pressure level in the yz-plane at a distance of 2 m.

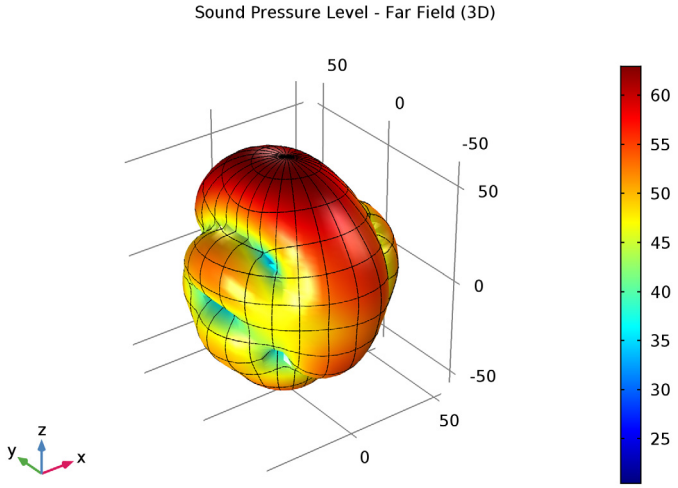


Figure 9: Far-field sound pressure level at a distance of 2 m.

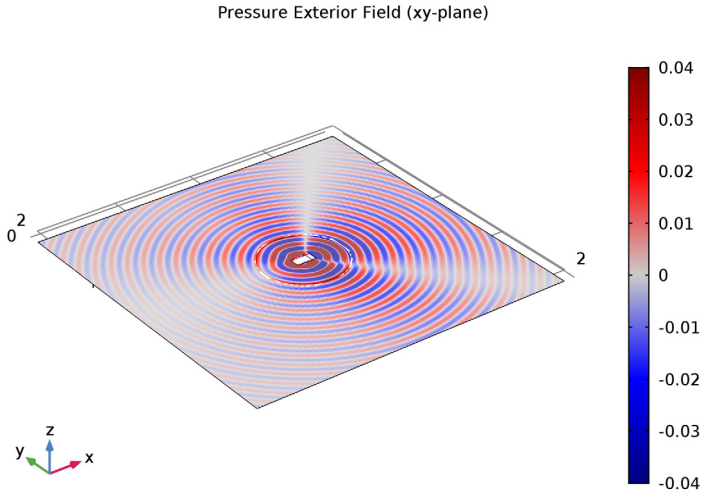


Figure 10: The acoustic wave propagation in the exterior domain.

Notes About the COMSOL Implementation

- The acoustics is set up in a separate component so that the multibody model setup is unaffected. A General Extrusion coupling operator is used to map the acceleration values between the geometries of the two components.
- The normal acceleration of the housing is in the time domain. A Time to Frequency FFT study is used to convert it to the frequency domain.
- The presence of Multibody Dynamics physics, by default, forces the solver to be nonlinear. Hence it is manually set to linear for the acoustic analysis.

Application Library path: `Acoustics_Module/Vibrations_and_FSI/gear_train_noise`

Modeling Instructions

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

From the Application Libraries root, browse to the folder `Multibody_Dynamics_Module/Tutorials` and double-click the file `gear_train.mph`.

ROOT

Enter the required set of parameters for the acoustic analysis.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, expand the **Global Definitions** node, then click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
x0	0.07[m]	0.07 m	Sphere offset
R	0.5[m]	0.5 m	Sphere radius
f0	2000[Hz]	2000 Hz	Frequency of interest

COMPONENT 2 (COMP2)

Create a selection for the exterior boundaries of the gear train housing. The acceleration computed on these boundaries is used as a noise source for the acoustic analysis.

DEFINITIONS

Explicit 7

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `14,38,46,53,65,132,219,220,225,455,456,461,919,920,925` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

8 Select the **Group by continuous tangent** check box.

General Extrusion 1 (genext1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.

2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.

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

4 From the **Selection** list, choose **Explicit 7**.

5 Click to expand the **Advanced** section. From the **Mesh search method** list, choose **Closest point**.

STUDY 1

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

2 In the **Settings** window for **Study**, type `Study 1 - Gear 2D` in the **Label** text field.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - Gear Train` in the **Label** text field.

3 On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

GEOMETRY 3

In the **Model Builder** window, under **Component 3 (comp3)** click **Geometry 3**.

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

Create a domain surrounding the gear train assembly for the acoustic analysis. In order to do this, add a spherical domain of a finite size and remove the gear train assembly from it.

Convert to Solid 1 (csol1)

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

2 Click the **Transparency** button on the **Graphics** toolbar.

3 Click in the **Graphics** window and then press Ctrl+A to select all objects.

4 In the **Settings** window for **Convert to Solid**, click **Build Selected**.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

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

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

4 Locate the **Position** section. In the **x** text field, type `x0`.

5 Click **Build Selected**.

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

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **sph1** only.

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

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **csol1** only.

6 Click **Build Selected**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1: Gear 2D**, and **Study 2: Gear Train**.

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

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

DEFINITIONS

Explicit 8

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundary 1 only.

5 Select the **Group by continuous tangent** check box.

6 In the **Label** text field, type `Outer Boundaries`.

Explicit 9

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select the **All boundaries** check box.

5 In the **Label** text field, type `All Boundaries`.

Difference 1

1 On the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **All Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **Add**.

9 In the **Add** dialog box, select **Outer Boundaries** in the **Selections to subtract** list.

10 Click **OK**.

11 In the **Settings** window for **Difference**, type `Inner Boundaries` in the **Label** text field.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Spherical Wave Radiation 1

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

2 In the **Settings** window for **Spherical Wave Radiation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outer Boundaries**.

4 Locate the **Spherical Wave Radiation** section. Specify the $\mathbf{r}_0$ vector as

x0	x
0	y
0	z

The normal acceleration computed in **Study 2 - Gear Train** is used as an acceleration source for the acoustic analysis.

Normal Acceleration 1

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

2 In the **Settings** window for **Normal Acceleration**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inner Boundaries**.

4 Locate the **Normal Acceleration** section. From the **Type** list, choose **Acceleration**.

5 Specify the $\mathbf{a}_0$ vector as

comp2.genext1(u2)/7e-3[s^2]	x
comp2.genext1(v2)/7e-3[s^2]	y
comp2.genext1(w2)/7e-3[s^2]	z

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 In the **Settings** window for **Far-Field Calculation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outer Boundaries**.

4 Locate the **Far-Field Calculation** section. From the **Type of integral** list, choose **Full integral**.

5 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

6 Click to expand the **Advanced settings** section. Locate the **Advanced Settings** section. Clear the **Use polynomial-preserving recovery for the normal gradient** check box.

MESH 3

The mesh size is chosen such that there are 5 elements per wavelength at the frequency of interest.

Size

1 In the **Model Builder** window, under **Component 3 (comp3)** right-click **Mesh 3** and choose **Free Tetrahedral**.

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 `343[m/s]/f0/5`.

5 In the **Model Builder** window, click **Mesh 3**.

6 Click **Build All**.

7 In the **Model Builder** window, expand the **Mesh 3** node.

Boundary Layer Properties

1 Right-click **Mesh 3** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outer Boundaries**.

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type `1`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `343[m/s]/f0/5/10`.

7 In the **Model Builder** window, click **Mesh 3**.

8 Click **Build All**.

ADD STUDY

1 On 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 **Empty Study**.

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

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

STUDY 3

In the **Settings** window for **Study**, type `Study 3 - Acoustics` in the **Label** text field.

Transform the time-dependent acceleration data to the frequency domain using a **Time to Frequency FFT** study step, then use this data as input for the **Frequency Domain** study step.

Time to Frequency FFT

On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>**
Time to Frequency FFT.

STUDY 3 - ACOUSTICS

Step 1: Time to Frequency FFT

1 In the **Settings** window for **Time to Frequency FFT**, locate the **Study Settings** section.

2 From the **Input study** list, choose **Study 2 - Gear Train, Time Dependent**.

3 In the **End time** text field, type `7e-3`.

4 In the **Maximum output frequency** text field, type `1/(7e-5)`.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve** check box for **Multibody Dynamics (mbd)** and **Pressure Acoustics, Frequency Domain (acpr)**.

Step 2: Frequency Domain

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain**.

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

3 In the **Frequencies** text field, type `f0`.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve** check box for **Multibody Dynamics (mbd)** and **Multibody Dynamics 2 (mbd2)**.

5 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Study 3 - Acoustics, Time to Frequency FFT**.

8 From the **Selection** list, choose **All**.

Set the solver to linear as by default Multibody Dynamics sets it to nonlinear.

Solution 3 (sol3)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 From the **Linearity** list, choose **Linear**.

5 In the **Model Builder** window, under **Study 3 - Acoustics>Solver Configurations> Solution 3 (sol3)** click **FFT Solver 1**.

6 In the **Settings** window for **FFT Solver**, locate the **General** section.

7 From the **Defined by study step** list, choose **User defined**.

8 From the **Input** list, choose **Second time derivative**.

9 In the **Model Builder** window, click **Study 3 - Acoustics**.

10 In the **Settings** window for **Study**, locate the **Study Settings** section.

11 Clear the **Generate default plots** check box.

12 On the **Study** toolbar, click **Compute**.

RESULTS

Study 3 - Acoustics/Solution 3 (8) (sol3)

In the **Model Builder** window, expand the **Results>Data Sets** node.

Selection

1 Right-click **Study 3 - Acoustics/Solution 3 (8) (sol3)** and choose **Duplicate**.

2 On the **Results** toolbar, click **Selection**.

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

4 From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Inner Boundaries**.

6 Select the **Propagate to lower dimensions** check box.

Selection

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Study 3 - Acoustics/Solution 3 (9) (sol3)** and choose **Duplicate**.

2 In the **Model Builder** window, expand the **Study 3 - Acoustics/Solution 3 (10) (sol3)** node, then click **Selection**.

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

4 Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `5-7,10-14,16-27,29-39,43-45,48-49,53-55,57,59,67-69,71,73-96` in the **Selection** text field.

7 Click **OK**.

Selection

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Study 3 - Acoustics/Solution 3 (10) (sol3)** and choose **Duplicate**.

2 In the **Model Builder** window, expand the **Study 3 - Acoustics/Solution 3 (11) (sol3)** node, then click **Selection**.

3 Select Boundaries 5–39 and 74–96 only.

Grid 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Grid>Grid 3D**.

2 In the **Settings** window for **Grid 3D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3 - Acoustics/Solution 3 (8) (sol3)**.

4 Locate the **Parameter Bounds** section. Find the **First parameter** subsection. In the **Minimum** text field, type `-2`.

5 In the **Maximum** text field, type `2`.

6 Find the **Second parameter** subsection. In the **Minimum** text field, type `-2`.

7 In the **Maximum** text field, type `2`.

8 Find the **Third parameter** subsection. In the **Maximum** text field, type `0`.

9 Click to expand the **Resolution** section. In the **x resolution** text field, type `100`.

10 In the **y resolution** text field, type `100`.

11 In the **z resolution** text field, type `2`.

Use the following instructions to plot the amplitude and phase of the normal acceleration of the gear train housing as shown in Figure 3.

3D Plot Group 11

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

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3 - Acoustics/Solution 3 (11) (sol3)**.

4 Right-click **3D Plot Group 11** and choose **Rename**.

5 In the **Rename 3D Plot Group** dialog box, type `Housing normal acceleration: amplitude and phase` in the **New label** text field.

6 Click **OK**.

Surface 1

1 Right-click **Housing normal acceleration: amplitude and phase** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `abs(acpr.nacc)`.

Surface 2

1 In the **Model Builder** window, under **Results** right-click **Housing normal acceleration: amplitude and phase** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `arg(acpr.nacc)`.

Deformation 1

1 Right-click **Results>Housing normal acceleration: amplitude and phase>Surface 2** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **X component** text field, type `0.4`.

4 In the **Y component** text field, type `0`.

5 In the **Z component** text field, type `0`.

6 Locate the **Scale** section. Select the **Scale factor** check box.

7 In the associated text field, type `1`.

Surface 2

1 In the **Model Builder** window, under **Results>Housing normal acceleration: amplitude and phase** click **Surface 2**.

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

3 From the **Color table** list, choose **Disco**.

Housing normal acceleration: amplitude and phase

1 In the **Model Builder** window, under **Results** click **Housing normal acceleration: amplitude and phase**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

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

4 In the **Title** text area, type `freq(1)=2000 Hz Surface: Amplitude (m/s^2) Surface: Phase (rad)`.

5 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

6 From the **Position** list, choose **Right double**.

7 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

8 On the **Housing normal acceleration: amplitude and phase** toolbar, click **Plot**.

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

10 Click the **Transparency** button on the **Graphics** toolbar.

Use the following instructions to plot the spatial distribution of the sound pressure level in the spherical domain and on the surface of gear train housing as shown in Figure 4 and Figure 5, respectively.

3D Plot Group 12

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Acoustics/ Solution 3 (8) (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Sound Pressure Level - Near Field`.

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Slice 1

1 Right-click **SPL Near Field** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.Lp`.

4 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

Slice 2

1 Right-click **Results>SPL Near Field>Slice 1** and choose **Duplicate**.

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

3 From the **Plane** list, choose **XY-planes**.

4 In the **Planes** text field, type `1`.

5 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Slice 3

1 Right-click **Results>SPL Near Field>Slice 2** and choose **Duplicate**.

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

3 From the **Plane** list, choose **ZX-planes**.

4 In the **Planes** text field, type `1`.

5 On the **SPL Near Field** toolbar, click **Plot**.

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

3D Plot Group 13

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Acoustics/ Solution 3 (9) (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Sound Pressure Level - Casing Surface`.

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 Right-click **SPL Casing Surface** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.Lp`.

4 On the **SPL Casing Surface** toolbar, click **Plot**.

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

Follow the instructions below to plot the spatial distribution of the sound pressure level in the far field region as shown in Figure 6, Figure 7, Figure 8, and Figure 9, respectively.

Polar Plot Group 14

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

2 In the **Settings** window for **Polar Plot Group**, type `Polar SPL xy-plane` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Acoustics/ Solution 3 (8) (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Sound Pressure Level - Far Field (xy-plane)`.

Far Field 1

1 On the **Polar SPL xy-plane** toolbar, click **More Plots** and choose **Far Field**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type `180`.

4 Find the **Center** subsection. In the **x** text field, type `x0`.

5 Find the **Evaluation distance** subsection. In the **Radius** text field, type `4*R`.

6 On the **Polar SPL xy-plane** toolbar, click **Plot**.

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

Polar SPL xy-plane 1

1 In the **Model Builder** window, under **Results** right-click **Polar SPL xy-plane** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Polar SPL xz-plane` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Sound Pressure Level - Far Field (xz-plane)`.

Far Field 1

1 In the **Model Builder** window, expand the **Polar SPL xy-plane 1** node, then click **Results> Polar SPL xz-plane>Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Normal** subsection. In the **y** text field, type `1`.

4 In the **z** text field, type `0`.

5 On the **Polar SPL xz-plane** toolbar, click **Plot**.

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

Polar SPL xz-plane 1

1 In the **Model Builder** window, under **Results** right-click **Polar SPL xz-plane** and choose **Duplicate**.

2 In the **Settings** window for **Polar Plot Group**, type `Polar SPL yz-plane` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Sound Pressure Level - Far Field (yz-plane)`.

Far Field 1

1 In the **Model Builder** window, expand the **Polar SPL xz-plane 1** node, then click **Results> Polar SPL yz-plane>Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Normal** subsection. In the **y** text field, type `0`.

4 In the **x** text field, type `1`.

5 On the **Polar SPL yz-plane** toolbar, click **Plot**.

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

3D Plot Group 17

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Acoustics/ Solution 3 (8) (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Sound Pressure Level - Far Field (3D)`.

Far Field 1

1 On the **3D Polar SPL** toolbar, click **More Plots** and choose **Far Field**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of elevation angles** text field, type `50`.

4 In the **Number of azimuth angles** text field, type `50`.

5 Find the **Sphere** subsection. From the **Sphere** list, choose **Manual**.

6 In the **X** text field, type `x0`.

7 In the **Radius** text field, type `4*R`.

8 Locate the **Coloring and Style** section. From the **Grid** list, choose **Fine**.

9 On the **3D Polar SPL** toolbar, click **Plot**.

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

Finally, plot the spatial distribution of the sound pressure level in the exterior field as shown in the Figure 10.

3D Plot Group 18

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Grid 3D 1**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Pressure Exterior Field (xy-plane)`.

Surface 1

1 Right-click **Pressure Exterior Field** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `if(sqrt((x-x0)^2+y^2+z^2)>R,pfar(x,y,z), NaN)`.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** text field, type `-.04`.

6 In the **Maximum** text field, type `.04`.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

8 On the **Pressure Exterior Field** toolbar, click **Plot**.

Slice 1

1 In the **Model Builder** window, under **Results** right-click **Pressure Exterior Field** and choose **Slice**.

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

3 From the **Data set** list, choose **Study 3 - Acoustics/Solution 3 (9) (sol3)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.

5 In the **Planes** text field, type `1`.

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

7 On the **Pressure Exterior Field** toolbar, click **Plot**.

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

SPL Casing Surface

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

3D Plot Group 19

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Acoustics/Solution 3 (10) (sol3)**.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

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

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

3 Select the **Symmetrize color range** check box.

Surface 2

1 In the **Model Builder** window, under **Results** right-click **Thumbnail** and choose **Surface**.

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

3 From the **Data set** list, choose **Study 2 - Gear Train/Solution 2 (4) (sol2)**.

4 From the **Time (s)** list, choose **0.00301**.

5 Locate the **Expression** section. In the **Expression** text field, type `mbd2.disp`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraAustralis**.

Deformation 1

1 Right-click **Results>Thumbnail>Surface 2** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `1`.

Isosurface 1

1 In the **Model Builder** window, under **Results** right-click **Thumbnail** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, locate the **Levels** section.

3 In the **Total levels** text field, type `10`.

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Filter 1

1 Right-click **Results>Thumbnail>Isosurface 1** and choose **Filter**.

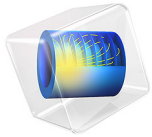
2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `y>0`.

Thumbnail

1 In the **Model Builder** window, under **Results** click **Thumbnail**.

2 On the **Thumbnail** toolbar, click **Plot**.



Generic 711 Coupler—An Occluded Ear-Canal Simulator

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of an 711 coupler[1], an occluded ear-canal simulator that follows the specifications given in the IEC 60318-4 international standard (Ref. 1). A coupler is a device for measuring the acoustic output of sound sources with a calibrated microphone. The microphone is coupled to the source by a cavity of known shape and volume (see Figure 1 bottom). The 711 coupler approximates the acoustic transfer impedance of the inner part of the ear canal from the tip of an ear plug (ear insert or ear mould) located at the reference plane to the eardrum (see Figure 1 top right). It is thus a device that is intended to have the same acoustic properties as the average occluded human ear-canal and eardrum system, approximately from the second bend in the ear canal to the eardrum. The 711 coupler is intended for measurements of hearing aids and earphones that are coupled to the ear by means of an insert earphone.

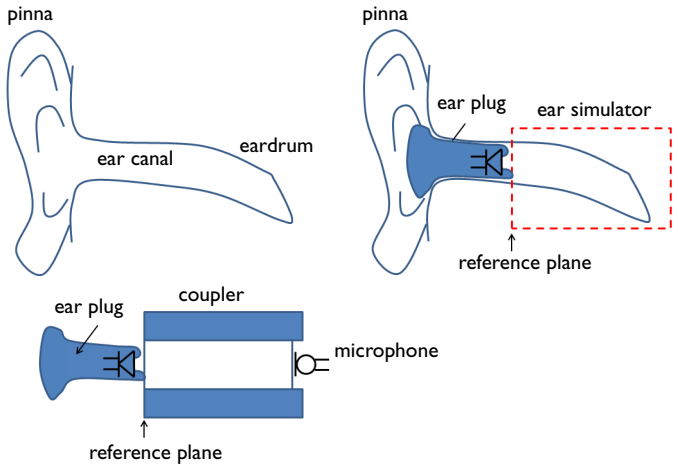


Figure 1: (top left) sketch of the ear with pinna, ear canal, and eardrum, (top right) ear canal occluded by an earplug with a loudspeaker, the occluded-ear canal is the part that the coupler is intended to model, (bottom) ear plug placed at the reference plane of the acoustic coupler including the location of the recording microphone.

Using a coupler for measurement of the acoustic response enables standardized measurements and comparisons. Coupler measurements on ear-canal simulators are also often used for prototype development and testing. Such measurements do, however, not include leakage between ear mold and ear canal. As the coupler represents a normal average human ear it does not mimic the large acoustic performance variations that exist

1. Here 711 are the last digits in the IEC 60711 standard (1981), which is the standard that the current updated IEC 60318-4 replaces. This has given the occluded ear-canal simulator its commonly used name.

between individual ears (see for example, Ref. 2).

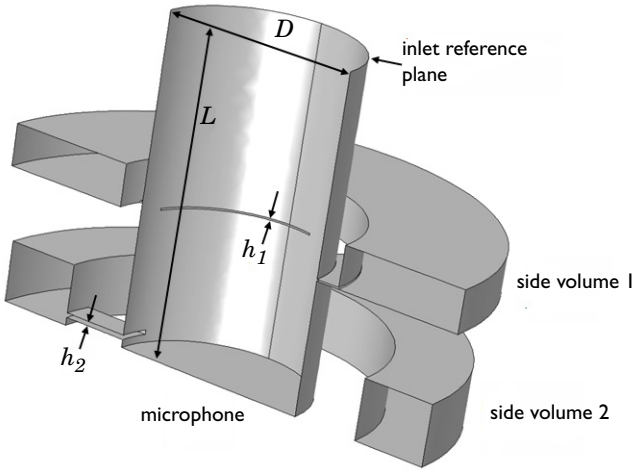


Figure 2: Sketch of the geometry used in the model (cut in half due to symmetry). The main volume is a cylinder of length L and diameter D. The two side volumes are attached to the main volume via slender slits of height h_1 and h_2.

Model Definition

GEOMETRY

The coupler is made of hard nonporous material and it is terminated at one end by a measurement microphone. This end corresponds to the eardrum (the tympanic membrane) and the measured microphone pressure should correspond the to that perceived by the eardrum. The human eardrum has a nontrivial acoustic behavior (see Ref. 3); the coupler has to account for

1. the acoustic energy losses at the eardrum, and
2. the acoustics of the cylinder like volume of the ear canal.

In order for the coupler to do this it is constructed as a main cylinder of length L and diameter D with two attached side volumes, see Figure 2. The side volumes are connected to the main volume via shallow slits of height h_1 and h_2. The coupler geometry meets the requirements of the IEC 60318-4 international standard (Ref. 1) and besides certain details corresponds to the Brüel & Kjær Ear Simulator Type 4157.

The inclusion of the side volumes and the slits is necessary to mimic the complex eardrum mechanical losses using an acoustic system. In the coupler the losses are mainly due to the high thermal and viscous damping in the slits. The diameter D of the main cylinder is given

by the standard and is 7.5 mm. The length of the cylinder is prescribed by the IEC standard to be such as to produce a half-wavelength resonance at around 13.5 kHz. In this model L = 12.5 mm, which gives a resonance at 13.8 kHz. The slit heights are h_1 = 69 μm and h_2 = 170 μm.

THERMOVISCOUS ACOUSTICS

As the thermal and viscous losses are important in the slits the finite element model is set up in COMSOL using the Thermoviscous Acoustics, Frequency Domain interface. This enables the direct inclusion and modeling of the thermal and viscous losses in the slits. The losses arise in the viscous and thermal boundary layers that are characterized by the length scales

$$\delta_v = \sqrt{\frac{2\mu}{\omega\rho}} \qquad \delta_{th} = \sqrt{\frac{2k}{\omega\rho C_p}}$$

where ω is the angular frequency, μ is the dynamic viscosity, k is the coefficient of thermal conduction, ρ is density, and C_p is the specific heat at constant pressure. In the modeled frequency range, from 100 Hz to 20 kHz, this yields length scales from 220 μm to 15 μm (for air at 23° C). Thermal and viscous losses are hence important in most of the frequency range when comparing these length scales to the slit heights. However, they need not be included in the main cylinder and the side volumes, where pressure acoustics may be used. The pressure acoustics and the thermoviscous acoustics domains are coupled using the built in Acoustics-Thermoviscous Acoustics Boundary multiphysics coupling found under the Multiphysics node in the model tree.

COUPLER CHARACTERIZATION

The 711 coupler is characterized in the IEC 60318-4 standard in terms of its transfer impedance Z_{trans} and the microphone response L_0 (for a constant volume displacement source):

$$Z_{trans} = \frac{p_{mic}}{Q_{in}}$$
$$L_0 = 10\log\left(\frac{\langle p_{mic}\rangle^2}{p_{ref}^2}\right) = 10\log(\langle p_{mic}\rangle^2) - L_{ref} \qquad (1)$$

where Q_{in} is the volume flow rate at the inlet reference plane, $\langle p_{mic}\rangle$ is the root mean square (rms) pressure at the measurement microphone, p_{ref} is a reference pressure (here the rms pressure at 500 Hz), and L_{ref} is the corresponding reference level. The transfer impedance of a coupler is easily measured using, for example, a microphone as a sound

source (it has a high output impedance and thus a nearly constant Q_{in}). Moreover, the pressure is directly determined by the measurement microphone. In a real ear, Z_{trans} is somewhat more complicated to measure because it requires the insertion of a probe tube into the ear to measure p_{mic} (now the pressure at the eardrum). The transfer impedance and microphone response are specified in the frequency range 100 Hz to 10 kHz. Above 10 kHz the 711 coupler does not simulate a human ear.

Generally, to mimic an ear with a coupler one should require both systems to have the same two-port parameters. These are four parameters that relate pressure and volume velocity at the inlet and outlet. If only the acoustic input load and the pressure at the eardrum are of interest, the transfer impedance characterizes the coupler together with the input impedance Z_{in}. The latter is defined as

$$Z_{in} = \frac{p_{in}}{Q_{in}} \tag{2}$$

where p_{in} is the pressure at the inlet reference plane.

Modeling an acoustic coupler is of interest for design and for optimizing the acoustic response with respect to different input systems (see for example Ref. 5). The coupler cannot always be modeled as a simple acoustic load (impedance). Interactions may exist between the sound source outlets (hearing aid ear-mold or earplug) and the coupler acoustics. Nonplane waves may, for example, propagate into the coupler.

BOUNDARY CONDITIONS

At the inlet reference plane a constant volume source is applied by specifying the inward normal displacement d_n of the boundary

$$d_n = d_0 \tag{3}$$

where d_0 is the sound source displacement.

At the location of the measurement microphone, an impedance corresponding to the mechanical properties of a Brüel & Kjær 4192 microphone is specified (see Ref. 4 pp 6–18). The impedance is given by

$$Z_{mic} = \frac{1}{i\omega C_{mic}} + R_{mic} + i\omega L_{mic} \tag{4}$$

where $C_{mic} = 0.62 \cdot 10^{-13}$ m^5/N is the acoustic compliance, $R_{mic} = 119 \cdot 10^6$ N·s/m^5 is the acoustic resistance, and $L_{mic} = 710$ kg/m^4 is the acoustic mass. This is modeled using the built-in RCL option in the impedance boundary condition.

To reduce the model size, model only half of the geometry and use symmetry conditions on both the pressure acoustic and thermovicous acoustic domains.

Results and Discussion

Figure 3 depicts the transfer impedance for the system as a function of frequency. The modeled system is seen to comply well with the IEC standard curve.

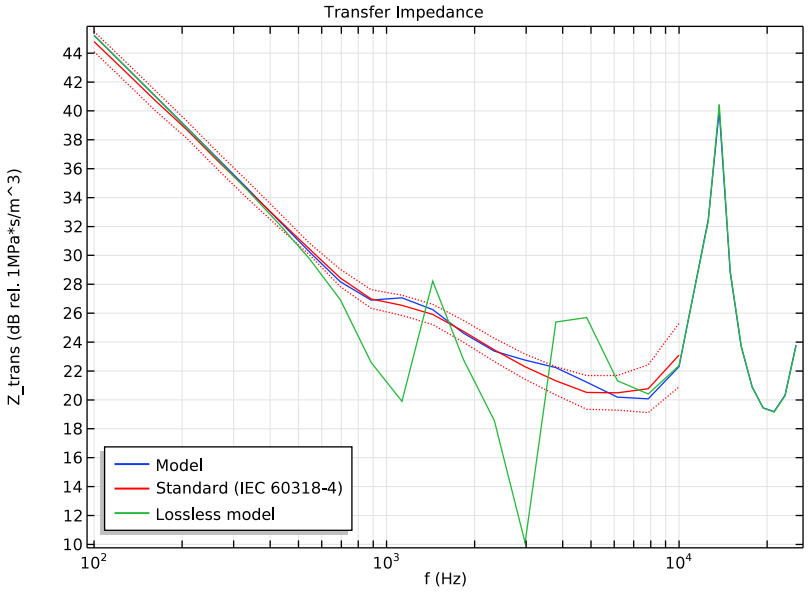


Figure 3: Transfer impedance as function of frequency for the model (blue line), the IEC standard curve (red line) including upper and lower tolerances (red dotted line), and the system modeled without losses (green line).

Also evident from the graph is that it is important to use thermoviscous acoustics to model such acoustic systems with small geometrical dimensions. In the frequency range where the acoustic boundary-layer thickness is comparable to the small slit heights, the lossless model is completely off because all the resonances are undamped. Above about 10 kHz the thermal and viscous losses on the acoustic boundary are much less pronounced and the system is effectively lossless.

Figure 4 depicts the microphone response measured for a constant displacement source, again comparing model results to the IEC standard and the lossless model.

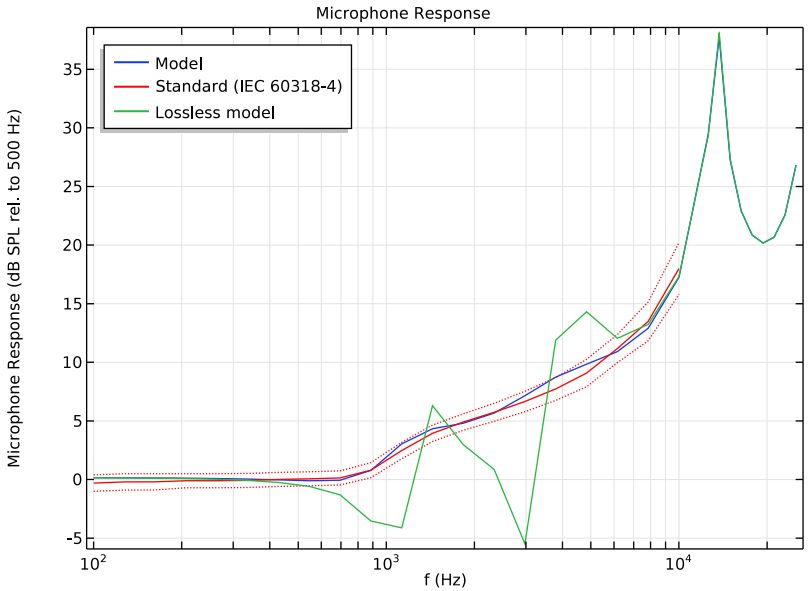


Figure 4: Microphone response as function of frequency for the model (blue line), the IEC standard curve (red line) including upper and lower tolerances (red dotted line), and the system modeled without losses (green line).

The input impedance of the system derived at the reference plane is shown in Figure 5 where the model results are compared to the results of the fully lossless model.

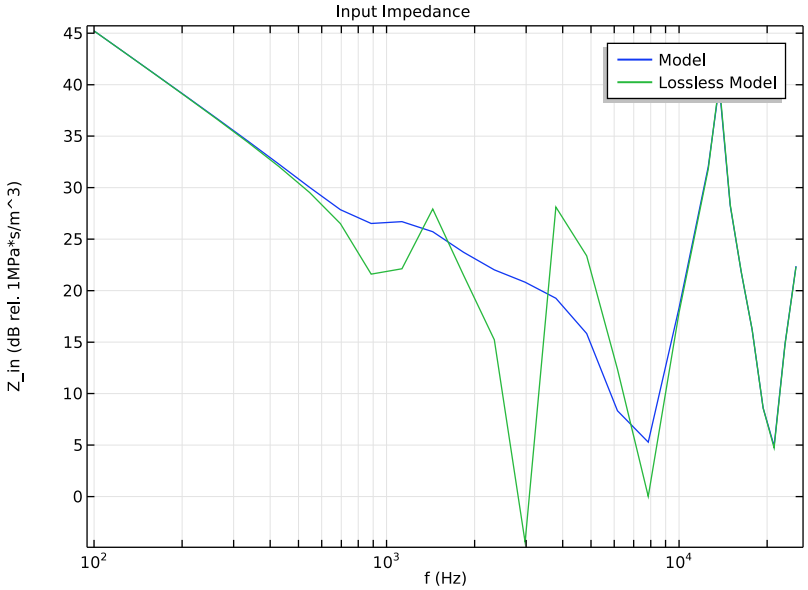


Figure 5: Input impedance as function of frequency for the model (blue line) and the system modeled without losses (green line).

Figure 6, Figure 7, and Figure 8 depict the pressure distribution inside the 711 coupler at frequencies of 25.1 kHz, 13.7 kHz, and 885 Hz, respectively. The first figure represents the standing wave mode inside the inner tube and the second figure the half wave standing mode. The last figure represents a Helmholtz-like resonance in the lower side volume.

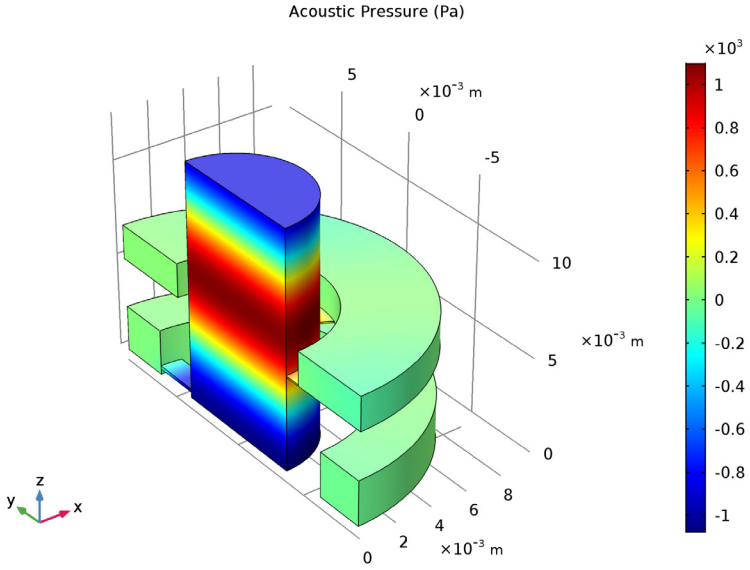


Figure 6: Instantaneous pressure distribution at f = 25.1 kHz.

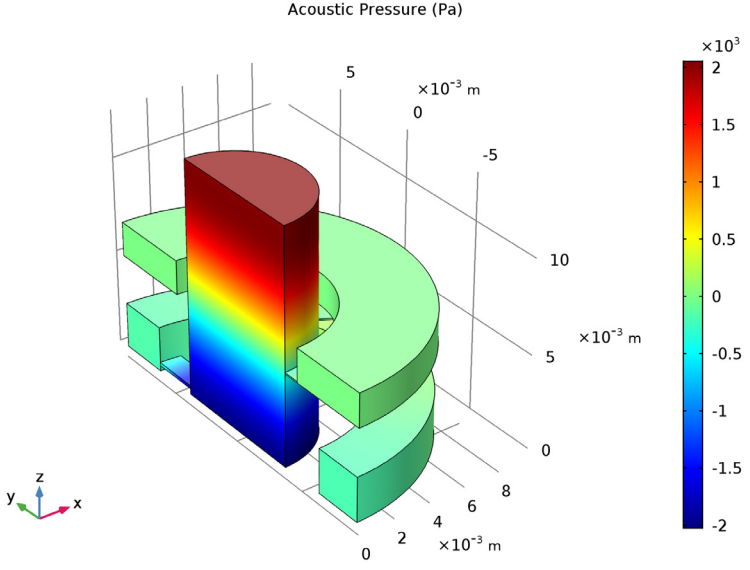


Figure 7: Instantaneous pressure distribution at f = 13.7 kHz.

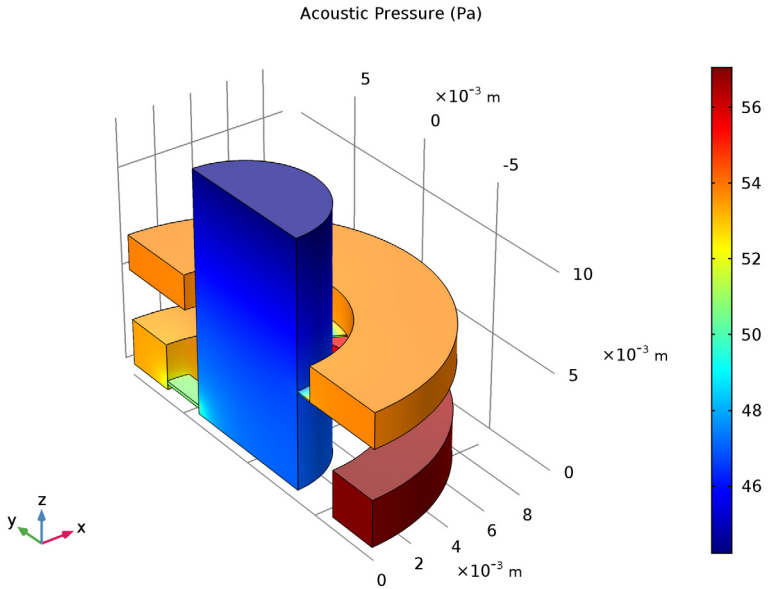


Figure 8: Instantaneous pressure distribution at f = 885 Hz.

References

1. IEC 60318-4, *Electroacoustics—Simulators of human head and ear—Part 4: Occluded-ear simulator for the measurement of earphones coupled to the ear by means of ear inserts*, edition 1.0, 2010.

2. M.R. Stinson and B.W. Lawton, "Specification of the geometry of the human ear canal for the prediction of sound-pressure level distribution," *J. Acoust. Soc. Am.*, vol. 85, pp. 2492–2503, 1989.

3. M.R. Stinson, "The spatial distribution of sound pressure within scaled replicas of the human ear canal," *J. Acoust. Soc. Am.*, vol. 78, no. 5, pp. 1596–1602, 1985.

4. Brüel and Kjær, *Microphone Handbook: For the Falcon Range Microphone Products*, Technical Documentation, 1995.

5. B.L. Zhang, S. Jønsson, A. Schuhmacher, and L.B. Nielsen, *A Combined BEM/FEM Acoustic Model of an Occluded Ear Simulator*, InterNoise 2004, Prague, Czech Republic, 2004.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/generic_711_coupler`

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics> Thermoviscous Acoustics, Frequency Domain (ta)**.
5. Click **Add**.
6. Click **Done**.

GLOBAL DEFINITIONS

Load the parameters for the model. The list of parameters include the maximal mesh size, the microphone impedance parameters, and other reference values.

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `generic_711_coupler_parameters.txt`.

   To make the data for the transfer impedance and microphone response available in the model, create six interpolation functions. The data are located in two text files and comprise the nominal values as well as upper and lower tolerances as defined in the IEC 60318-4 standard (Table 1 and Table B.1 in Ref. 1).

Interpolation 1 (int1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `generic_711_coupler_transfer_impedance.txt`.

6 In the **Number of arguments** text field, type `1`.

7 Click **Import**.

8 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
`int_trans`	1
`int_trans_upper`	2
`int_trans_lower`	3

Interpolation 2 (int2)

1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `generic_711_coupler_mic_response.txt`.

6 In the **Number of arguments** text field, type `1`.

7 Click **Import**.

8 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
`int_mic`	1
`int_mic_upper`	2
`int_mic_lower`	3

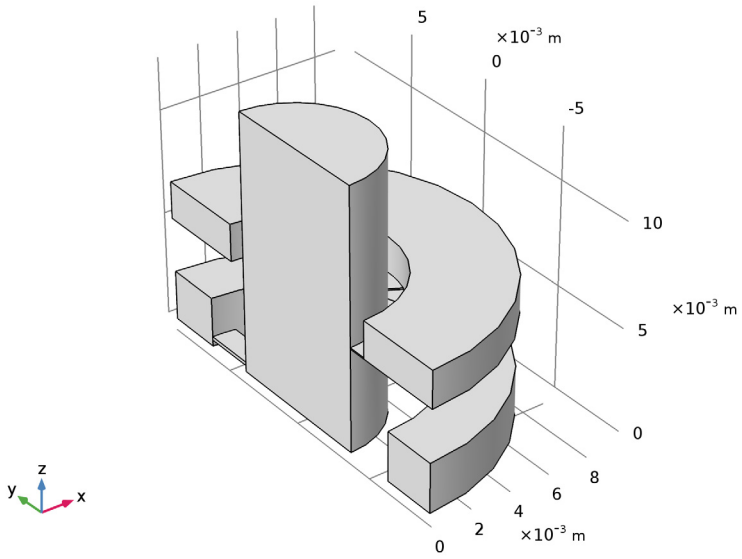
GEOMETRY 1

Import the model geometry from file by following these steps.

Import 1 (imp1)

1. On the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `generic_711_coupler.mphbin`.
5. Click **Import**.
6. Click the **Zoom Extents** button on the **Graphics** toolbar.

The figure below shows the model geometry.



DEFINITIONS

Load a set of variables that define the microphone impedance (Equation 4), the transfer impedance (Equation 1), and the input impedance (Equation 3) from a file. Add operators to integrate values across the reference plane (the inlet) and the microphone plane.

Variables 1

1. On the **Home** toolbar, click **Variables** and choose **Local Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.
3. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `generic_711_coupler_variables.txt`.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_in` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 19 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_mic` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 17 only.

Next, make selections of different parts of the geometry to ease the subsequent application of boundary conditions. Then proceed and set up the acoustic model and the boundary conditions.

ROOT

Explicit 1

On the **Definitions** toolbar, click **Explicit**.

DEFINITIONS

Explicit 1

1 Select Domains 1, 2, and 4 only.

2 In the **Settings** window for **Explicit**, type `Pressure acoustics` in the **Label** text field.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 Select Domains 3, 5, and 6 only.

3 In the **Settings** window for **Explicit**, type `Thermoviscous acoustics` in the **Label** text field.

Explicit 3

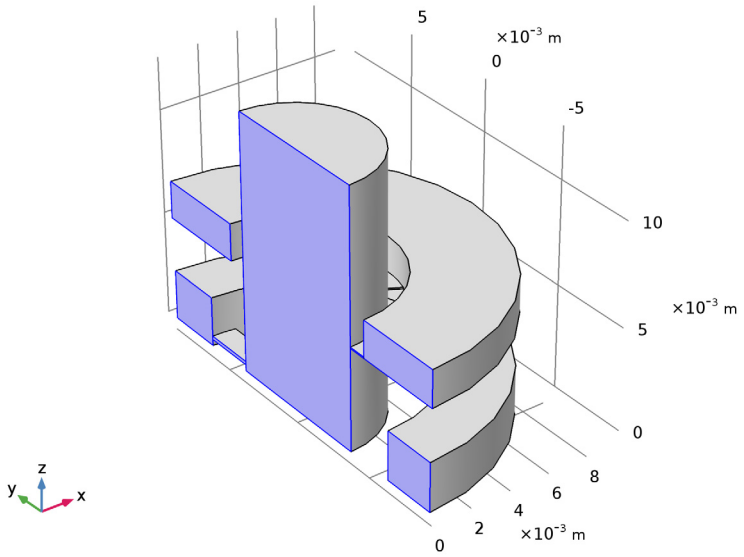
1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundaries 1, 5, 10, 15, 21, 25, and 27 only.

5 In the **Label** text field, type `Symmetry`.



ADD MATERIAL

1 On 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>Air**.

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

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	muB0	Pa·s	Basic

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 **Pressure acoustics**.

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 **Model Inputs** section.

3 In the T text field, type `Tref`.

Symmetry 1

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

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

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

Normal Displacement 1

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

2 Select Boundary 19 only.

3 In the **Settings** window for **Normal Displacement**, locate the **Normal Displacement** section.

4 In the d_n text field, type `d0`.

Impedance 1

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

2 Select Boundary 17 only.

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

4 From the **Impedance model** list, choose **RCL**.

5 In the R_{ac} text field, type `Rmic`.

6 In the C_{ac} text field, type `Cmic`.

7 In the L_{ac} text field, type `Lmic`.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Thermoviscous Acoustics Model 1

1 In the **Model Builder** window, expand the **Thermoviscous Acoustics, Frequency Domain (ta)** node, then click **Thermoviscous Acoustics Model 1**.

2 In the **Settings** window for **Thermoviscous Acoustics Model**, locate the **Model Input** section.

3 In the T_0 text field, type `Tref`.

4 In the **Model Builder** window, click **Thermoviscous Acoustics, Frequency Domain (ta)**.

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

6 From the **Selection** list, choose **Thermoviscous acoustics**.

Symmetry 1

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

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

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

Next, create and set up the multiphysics coupling between the pressure acoustics domain and the thermoviscous acoustics domain.

MULTIPHYSICS

Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)

1 On the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic-Thermoviscous Acoustic Boundary**.

2 In the **Settings** window for **Acoustic-Thermoviscous Acoustic Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Add another Pressure Acoustics, Frequency Domain physics interface. You will use this to illustrate the effect of thermoviscous acoustics in the slit channels by comparing to a solution of the same problem using regular lossless pressure acoustics here.

ADD PHYSICS

1 On the **Physics** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

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

5 On the **Physics** toolbar, click **Add Physics** to close the **Add Physics** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 2 (ACPR2)

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

2 From the **Selection** list, choose **Thermoviscous acoustics**.

Pressure Acoustics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Frequency Domain 2 (acpr2)** click **Pressure Acoustics 1**.

2 In the **Settings** window for **Pressure Acoustics**, locate the **Model Inputs** section.

3 In the T text field, type `Tref`.

Symmetry 1

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

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

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

Couple the two pressure acoustics interfaces together. The simplest approach is to give the dependent variable, the pressure, the same name p. Note that this approach is only valid when coupling identical physics!.

4 In the **Model Builder** window, click **Pressure Acoustics, Frequency Domain 2 (acpr2)**.

5 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, click to expand the **Dependent variables** section.

6 Locate the **Dependent Variables** section. In the **Pressure** text field, type `p`.

Now, build the mesh using a swept mapped mesh in the slits and a tetrahedral mesh in the remaining domains.

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundaries 14 and 35 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 18, 62, 73, and 80 only.

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

4 In the **Number of elements** text field, type 4.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 19 and 27 only.

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

4 In the **Number of elements** text field, type 25.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 63 and 66 only.

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

4 In the **Number of elements** text field, type 36.

Distribution 4

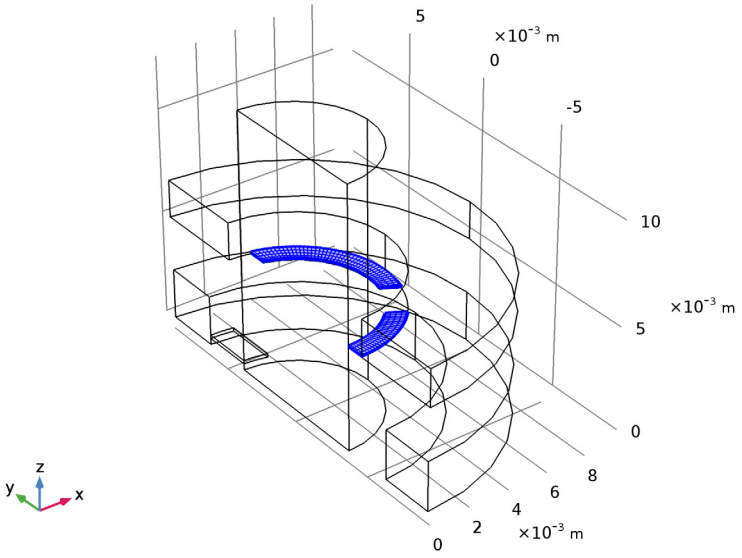
1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 79 and 86 only.

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

4 In the **Number of elements** text field, type 8.

5 Click **Build Selected**.



The mapped mesh on the upper side of the upper slits looks like that in the figure above (switch to wireframe rendering).

Now, proceed to mesh the upper side of the lower slit and then sweep the mesh in the slits.

Mapped 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 24 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 2** and choose **Distribution**.

2 Select Edges 36 and 69 only.

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

4 In the **Number of elements** text field, type 7.

Distribution 2

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 37 and 50 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `6`.

5 Click **Build Selected**.

Size

1 In the **Settings** window for **Size**, locate the **Element Size** section.

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `Lmax`.

4 In the **Minimum element size** text field, type `0.2[mm]`.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 3, 5, and 6 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type `3`.

4 Click **Build Selected**.

Convert 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Convert**.

2 In the **Settings** window for **Convert**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 11, 18, 22, 29, 33, 36, 39, and 41 only.

5 Click **Build Selected**.

Size 1

1 Right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 17 and 19 only.

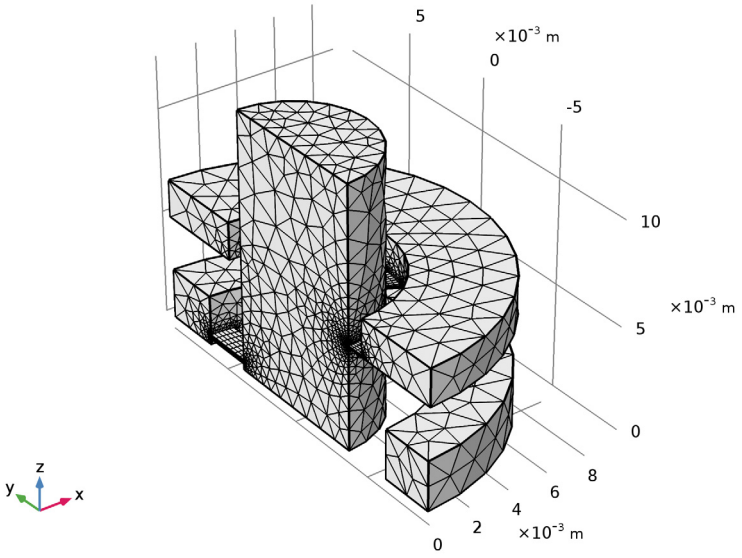
6 Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `1[mm]`.

9 Click **Build All**.

The mesh should look similar to that shown in the figure below.



ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

Turn off the generation of the default plots and create your own. Leave this option on to have the default plot of Pressure Acoustics and Thermoviscous Acoustics generated.

STUDY 1

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

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 `10^{range(2,2/19,4)} 10^{range(4.1,0.3/8, 4.4)}`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Pressure Acoustics, Frequency Domain 2 (acpr2)**.

6 Click **Disable**.

7 On the **Home** toolbar, click **Compute**.

Note that the computation may take several minutes.

Add a second study to solve the model neglecting the thermal and viscous losses in the slits. Disable thermoviscous acoustics and enable the second pressure acoustics model, in this way the system will be lossless (no thermoviscous losses in the slits). The results from the second study are stored in a separate data set. Compare this second solution to the model that includes the thermal and viscous losses.

ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Thermoacoustics, Frequency Domain (ta)** interface.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

3 In the **Model Builder** window, click **Step 1: Frequency Domain**.

4 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

5 In the **Frequencies** text field, type `10^{range(2,2/19,4)} 10^{range(4.1,0.3/8, 4.4)}`.

Make some small changes to the default solver to use a fully coupled approach when solving the two pressure acoustics physics together.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Stationary Solver 1** node.

4 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** and choose **Fully Coupled**.

Select the direct solver that uses MUMPS, or change the solver in the selected Direct 1 solver.

5 In the **Settings** window for **Fully Coupled**, locate the **General** section.

6 From the **Linear solver** list, choose **Direct**.

7 On the **Study** toolbar, click **Compute**.

First, create plots of the pressure distribution inside the coupler. Then move on to plotting the transfer impedance, input impedance, and the microphone response.

RESULTS

3D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Acoustic Pressure (Pa)`.

Volume 2

1 Right-click **Acoustic Pressure** and choose **Volume**.

2 In the **Model Builder** window, under **Results** right-click **Acoustic Pressure** and choose **Volume**.

3 In the **Settings** window for **Volume**, locate the **Expression** section.

4 In the **Expression** text field, type `ta.p_t`.

5 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.

6 On the **Acoustic Pressure** toolbar, click **Plot**.

The plot should look like Figure 6. To create Figure 7, and Figure 8 change the evaluation frequency to 13725 Hz and 886 Hz, respectively.

Plot the transfer impedance for the lossy and the lossless model, as well as the curves given by the standard Figure 3.

1D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Transfer Impedance` in the **Label** text field.

3 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

4 From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Transfer Impedance`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `f (Hz)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Z_trans (dB rel. 1MPa*s/m^3)`.

10 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 Right-click **Transfer Impedance** 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
`20*log10(abs(Ztrans/1e6[Pa*s/m^3]))`		`Model`

Global 2

1 In the **Model Builder** window, under **Results** right-click **Transfer Impedance** 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
int_trans(freq)		Standard (IEC 60318-4)

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

5 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (sol1)**.

Only plot the curves given by the IEC standard in the range from 100 to 10 kHz, where they are defined.

6 From the **Parameter selection (freq)** list, choose **From list**.

7 In the **Parameter values** list, choose values from 100 to 10000.

8 On the **Transfer Impedance** toolbar, click **Plot**.

9 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Global 3

1 Right-click **Results>Transfer Impedance>Global 2** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
int_trans_upper(freq)		Upper tolerance
int_trans_lower(freq)		Lower tolerance

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

5 Click to expand the **Legends** section. Clear the **Show legends** check box.

Global 4

1 In the **Model Builder** window, under **Results>Transfer Impedance** right-click **Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
20*log10(abs(Ztrans/1e6[Pa*s/m^3]))		Lossless model

5 On the **Transfer Impedance** toolbar, click **Plot**.

Transfer Impedance 1

1 In the **Model Builder** window, under **Results** right-click **Transfer Impedance** and choose **Duplicate**.

Proceed to reproduce the plot of the microphone response Figure 4.

2 In the **Settings** window for **1D Plot Group**, type `Microphone Response` in the **Label** text field.

3 In the **Settings** window for **1D Plot Group**, locate the **Title** section.

4 In the **Title** text area, type `Microphone Response`.

5 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Microphone Response (dB SPL rel. to 500 Hz)`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 In the **Model Builder** window, expand the **Results>Microphone Response** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`L0`		`Model`

Global 2

1 In the **Model Builder** window, under **Results>Microphone Response** click **Global 2**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`int_mic(freq)`		`Standard (IEC 60318-4)`

Global 3

1 In the **Model Builder** window, under **Results>Microphone Response** click **Global 3**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
int_mic_upper(freq)		Upper tolerance
int_mic_lower(freq)		Lower tolerance

Global 4

1 In the **Model Builder** window, under **Results>Microphone Response** click **Global 4**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
20*log10(abs(intop_mic(p)/intop_mic(1)))-Lref		Lossless model

4 On the **Microphone Response** toolbar, click **Plot**.

Finally, plot the input impedance for the lossy and the lossless model as seen in Figure 5.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Input Impedance` in the **Label** text field.

3 In the **Settings** window for **1D Plot Group**, locate the **Title** section.

4 From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Input Impedance`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `f (Hz)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Z_in (dB rel. 1MPa*s/m^3)`.

Global 1

1 Right-click **Input Impedance** 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
20*log10(abs(Zin/1e6[Pa*s/m^3]))		Model

Global 2

1 Right-click **Results>Input Impedance>Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

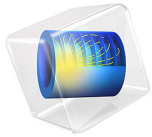
4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(Zin/1e6[Pa*s/m^3]))`		`Lossless Model`

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

6 On the **Input Impedance** toolbar, click **Plot**.

In the same way as you have created the plot of the pressure distribution (the first 3D plot), by plotting `acpr.p_t` and `ta.p_t` together, other acoustic quantities can be depicted. For example, to plot the sound pressure level plot `acpr.Lp` and `ta.Lp` in the same plot. More details, resolved by thermoviscous acoustics, can be plotted inside the slits by, for example, plotting the acoustic temperature variations `ta.T_t`, the rms velocity `ta.v_rms`, or the instantaneous velocity `ta.v_inst`.



Helmholtz Resonator Analyzed with Different Frequency Domain Solvers

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example simulates a simple three-dimensional axisymmetric Helmholtz resonator, a classic acoustics model of a resonating circuit with a known theoretical solution. The idealized version considered here consists of a tube and a closed volume in series, which are exposed to a harmonically oscillating pressure. Real-world phenomena explained by the resonator include, among others, the resonance from blowing across the top of an empty bottle and the sound produced by closed-cavity drums such as the djembe and by subwoofers. This model illustrates the use of different numerical solvers. The model shows how to solve this pressure acoustics problem for a range of frequencies using the following solvers: (a) Frequency Domain, with and without Asymptotic Waveform Evaluation (AWE) for faster solution, and (b) Frequency Domain-Modal, which reconstructs the frequency response based on a specified set of eigenmodes.

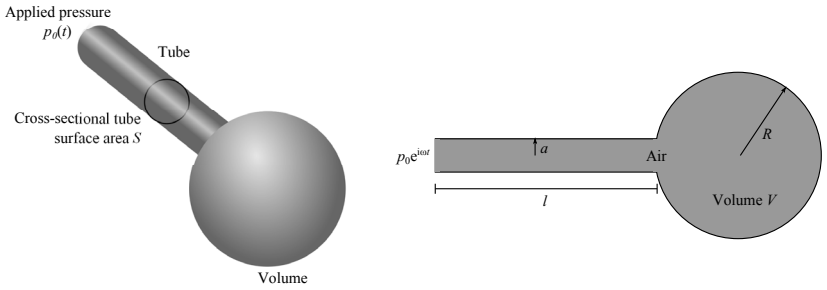


Figure 1: Illustrations of the Helmholtz resonator.

Model Definition

The model consists of a tube and volume coupled in series driven by a harmonically oscillating pressure $p_0(t) = p_0 e^{i\omega t}$ at the tube inlet, the Helmholtz resonator (schematically depicted in Figure 1) is one of the simplest resonating circuits. This circuit is typically described using lumped-parameter (equivalent circuit) modeling (see, for example, Ref. 1) as a serial coupling of an acoustic inertance L (equivalent to inductance in electric circuits and to mass in point-mass mechanics) caused by acceleration of the fluid in the tube and an acoustic compliance C (equivalent to capacitance in electric circuits and a spring in

point-mass mechanics) arising from compression of the volume; see Figure 1 and Figure 3.

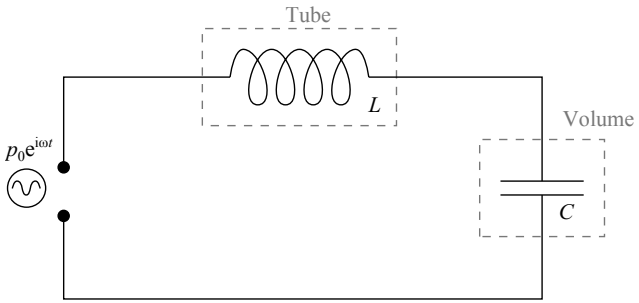


Figure 2: Diagram of the equivalent electric circuit to the Helmholtz resonator.

Because compression of the fluid volume and acceleration of the fluid in the tube are not instantaneous events but instead occur on specific time scales (each given by the geometry together with the properties of the fluid), the response of the resonator depends on the frequency, and there exists a frequency that maximizes the response — in other words, a resonance frequency ω_R. This is of course to be expected given the direct analogy to electric LC circuits, and by the same analogy we find that the resonance frequency is given by

$$\omega_R = \frac{1}{\sqrt{LC}} \tag{1}$$

As detailed in Ref. 1, the acoustic lumped-parameter elements are given by

$$\begin{aligned} L &= \frac{\rho_0(l+\gamma a)}{S} \\ C &= \frac{V}{\rho_0 c_0^2} \end{aligned} \tag{2}$$

where ρ_0 is the background quiescent density of the fluid, c_0 is the background quiescent speed of sound, l is the tube length, a is the tube radius, S is the cross-sectional area of the tube transverse to the direction of the flow, γ is the end correction factor (detailed below), and V is the closed volume. Thus, using Equation 2 in Equation 1 we find that the resonance frequency ω_R is given by

$$\omega_R = c_0\sqrt{\frac{S}{V(l+\gamma a)}} = c_0\sqrt{\frac{a^2}{\frac{4}{3}R^3(l+\gamma a)}} \tag{3}$$

where in the last equation we have assumed that the volume is a sphere with $V = \frac{4}{3}\pi R^3$ and the tube is cylindrical so $S = \pi a^2$.

END CORRECTION FACTOR γ: PHYSICAL ORIGIN AND APPROXIMATE NUMERICAL VALUE

When the fluid exits the tube and enters the volume, the acoustic waves disperse and the acoustic pressure drops. However, the waves initially continue along the axis of the tube when they just leave it, and moreover, they cannot move into the region occupied by the tube. Consequently, they do not completely disperse immediately as they leave the tube and the immediate region downstream of the tube is therefore still felt by the fluid in the tube where it imposes an acoustic load. In ideal models, this load results in an additional acoustic inertance corresponding to an effective increase in the length of the tube by ωa. In other words, the total length of the lumped-parameter inertance L in Equation 2 is longer than the actual tube length l.

The factor ω depends on the specific geometry of the tube-volume connection and is of the order unity, Ref. 1 and 2. We shall take for reference an infinite flange for which the correction factor is $\gamma = 0.82$ (Ref. 2); this is not fully correct because the acoustic pressure disperses less in the circular geometry resulting in a larger acoustic load.

SOLVER DESCRIPTIONS

Using the default settings for the Frequency Domain solver, it solves the problem subjected to harmonic excitation at a set of specified excitation frequencies. While this can be time-consuming for larger frequency sweeps, the (numerically) exact solution is calculated explicitly at every frequency, and so the solutions from this solver can always be expected to be correct (assuming convergence of the model and appropriate meshing to resolve all length scales of the physics).

To accommodate larger sweeps, this solver also contains the option to apply asymptotic waveform evaluation (AWE). This approach does not explicitly compute the exact solution at all frequencies but instead performs a Taylor expansion of the solution about a few exact solutions and otherwise uses a lower-order approximation (Padé or Taylor) to estimate the solution across the required frequency range.

Finally, the Frequency Domain Modal solver can also be used to perform a frequency sweep. For this, it first computes a set of system eigenfrequencies and associated eigenmodes (searching either within a user-defined range, or for a user-defined number of frequencies). The full solution across the frequency sweep is then approximated by a linear combination of the basis set formed by the eigensolutions (see Notes About the COMSOL Implementation).

GOALS OF ANALYSIS

This pressure acoustics problem is solved in a specified frequency regime using different solvers, with the dual purpose of illustrating the capabilities of the solvers and also highlight the solver-specific settings to be aware of. The following preset solvers are used: (a) Frequency Domain, with and without Asymptotic Waveform Evaluation (AWE) for faster solution, and (b) Frequency Domain-Modal, which reconstructs the frequency response based on the eigenfrequencies in the specified range.

RESISTANCE DUE TO ACOUSTIC RADIATION

As a final note, it should be mentioned that the equivalent circuit diagram in Figure 2 is incorrect. A full equivalent circuit description of the resonator should also include the acoustic radiation resistance R caused by the dissipated energy out of the tube when the fluid in the tube moves into the volume, see Figure 3. Mathematical formulas for the resistance in a number of situations may be found in Ref. 1. However, the value of the resistance does not affect the resonance frequency (the only read-out used here), only the absolute level of the impedance of the system, so we are well-justified in ignoring R.

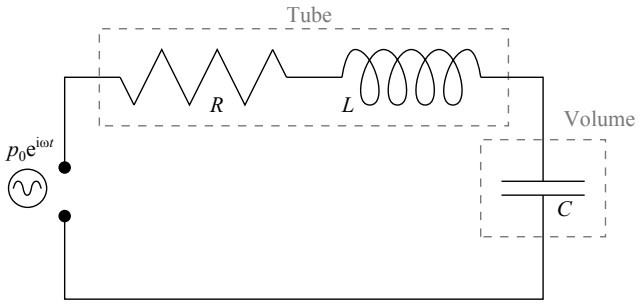


Figure 3: Diagram of the equivalent electric circuit to the Helmholtz resonator including the radiation resistance R.

Results and Discussion

This model uses the absolute acoustic pressure averaged over the end volume

$$\langle p \rangle = \frac{1}{V}\int_V \sqrt{p^2}\,\mathrm{d}\mathbf{r} \tag{4}$$

to investigate the response. We find that the lowest eigenfrequency indeed corresponds to the resonance predicted by the lumped-parameter model, and that this theoretical prediction indeed is in good agreement with the numerical results. However, the full

numerical COMSOL Multiphysics solutions illustrate the many higher modes that are ignored in the simple lumped-parameter model.

Comparing the different solvers, it is noted that they all produce the same response for the frequency sweep; see Figure 4 and Figure 5 on logarithmic and linear scales, respectively. The sound pressure level at 1043 Hz is depicted in Figure 6.However, no speed-up in CPU time is experienced with either the AWE option or the Modal solver for this particular problem. This is due to the relative simplicity of the geometry and especially because of the number of resonances included in the frequency sweep. See the next section for details.

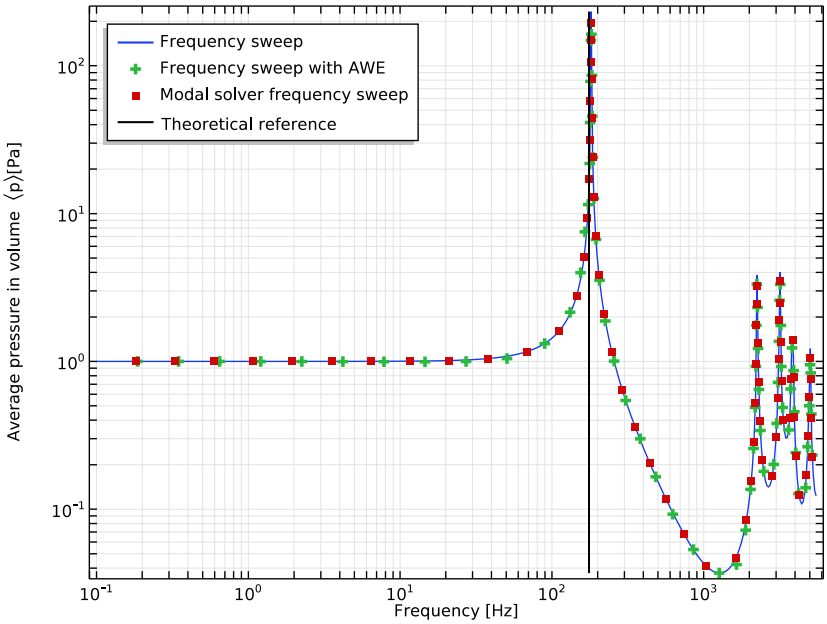


Figure 4: Frequency sweep for the readout $\langle p \rangle$ illustrating agreement between all solvers and the theory for the first eigenfrequency and also illustrating agreement between all solvers for the higher frequencies.

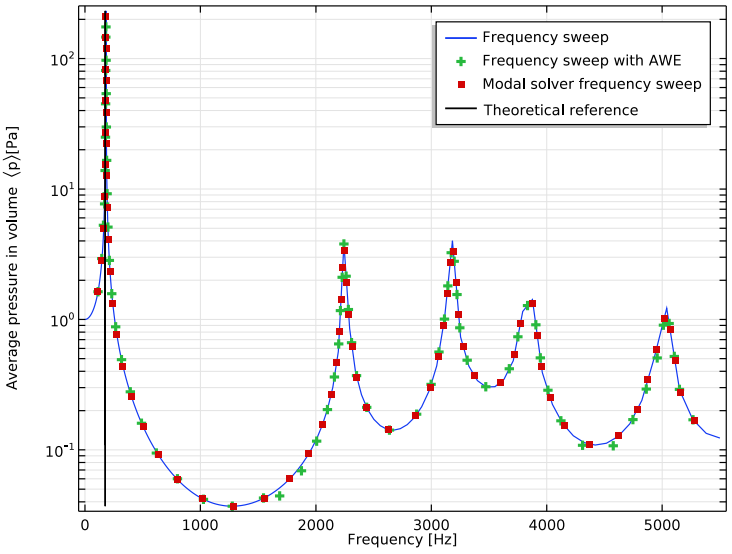


Figure 5: Same frequency sweep as in Figure 4 *here shown on a linear frequency scale to emphasize the higher frequencies.*

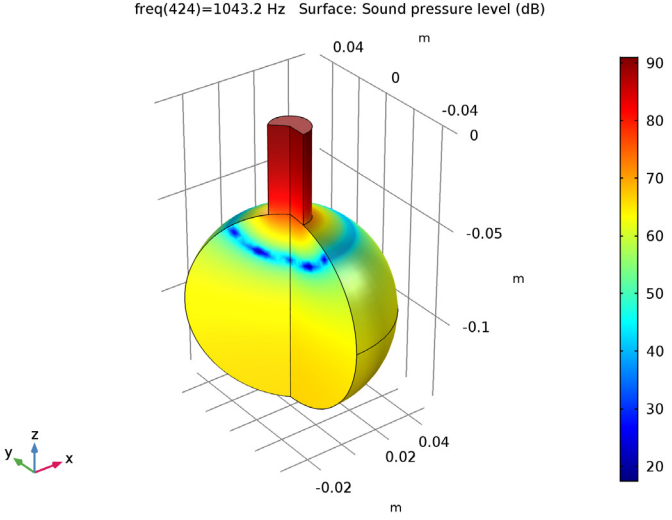


Figure 6: Sound pressure level computed as a default output from the simulations, here shown for f = 1043.2 Hz.

- As a general rule of thumb, at least five quadratic elements should be used to resolve each wavelength. Therefore, the maximum mesh element size is set to 1/5 of the shortest wavelengths present, `c_air/fmax`.
- The AWE option requires an expression to evaluate the solver performance (AWE expression). An integral of the squared pressure across the entire system is used, but other functions defined on either the whole system or parts of it could be used as well.

When several resonance frequencies are present, the default parameters for the AWE option must be changed to correctly capture the system response. This is done by lowering the relative tolerance as shown in Study 2. Other AWE options such as changing the number of points to linearize about (Evaluation points), changing the number of terms in the Taylor expansion about each point (Expansion size) and changing from Padé to Taylor expansion of the approximating solution (Expansion type) are less important for improving the results. In the current setup we chose a relatively high upper frequency bound so about 10 resonance frequencies are present in the sweep. Had the upper frequency limit been lowered so only the first resonance was included (for instance, by setting `fmax` = 100 Hz), then the default relative AWE tolerance would have sufficed. Note that if either end of the frequency range is close to a resonance, the AWE solver can become unreliable.

- The modal solver requires a different inlet pressure boundary condition `p0 = linper(1)` (Pressure 2 boundary condition) compared to `p0 = 1` used in the two other solvers (Pressure 1 boundary condition). This change in boundary condition is required because the modal solver linearizes the problem and would otherwise ignore the applied pressure at the boundary altogether, yielding a zero solution. The `linper` operator informs the solver that this pressure is a perturbation (a source term) that must be included in the linearized problem.
- To obtain good results with the Modal solver up to `fmax`, we must set the upper limit in its eigenfrequency search to `2*fmax` to satisfy the Nyquist sampling theorem. Using only `fmax` as the upper limit results in poorer estimates of the solution.
- The AWE option and the Frenquency Domain-Modal solver both rely on approximating the solution using a few exact solutions in the sweep range. Thus, these methods provide greater speed-up in CPU time if only few resonances fall in the sweep range, or if it is comparatively easier to find the eigensolutions relative to all the full solutions. Thus, these methods will be especially useful for large models and fine frequency sweeps over narrow ranges. As an example, solving for 10 times as many frequencies in the same range, the CPU time increase would be negligible with AWE,

quite small with Frequency Domain-Modal but a 10-fold increase in CPU time would be experienced with the regular solver.

- Finally, note that we define the model in terms of geometric parameters (a, L, and R). This makes it easy to quickly include parametric sweeps in the geometry, which, for instance, could be used to tune the lowest eigenfrequency.

References

1. D. T. Blackstock: "Fundamentals of Physical Acoustics", John Wiley & Sons, 2000

2. A. D. Pierce: Acoustics: "An introduction to its physical principles and applications", Acoustical Society of America, 1989

Application Library path: `Acoustics_Module/Tutorials/helmholtz_resonator_solvers`

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>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

Load all model parameters from a file; these include geometrical parameters and physical properties of the air.

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `helmholtz_resonator_solvers_parameters.txt`.

GEOMETRY 1

Construct the simple 2D axisymmetric geometry of the resonator from Figure 1 using rectangles and circles.

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

   You start by making the tube. It has length `L` and radius `a`.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `a`.
4. In the **Height** text field, type `L`.
5. Locate the **Position** section. In the **z** text field, type `-L`.

Circle 1 (c1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

   Proceed to make the volume which has radius `Rv` and is placed after the tube. You ensure a overlap of `a/2` knowing that you will merge the tube and volume into one object (the resonator). Note that objects or parts of objects extending into the left-hand side of the symmetry line (r=0) will be removed, so you will not have to manually remove the left half of the circle.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `Rv`.
4. Locate the **Position** section. In the **z** text field, type `-(L+Rv-a/2)`.

Union 1 (uni1)

1. On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

   Clean up the remaining parts of the geometry.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

Delete Entities 1 (del1)

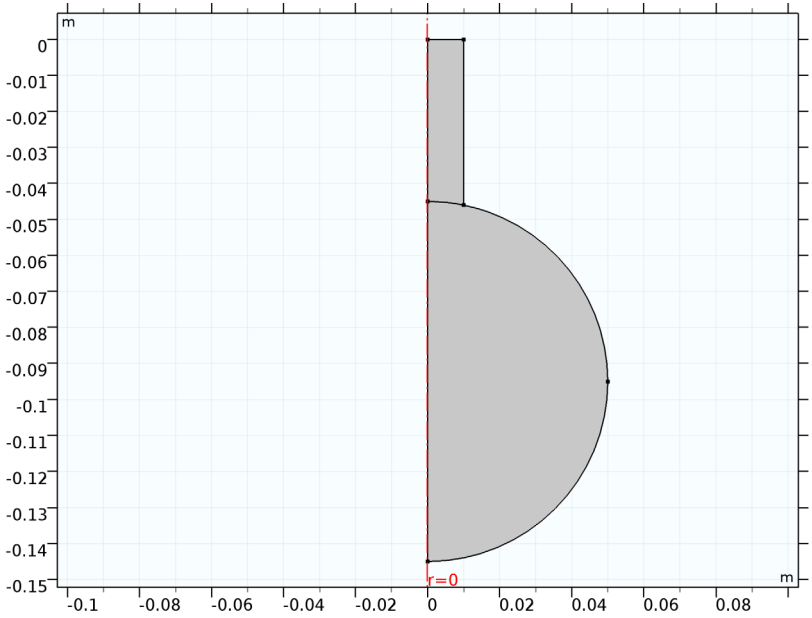
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **uni1**, select Boundaries 1, 2, and 5 only.

3 In the **Settings** window for **Delete Entities**, click **Build All Objects**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

5 On the **Geometry** toolbar, click **Build All**.



MATERIALS

Now add the material. For this simple model it suffices to use tabulated values for the physical parameters of air, which can be defined manually by adding a blank material and assign to it the values `rho_air` and `c_air`. Alternatively load air from the Material Library.

Material 1 (mat1)

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 `My air` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho_air	kg/m³	Basic
Speed of sound	c	c_air	m/s	Basic

Next define integrated variables and integration operations. First define an integration coupling that integrates over the volume `V` which will be used for evaluating the system response `intop1`. Also add another coupling that integrates over the entire system which will be used in the AWE solver option `intop2`. Then define the average absolute pressure in the volume which will be used as a readout of the system response.

Integration 1 (intop1)

On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

DEFINITIONS

Integration 1 (intop1)

1 Select Domain 1 only.

2 In the **Settings** window for **Integration**, locate the **Advanced** section.

3 Clear the **Compute integral in revolved geometry** check box.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 Click in the **Graphics** window and then press Ctrl+A to select both domains.

3 In the **Settings** window for **Integration**, locate the **Advanced** section.

4 Clear the **Compute integral in revolved geometry** check box.

Variables 1

1 Right-click **Integration 2 (intop2)** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `helmholtz_resonator_solvers_variables.txt`.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Pressure Acoustics 1

Next, you define that you will be solving Pressure Acoustics in the entire domain, and you apply the boundary conditions.

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

Pressure 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Pressure**, locate the **Pressure** section.

4 In the p_0 text field, type `1`.

Pressure 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure**.

Add a second pressure boundary condition to the same boundary as **Pressure 1**, and apply the `linper` operator. You will use this when running the Frequency Domain-Modal solver as described in Notes About the COMSOL Implementation. You will notice that the **Pressure 1** boundary condition is overridden when this additional Pressure 2 condition is added; you will toggle between the two when you set up the solvers.

2 Select Boundary 3 only.

3 In the **Settings** window for **Pressure**, locate the **Pressure** section.

4 In the p_0 text field, type `linper(1)`.

MESH 1

Next, define the mesh to have a maximum size corresponding to one fifth of the smallest wavelength in the system (see Notes About the COMSOL Implementation).

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

3 In the **Settings** window for **Size**, locate the **Element Size** section.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `hmax`.

STUDY 1

Next, set up the first of the three solvers (Frequency Domain solver). You already included this solver(this study) when you started this analysis, so you can immediately set it up.

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study 1 - Frequency sweep` in the **Label** text field.

STUDY 1 - FREQUENCY SWEEP

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - Frequency sweep** 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 `10^{range(log10(fmin),(log10(fmax)-(log10(fmin)))/499,log10(fmax))}`.

This command selects 500 frequencies in the range `fmin` - `fmax` which will be evenly spaced when shown on a logarithmic axis.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)>Pressure 2**.

6 Click **Disable**.

7 On the **Home** toolbar, click **Compute**.

ROOT

Now use the same solver as before, but this time with the Asymptotic Waveform Evaluation (AWE) option. You perform this as a separate study to be able to compare the two solutions, and therefore you first add a new Frequency Domain study.

ADD STUDY

1 On 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 **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On 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. In the **Frequencies** text field, type `10^{range(log10(fmin),(log10(fmax)-(log10(fmin)))/499,log10(fmax))}`.
3. Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
4. In the **Physics and variables selection** tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)>Pressure 2**.
5. Click **Disable**.
6. Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Use asymptotic waveform evaluation** check box.
7. In the **AWE expressions** text field, type `comp1.intop2(comp1.p^2)`.

   This function is used during the AWE algorithm to evaluate its performance. Other functions, including the value in a point somewhere in the system, could be used as well.
8. In the **Model Builder** window, click **Study 2**.
9. In the **Settings** window for **Study**, locate the **Study Settings** section.
10. Clear the **Generate default plots** check box.
11. In the **Label** text field, type `Study 2 - Frequency sweep with AWE`.
12. On the **Home** toolbar, click **Compute**.

ADD STUDY

1. On 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 **Preset Studies>Frequency Domain Modal**.
4. Click **Add Study** in the window toolbar.
5. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

You then set up the third solver; the Frequency Domain Modal study.

STUDY 3

Step 1: Eigenfrequency

1. In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 3 - Modal solver frequency sweep`.

STUDY 3 - MODAL SOLVER FREQUENCY SWEEP

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 3 - Modal solver frequency sweep** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 From the **Eigenfrequency search method** list, choose **Region**.

Next, change the upper limit of the real part of the frequencies you want to investigate to make sure the entire frequency range `fmin - fmax` is appropriately resolved.

4 Find the **Search region** subsection. In the **Largest real part** text field, type `2*fmax`.

Step 2: Frequency Domain Modal

1 In the **Model Builder** window, under **Study 3 - Modal solver frequency sweep** click **Step 2: Frequency Domain Modal**.

2 In the **Settings** window for **Frequency Domain Modal**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `10^{range(log10(fmin),(log10(fmax)-(log10(fmin)))/499,log10(fmax))}`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

1D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Next, you add a plot of the system response across all investigated frequencies.

2 In the **Settings** window for **1D Plot Group**, type `Frequency sweeps - logarithmic scale` in the **Label** text field.

Global 1

1 Right-click **Frequency sweeps - logarithmic scale** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1 - Frequency sweep/Solution 1 (sol1)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
p_avg	Pa	Average pressure in volume

5 In the **Label** text field, type `Frequency sweep`.

Global 2

1 In the **Model Builder** window, under **Results** right-click **Frequency sweeps - logarithmic scale** and choose **Global**.

2 In the **Settings** window for **Global**, type `Frequency sweep with AWE` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 - Frequency sweep with AWE/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
p_avg	Pa	Average pressure in volume

Global 3

1 Right-click **Frequency sweeps - logarithmic scale** and choose **Global**.

2 In the **Settings** window for **Global**, type `Modal solver frequency sweep` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
p_avg	Pa	Average pressure in volume

4 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Modal solver frequency sweep/Solution 3 (sol3)**.

Global 4

1 Right-click **Frequency sweeps - logarithmic scale** and choose **Global**.

2 In the **Settings** window for **Global**, type `Theoretical reference` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1 - Frequency sweep/Solution 1 (sol1)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
p_avg	Pa	Average pressure in volume

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `f_theo`.

7 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

8 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

9 On the **Frequency sweeps - logarithmic scale** toolbar, click **Plot**.

Frequency sweep

1 In the **Model Builder** window, under **Results>Frequency sweeps - logarithmic scale** click **Frequency sweep**.

2 In the **Settings** window for **Global**, click to expand the **Coloring and style** section.

Now change plot styles and update legend texts to produce presentation-worthy plots of the system response computed from all the solvers. These will look like Figure 3 and Figure 4.

Frequency sweep with AWE

1 In the **Model Builder** window, under **Results>Frequency sweeps - logarithmic scale** click **Frequency sweep with AWE**.

2 In the **Settings** window for **Global**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.

5 In the **Number** text field, type `70`.

Modal solver frequency sweep

1 In the **Model Builder** window, under **Results>Frequency sweeps - logarithmic scale** click **Modal solver frequency sweep**.

2 In the **Settings** window for **Global**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

5 In the **Number** text field, type `72`.

Frequency sweeps - logarithmic scale

1 In the **Model Builder** window, under **Results** click **Frequency sweeps - logarithmic scale**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Frequency [Hz]`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Average pressure in volume \langle p \rangle [Pa]`.

8 Click to expand the **Title** section. From the **Title type** list, choose **None**.

9 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Average pressure in volume  \langle p\rangle [Pa]`.

10 On the **Frequency sweeps - logarithmic scale** toolbar, click **Plot**.

Frequency sweep

1 In the **Model Builder** window, under **Results>Frequency sweeps - logarithmic scale** click **Frequency sweep**.

2 In the **Settings** window for **Global**, click to expand the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
Frequency sweep

Frequency sweep with AWE

1 In the **Model Builder** window, under **Results>Frequency sweeps - logarithmic scale** click **Frequency sweep with AWE**.

2 In the **Settings** window for **Global**, locate the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
Frequency sweep with AWE

Modal solver frequency sweep

1 In the **Model Builder** window, under **Results>Frequency sweeps - logarithmic scale** click **Modal solver frequency sweep**.

2 In the **Settings** window for **Global**, locate the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Modal solver frequency sweep`

Theoretical reference

1 In the **Model Builder** window, under **Results>Frequency sweeps - logarithmic scale** click **Theoretical reference**.

2 In the **Settings** window for **Global**, locate the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Theoretical reference`

5 In the **Settings** window for **Global**, locate the **Coloring and Style** section.

6 From the **Color** list, choose **Black**.

7 Click to expand the **Coloring and style** section.

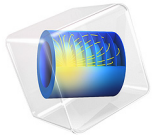
Frequency sweeps - logarithmic scale 1

1 In the **Model Builder** window, under **Results** right-click **Frequency sweeps - logarithmic scale** and choose **Duplicate**.

2 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

3 In the **Settings** window for **1D Plot Group**, type `Frequency sweeps - linear scale` in the **Label** text field.

4 Click to expand the **Legend** section. From the **Position** list, choose **Upper right**.



Helmholtz Resonator with Flow: Interaction of Flow and Acoustics

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Helmholtz resonators are used in exhaust systems as they can attenuate a specific narrow frequency band. The presence of a flow in the system alters the acoustic properties of the resonator and hence the transmission loss of the sub-system. In this model example a Helmholtz resonator is located as a side branch to a main duct. The transmission loss through the main duct is investigated when a mean flow is introduced.

The mean background flow in the system is calculated with the SST turbulence model for Mach numbers $\mathrm{Ma} = 0.05$ and $\mathrm{Ma} = 0.1$. The acoustic problem is then solved using the Linearized Navier-Stokes, Frequency Domain (LNS) physics interface. The mean flow velocity, pressure, and turbulent viscosity is coupled to the LNS model. The model uses a mapping study to interpolate the flow solution from the CFD mesh onto the acoustics mesh.

The geometry dimensions of the system are taken from those presented in Ref. 1. The results can be compacted to the measurements found in the journal paper.

Note: This model requires both the Acoustics Module and the CFD Module.

Model Definition

The geometry and dimensions are the same as the ones presented in Ref. 1. The geometry consists of a man cylindrical duct with a diameter of 4.859 cm and a Helmholtz resonator attached to the side. The resonator has a neck of length 8.05 cm and diameter 4.044 cm. The volume of the resonator is 4501 cm^3. The geometry is depicted in Figure 1. The symmetry of the system has been utilized.

The system is filled with air. A mean stationary background flow of average velocity $U_{\mathrm{in}} = c_0 \cdot \mathrm{Ma}$ enters at the left (downstream in the z-axis orientation), where c_0 is the speed of sound. Three flow configurations are studied with $\mathrm{Ma} = 0$ (no flow), $\mathrm{Ma} = 0.05$, and $\mathrm{Ma} = 0.1$.

A plane acoustic wave enters the system downstream. The wave is added as a **Background Acoustics Field** feature in a small domain backed with the perfectly matched layer (PML). In this way reflected waves are free to leave the computational domain both downstream and upstream. A PML is also added at the upstream outlet. The background plane wave variables are defined according to

$$p_b = 1\ \text{Pa} \cdot e^{-ik_0 z} \qquad k_0 = \frac{\omega}{c_0 + U_{in}}$$

$$\mathbf{u}_b = \left(0, 0, -\frac{1}{i\omega\rho_0}\frac{\partial p_b}{\partial z}\right)$$

$$T_b = \frac{\alpha_p T_0}{\rho_0 C_p} p_b$$

The plane wave expressions are imported into variables in the model. Since the acoustics are solved using the linearized Navier-Stokes equations, perturbations in pressure, velocity, and temperature need to be defined. These are also solved for.

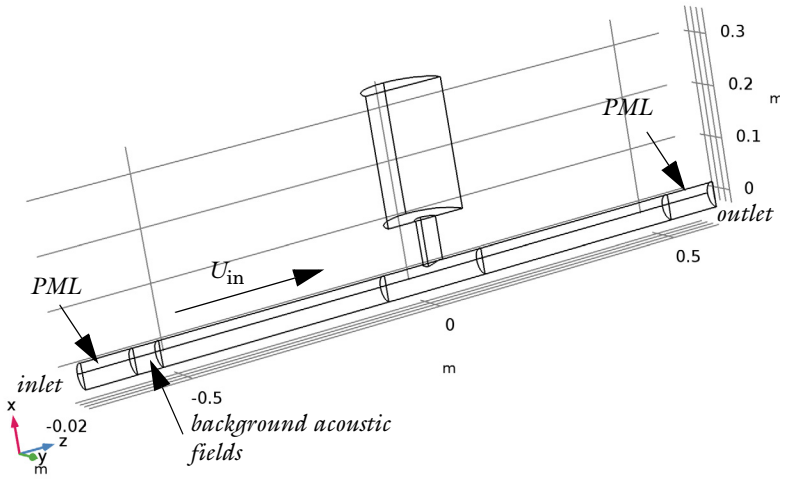


Figure 1: Geometry of the Helmholtz resonator model.

The transmission loss of the system is defined as

$$TL = 20\log_{10}\left(\left|\frac{1\ \text{Pa}}{p_{out}}\right|\right)$$

where p_{out} is the average pressure at the outlet and the incident field is defined with an amplitude of 1 Pa.

Two effects are important to capture correctly when analyzing this kind of system. It is the attenuation experience by the acoustic field as it interacts with turbulence (see Ref. 2 and Ref. 3) and the convective effects of the flow and the interaction with the flow gradients. The two phenomena have opposite effects on the resonance frequency of the system. The additional attenuation due to turbulence will shift the resonance towards a lower

frequency while the convective flow effects will shift it to a higher frequency. If the two are not correctly captured the results will be wrong. The transmission loss calculation is a good indicator that this is done correctly, this is seen in Figure 8 below. In order to capture the turbulence attenuation in detail, the SST turbulence model is used for the CFD as it introduces much less numerical diffusion and gives a better prediction of the eddy viscosity. The detailed interaction between flow and acoustics including the extra attenuation can only be modeled using the linearized Navier-Stokes equations as available in the Acoustics Module.

Results and Discussion

The magnitude of the mean background flow velocity is depicted in Figure 2 and the turbulent viscosity is depicted in Figure 3. The value of the turbulent viscosity should be compared to the dynamic viscosity of air $\mu = 2 \cdot 10^{-5}$ Pa·s. At the highest Mach number (depicted in the figure) the resulting experienced viscosity will be a factor 200 larger than the no-flow situation.

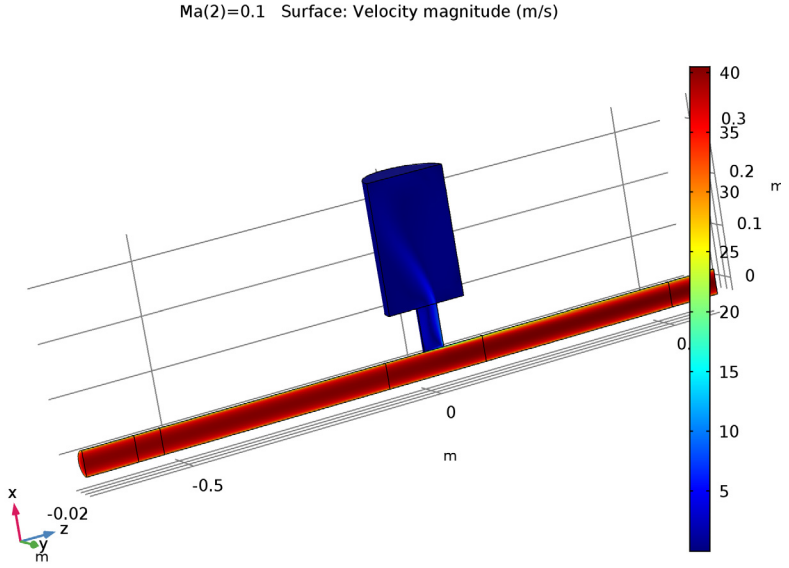


Figure 2: Background mens flow velocity magnitude for Ma = 0.1.

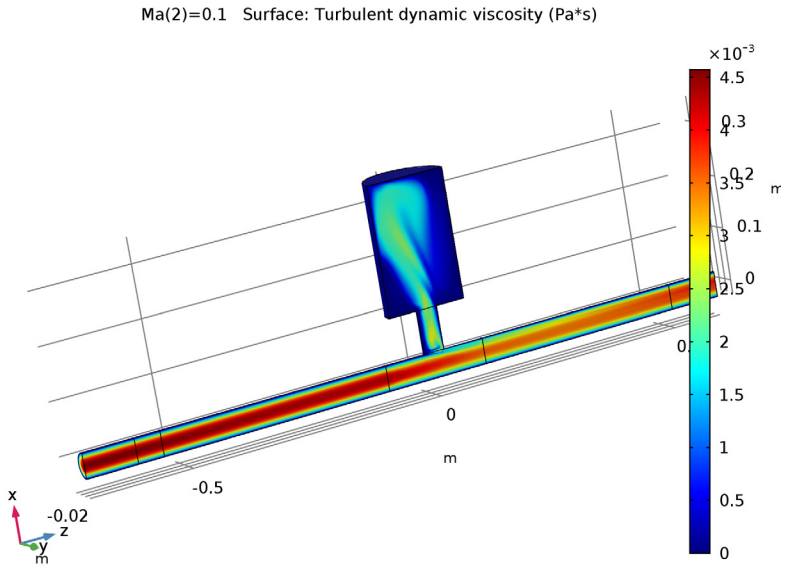


Figure 3: Turbulent viscosity for Ma = 0.1.

When a mapping study is used, it is good practice to visualize the mapped solution and compare it to the original solution on the CFD mesh. In Figure 4 the axial flow velocity and the turbulent viscosity evaluated on the CFD mesh is compared to the mapped values on the acoustics mesh.

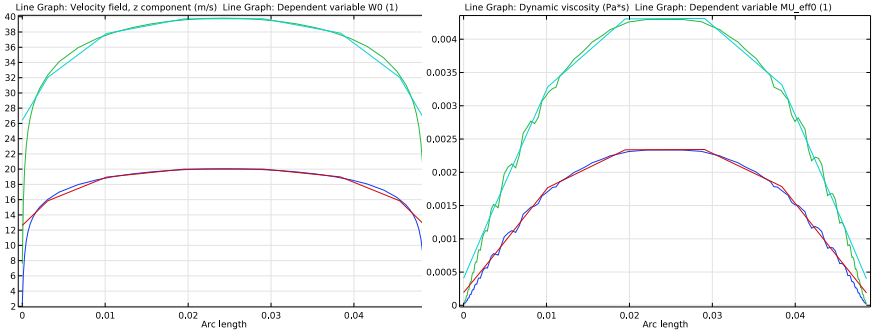


Figure 4: Mapped value of the axial velocity (left) and turbulent (viscosity) compared to the value as evaluated on the CFD mesh.

The total acoustic pressure in the system is depicrted at 200 Hz for the three cases, no flow in Figure 5, Ma = 0.05 in Figure 6, and Ma = 0.1 in Figure 7.

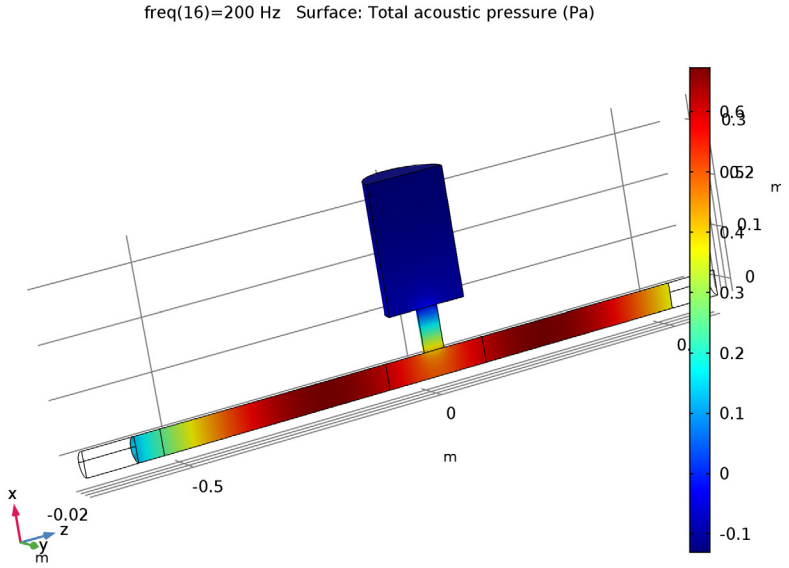


Figure 5: Acoustic pressure in the no.floe situation (Ma = 0).

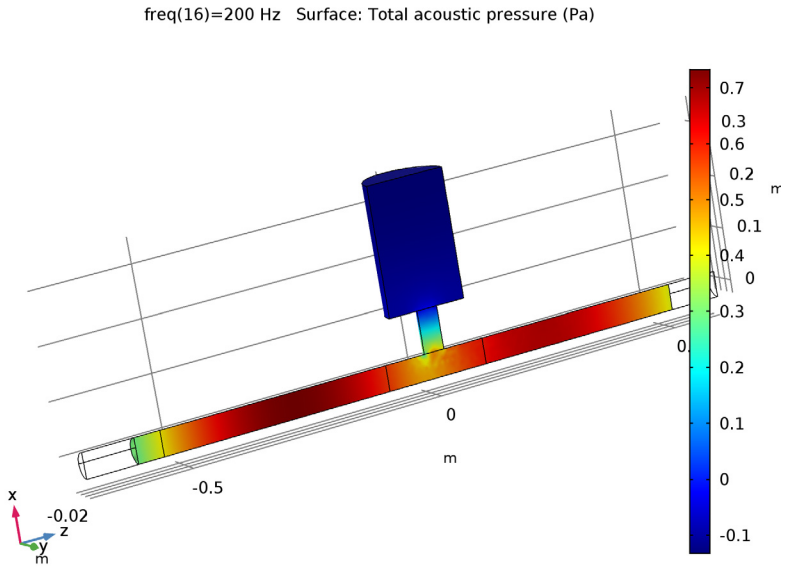


Figure 6: Acoustic pressure for Ma = 0.05.

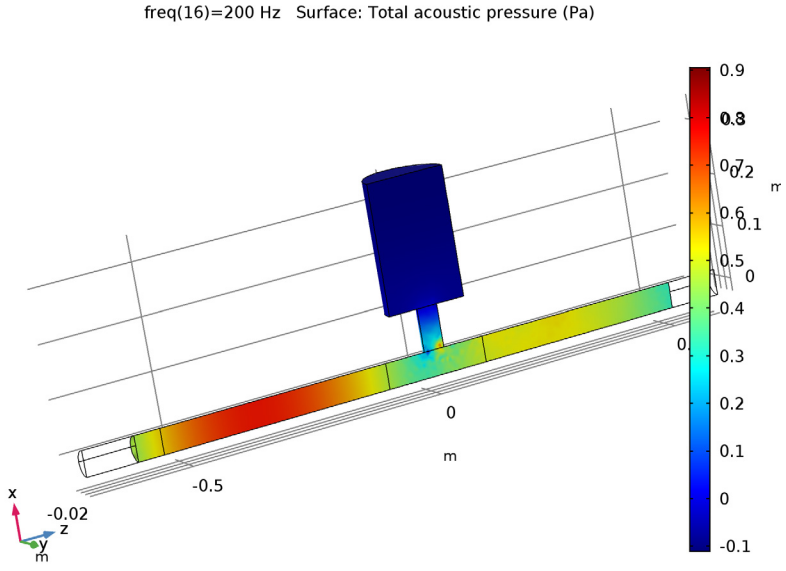


Figure 7: Acoustic pressure for Ma = 0.1.

Finally, the transmission loss of the system is depicted in Figure 8. The graph can be compared to measurements performed on the same system presented in Ref. 1. The results how good agreement.

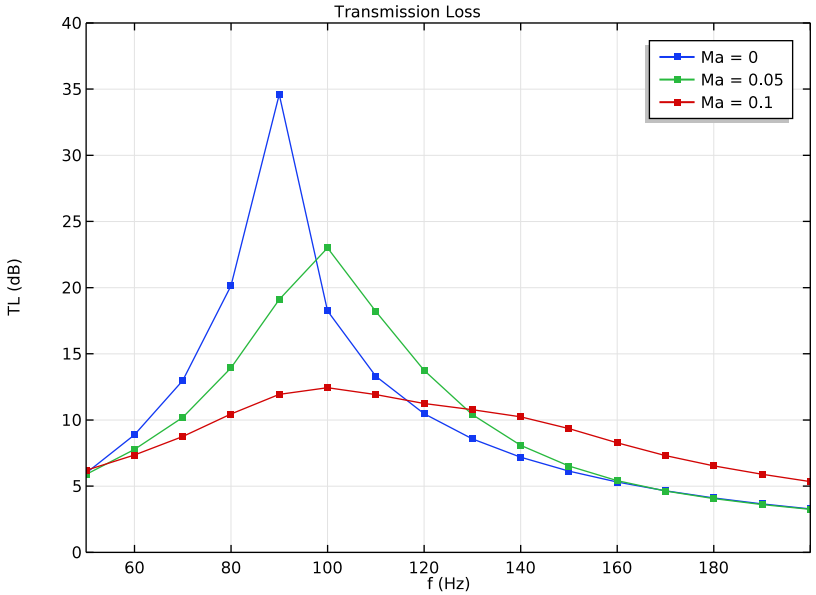


Figure 8: Transmission loss for the three flow configurations.

Notes About the COMSOL Implementation

MAPPING BETWEEN CFD AND ACOUSTICS

When the acoustics and the CFD models are not solved on the same computational mesh, careful mapping of the CFD solution from the CFD mesh onto the acoustics mesh should be done. This step is crucial in order not to introduce non-physical numerical noise into the acoustics solution. This can happen as the terms containing gradients of the background mean flow variables can become very noisy.

In this model two different mesh are used. In order to map the CFD solution a separate mapping study is used. The study solves an additional equation that maps and smooths the background flow variables. The mapping equations are set up using the **Weak Form PDE** interface from the **Mathematics** branch. In this model the mean background flow pressure p_0, velocity field $\mathbf{u}_0$, and turbulent viscosity μ_T variables are mapped onto corresponding

variables on the acoustics mesh $p_{0,\mathrm{aco}}$, $\mathbf{u}_{0,\mathrm{aco}}$, and $\mu_{\mathrm{T,aco}}$. The mapping and smoothing is achieved by solving

$$p_{0,\,\mathrm{aco}} - p_0 = \delta h^2 \nabla \cdot (\nabla p_{0,\,\mathrm{aco}})$$
$$u_{i,\,0,\,\mathrm{aco}} - u_{i,\,0} = \delta h^2 \nabla \cdot (\nabla u_{i,\,0,\,\mathrm{aco}})$$
$$\mu_{T,\,\mathrm{aco}} - \mu_T = \delta h^2 \nabla \cdot (\nabla \mu_{T,\,\mathrm{aco}})$$

where the term on the right hand side adds smoothing using isotropic diffusion. The amount of diffusion is controlled by the parameter δ (a constant that can be tuned) and the mesh size squared h^2. The term in some sense corresponds to source term stabilization as known form CFD.

In the implementation in the model the value of the CFD variable is fetched from the CFD mesh using the `withsol()` operator. This is an extrusion coupling operator that can refer directly to a solution object and parameter value. It is important to use such an operator such that the CFD solution is mapped and interpolated correctly to the integration (Gauss) points on the acoustics mesh.

In some cases it can be necessary to extend the mapping equation with boundary conditions for the mapped variables. If, for example, the flow details near walls is important then add a no-slip condition on $\mathbf{u}_{0,\mathrm{aco}}$. This is not necessary in this model. But we do add a symmetry condition on the mapped velocity field at the symmetry plane using a constraint $\mathbf{u}_{0,\mathrm{aco}} \cdot \mathbf{n} = 0$.

References

1. E. Selamet, A. Selamet, A. Iqbal, and H. Kim, "Effect of Flow in Helmholtz Resonator Acoustics: A Three-Dimensional Computational Study vs. Experiments," SAE International Journal, 2011.

2. L. Du, A. Holmberg, M. Karlsson, and M Åbom, "Sound amplification at a rectangular T-junction with merging mean flows," *J. Sound Vib.*, vol. 367, pp. 69–83 2016.

3. J. Gikadi, S. Föller, T. Sattelmayer, "Impact of turbulence on the prediction of linear aerodynamic interactions: Acoustic response of a turbulent shear layer," *J. Sound Vib.*, vol. 333, pp. 6548–6559, 2014.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/helmholtz_resonator_with_flow`

Modeling Instructions

The following model consists of three parts. First the background mean flow is solved using the CFD module, then the CFD solution is mapped to the acoustics mesh using a user-defined mapping study, and then the acoustics is solved with the mapped flow as input.

The first step is to set up the CFD model and solve the flow for two Mach numbers. Running this part of the model can take 8 h.

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 **Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, SST (spf)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Stationary with Initialization**.
6. Click **Done**.

GLOBAL DEFINITIONS

In the **Model Builder** window, collapse the **Global Definitions** node.

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `helmholtz_resonator_with_flow_parameters.txt`.

CFD Model

Start by setting up a 2D axisymmetric model that is used to solve a fully developed turbulent inlet profile that is used in the 3D model. This procedure is the same as in, for example, the Ultrasonic Flow Meter with Generic Time-of-Flight Configuration model found under the Ultrasound applications.

GEOMETRY 1

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `Dmain/2`.
4. In the **Height** text field, type `L2Daxi`.
5. Click **Build Selected**.

DEFINITIONS

General Extrusion 1 (genext1)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.
2. In the **Settings** window for **General Extrusion**, type `genext_2Daxi` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 3 only.
5. Locate the **Destination Map** section. In the **r-expression** text field, type `sqrt(x^2+y^2)`.
6. Clear the **z-expression** text field.
7. Locate the **Source** section. Select the **Use source map** check box.
8. Clear the **z?-expression** text field.

MATERIALS

On the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **Built-In>Air**.
3. Click **Add to Component 1**.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

TURBULENT FLOW, SST (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, SST (spf)**.

2 In the **Settings** window for **Turbulent Flow, SST**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

4 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Velocity field** text field, type `u2`.

5 In the **Velocity field components** table, enter the following settings:

u2
v2
w2

6 In the **Pressure** text field, type `p2`.

7 In the **Turbulent kinetic energy** text field, type `k2`.

8 In the **Specific dissipation rate** text field, type `om2`.

9 In the **Reciprocal wall distance** text field, type `G2`.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `Uin`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 3 only.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundary 3 only.

4 In the **Settings** window for **Distribution**, locate the **Distribution** section.

5 From the **Distribution properties** list, choose **Predefined distribution type**.

6 In the **Number of elements** text field, type 30.

7 In the **Element ratio** text field, type 15.

8 Select the **Reverse direction** check box.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 30.

6 In the **Element ratio** text field, type 15.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 200.

Now, set up the 3D model that solves for the flow inside the Helmholtz resonator system.

5 Click **Build All**.

ROOT

1 On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

The first step is to build the geometry of the Helmholtz resonator system. To save time you can **Insert Sequence** from file by referring to the `helmholtz_renoator_with_flow.mph` file in the applications folder of the COMSOL installation.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Dmain/2`.

4 In the **Height** text field, type `Lin+Lout+2*Lpml`.

5 Locate the **Position** section. In the **z** text field, type `-Lin-Lpml`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Lpml`

7 Clear the **Layers on side** check box.

8 Select the **Layers on bottom** check box.

9 Select the **Layers on top** check box.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Dneck/2`.

4 In the **Height** text field, type `1.2*Lneck`.

5 Locate the **Position** section. In the **x** text field, type `Dmain/2-0.2*Lneck`.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Cylinder 3 (cyl3)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Dreson/2`.

4 In the **Height** text field, type `Lreson`.

5 Locate the **Position** section. In the **x** text field, type `Dmain/2+Lneck`.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Cylinder 4 (cyl4)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Dmain/2`.

4 In the **Height** text field, type `Lsource`.

5 Locate the **Position** section. In the **z** text field, type `-Lin`.

Cylinder 5 (cyl5)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Dmain/2`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **z** text field, type `-0.1`.

6 Click **Build Selected**.

Partition Domains 1 (pard1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.

2 On the object **cyl2**, select Domain 1 only.

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 From the **Partition with** list, choose **Extended faces**.

5 On the object **cyl1**, select Boundaries 12 and 15 only.

6 From the **Repair tolerance** list, choose **Relative**.

7 Click **Build All Objects**.

8 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 2** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **pard1**, select Domain 1 only.

5 Click **Build Selected**.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Partition Objects 1 (par1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 Click **Build Selected**.

Delete Entities 2 (del2)

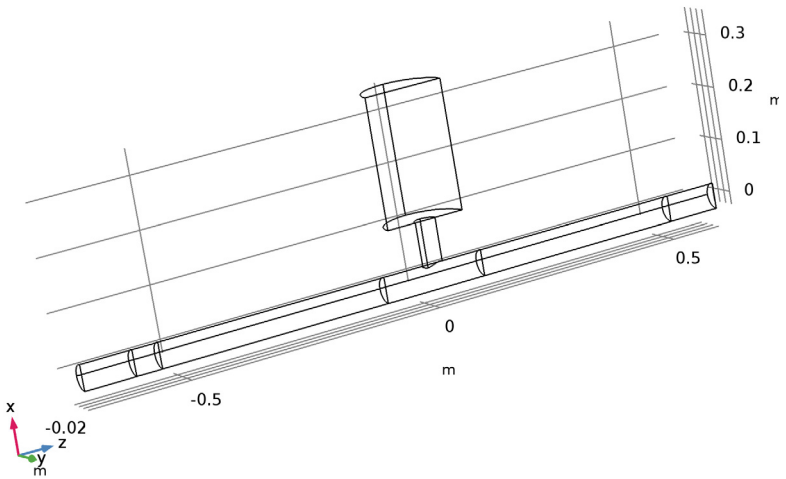
1 Right-click **Geometry 2** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **par1**, select Domains 2, 4, 6, 8, 10, 12, 14, and 16 only.

5 Click **Build All Objects**.



Rotate the gometry in the **Graphics** window. The geometry should look like the figure above.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Recently Used>Turbulent Flow, SST (spf)**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

TURBULENT FLOW, SST 2 (SPF2)

1 In the **Settings** window for **Turbulent Flow, SST**, locate the **Physical Model** section.

2 From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 2, 5, 8, 11, 14, 17, 29, and 34 only.

5 In the **Label** text field, type `Symmetry`.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1, 4, 7, 10, 13, 16, 20–25, 27, 28, 30–32, and 35 only.

5 In the **Label** text field, type `Walls`.

MATERIALS

On the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

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 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

TURBULENT FLOW, SST 2 (SPF2)

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 19 only.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

`comp1.genext_2Daxi(u2)*cos(atan2(y,x))`	`x`
`comp1.genext_2Daxi(u2)*sin(atan2(y,x))`	`y`
`comp1.genext_2Daxi(w2)`	`z`

6 Locate the **Turbulence Conditions** section. Click the **Specify turbulence variables** button.

7 In the k_0 text field, type `comp1.genext_2Daxi(k2)`.

8 In the ω_0 text field, type `comp1.genext_2Daxi(om2)`.

Generate the mesh used for the CFD simulation. In this model we set up a user-defined mesh consisting of hexahedral elements in the main part of the duct. Such a mesh will give the smallest amount of numerical diffusion. The turbulent viscosity will yield a good estimate of the eddy viscosity used in the acoustics model.

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, type `Mesh - CFD` in the **Label** text field.

3 Right-click **Component 2 (comp2)>Mesh - CFD** and choose **More Operations>Free Quad**.

MESH - CFD

Free Quad 1

Select Boundaries 3 and 19 only.

Size 1

1 Right-click **Component 2 (comp2)>Mesh - CFD>Free Quad 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 From the **Predefined** list, choose **Extremely fine**.

Size

1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh - CFD** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.005`.

6 In the **Minimum element size** text field, type `0.001`.

7 In the **Maximum element growth rate** text field, type `1.1`.

8 In the **Curvature factor** text field, type `0.4`.

9 In the **Resolution of narrow regions** text field, type `1`.

Swept 1

1 In the **Model Builder** window, right-click **Mesh - CFD** and choose **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1 and 2 only.

Distribution 1

1 Right-click **Component 2 (comp2)>Mesh - CFD>Swept 1** and choose **Distribution**.

2 Select Domain 1 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `8`.

Distribution 2

1 Right-click **Swept 1** and choose **Distribution**.

2 Select Domain 2 only.

Swept 2

1 In the **Model Builder** window, right-click **Mesh - CFD** and choose **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 3 only.

Distribution 1

1 Right-click **Component 2 (comp2)>Mesh - CFD>Swept 2** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 50.

5 In the **Element ratio** text field, type 4.

6 Select the **Reverse direction** check box.

Swept 3

1 In the **Model Builder** window, right-click **Mesh - CFD** and choose **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 6 only.

Distribution 1

1 Right-click **Component 2 (comp2)>Mesh - CFD>Swept 3** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type 8.

Swept 4

1 In the **Model Builder** window, right-click **Mesh - CFD** and choose **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 5 only.

Distribution 1

1 Right-click **Component 2 (comp2)>Mesh - CFD>Swept 4** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type 50.

5 In the **Element ratio** text field, type 4.

6 Select the **Reverse direction** check box.

Corner Refinement 1

1 In the **Model Builder** window, right-click **Mesh - CFD** and choose **Corner Refinement**.

2 In the **Settings** window for **Corner Refinement**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

Size 1

1 Right-click **Mesh - CFD** and choose **Free Tetrahedral**.

2 In the **Model Builder** window, under **Component 2 (comp2)>Mesh - CFD** right-click **Free Tetrahedral 1** and choose **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 26 only.

6 Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `0.0015`.

Boundary Layer Properties

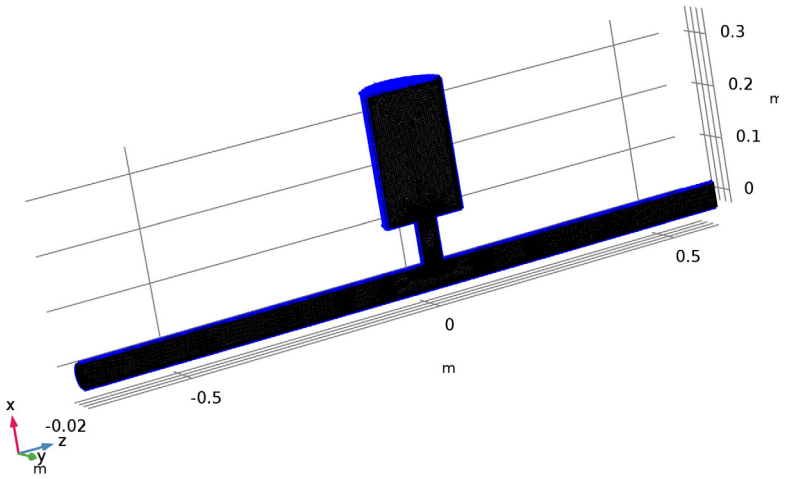
1 In the **Model Builder** window, right-click **Mesh - CFD** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Walls**.

4 Locate the **Boundary Layer Properties** section. In the **Thickness adjustment factor** text field, type `1.4`.

5 Click **Build All**.



The CFD mesh should look like the figure above.

6 Click **Build Selected**.

STUDY 1

1 In the **Settings** window for **Study**, type `Study 1 - CFD` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 1 - CFD

Step 2: Stationary

1 In the **Model Builder** window, under **Study 1 - CFD** click **Step 2: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Turbulent Flow, SST 2 (spf2)		physics

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - CFD, Wall Distance Initialization**.

Step 3: Stationary 2

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Turbulent Flow, SST (spf)		physics

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - CFD, Stationary**.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Ma	`0.05 0.1`	

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

After generating the solver sequence you can inspect the setup and make changes to the defaults if necessary.

Note that solving the model can take 5 h per value of the Mach number parameter (depending on your hardware).

2 Click **Compute**.

RESULTS

In the **Model Builder** window, expand the **Results** node.

Cut Line 3D 1

1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1 - CFD/Parametric Solutions 1 (8) (sol4)**.

4 Locate the **Line Data** section. In row **Point 1**, set **x** to `0`.

5 In row **Point 2**, set **x** to `0`.

6 In row **Point 1**, set **y** to `0`.

7 In row **Point 2**, set **y** to `0`.

8 In row **Point 1**, set **z** to `-1`.

9 In row **Point 2**, set **z** to `1`.

3D Plot Group 1

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `CFD: Velocity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1 - CFD/Parametric Solutions 1 (8) (sol4)**.

Surface 1

1 Right-click **CFD: Velocity** and choose **Surface**.

2 On the **CFD: Velocity** toolbar, click **Plot**.

The plot should reproduce Figure 2.

3D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `CFD: Turbulent Viscosity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1 - CFD/Parametric Solutions 1 (8) (sol4)**.

Surface 1

1 Right-click **CFD: Turbulent Viscosity** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `spf2.muT`.

4 On the **CFD: Turbulent Viscosity** toolbar, click **Plot**.

The plot should reproduce Figure 3.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `CFD: Turbulent Viscosity Axial` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1 - CFD/ Parametric Solutions 1 (7) (sol4)**.

Line Graph 1

1 Right-click **CFD: Turbulent Viscosity Axial** and choose **Line Graph**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `spf.muT`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `z-Lin-Lpml-100*Dmain`.

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **CFD: Turbulent Viscosity Axial** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

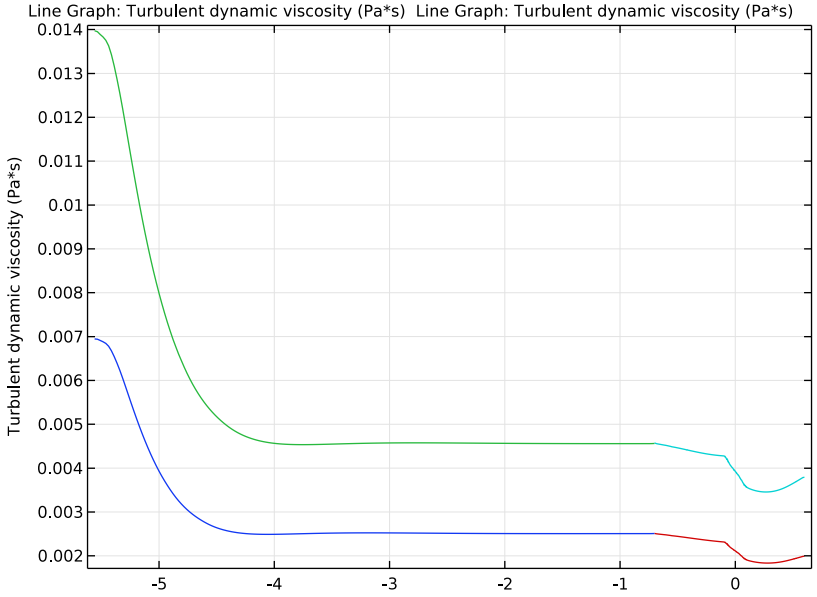
3 From the **Data set** list, choose **Cut Line 3D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `spf2.muT`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `z`.

7 On the **CFD: Turbulent Viscosity Axial** toolbar, click **Plot**.



The plot shows the turbulent viscosity along the central axis of the model including the 2D axi inlet section. Use the plot to inspect the convercence of the turbulent viscosity.

Mapping Model

An important part of solving aeroacoustic problems is to correctly map the CFD solution onto the acoustics domain. In the following set up a user defined equation that maps the solution from the CFD mesh onto the acoustics mesh. In this step a small amount of smoothing is used. The procedure is described in detail in the main part of the model documentation.

COMPONENT 2 (COMP2)

In the **Model Builder** window, click **Component 2 (comp2)**.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Mathematics>PDE Interfaces>Weak Form PDE (w)**.

4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1 - CFD	

5 Click to expand the **Dependent variables** section. Locate the **Dependent Variables** section. In the **Field name** text field, type `U0`.

6 In the **Number of dependent variables** text field, type `5`.

7 In the **Dependent variables** table, enter the following settings:

`P0`
`U0`
`V0`
`W0`
`MU_eff0`

8 Click **Add to Component** in the window toolbar.

9 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

WEAK FORM PDE (W)

1 In the **Settings** window for **Weak Form PDE**, type `Weak Form PDE: Mapping` in the **Label** text field.

2 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

3 Click to expand the **Discretization** section. From the **Element order** list, choose **Linear**.

WEAK FORM PDE: MAPPING (W)

Weak Form PDE 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Weak Form PDE: Mapping (w)** click **Weak Form PDE 1**.

2 In the **Settings** window for **Weak Form PDE**, locate the **Weak Expressions** section.

3 In the weak text-field array, type `(P0-withsol('sol4',spf2.pA,setval(Ma, Ma)))*test(P0)+delta*h^2*(P0x*test(P0x)+P0y*test(P0y)+P0z*test(P0z))` on the first row.

4 In the weak text-field array, type `(U0-withsol('sol4',u,setval(Ma,Ma)))* test(U0)+delta*h^2*(U0x*test(U0x)+U0y*test(U0y)+U0z*test(U0z))` on the second row.

5 In the weak text-field array, type `(V0-withsol('sol4',v,setval(Ma,Ma)))*test(V0)+delta*h^2*(V0x*test(V0x)+V0y*test(V0y)+V0z*test(V0z))` on the third row.

6 In the weak text-field array, type `(W0-withsol('sol4',w,setval(Ma,Ma)))*test(W0)+delta*h^2*(W0x*test(W0x)+W0y*test(W0y)+W0z*test(W0z))` on the fourth row.

7 In the weak text-field array, type `(MU_eff0-withsol('sol4',spf2.mu_eff,setval(Ma,Ma)))*test(MU_eff0)+delta*h^2*(MU_eff0x*test(MU_eff0x)+MU_eff0y*test(MU_eff0y)+MU_eff0z*test(MU_eff0z))` on the fifth row.

Make sure to enforce symmetry in the mapped velocity field by setting V0 = 0 on the symmetry plane. Do this by using a constraint.

Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Constraint**.

2 In the **Settings** window for **Constraint**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

4 Locate the **Constraint** section. In the R text-field array, type `V0` on the first row.

Create the acoustics mesh that the CFD solution is mapped onto. The mesh can be relatively coarse. At the walls we do not resolve the acoustic boundary layer since slip (and adiabatic) conditions are used here. A single boundary layer is added to resolve the background flow profile near the wall.

COMPONENT 2 (COMP2)

Mesh 3

On the **Mesh** toolbar, click **Add Mesh**.

MESH 3

1 In the **Settings** window for **Mesh**, type `Mesh - Acoustics` in the **Label** text field.

2 Right-click **Component 2 (comp2)>Meshes>Mesh - Acoustics** and choose **Free Tetrahedral**.

MESH - ACOUSTICS

Size

1 In the **Settings** window for **Size**, locate the **Element Size** section.

2 Click the **Custom** button.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `Dmain/5`.

4 In the **Minimum element size** text field, type `Dmain/15`.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Meshes>Mesh - Acoustics** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2–5, 7, and 8 only.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh - Acoustics** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `10`.

5 Click **Build Selected**.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Mesh - Acoustics** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.

3 Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 2 (comp2)>Meshes>Mesh - Acoustics> Boundary Layers 1** click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

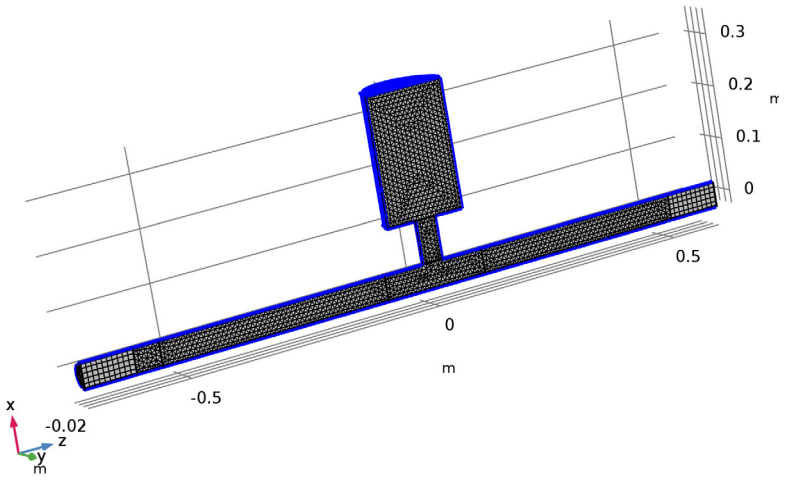
3 From the **Selection** list, choose **Walls**.

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type `1`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `Dmain/16`.

7 Click **Build All**.



The acoustics mesh should look like the figure above.

ROOT

On the **Home** toolbar, click **Windows** and choose **Add Study**.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Turbulent Flow, SST (spf)	
Turbulent Flow, SST 2 (spf2)	

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - Mapping` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Parametric Sweep

On the **Study** toolbar, click **Parametric Sweep**.

STUDY 2 - MAPPING

Parametric Sweep

1 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

2 Click **Add**.

3 Click to select row number 1 in the table.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Ma	`0.05 0.1`	

5 On the **Study** toolbar, click **Compute**.

Create two plots that compare the original CFD solution (evaluated on the CFD mesh) with the mapped solution (evaluated on the acoustics mesh).

RESULTS

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Mapping: Comparison w` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1 - CFD/ Parametric Solutions 1 (8) (sol4)**.

Line Graph 1

1 Right-click **Mapping: Comparison w** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Selection** section.

3 Select the **Active** toggle button.

4 Select Edge 12 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `w`.

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **Mapping: Comparison w** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - Mapping/Solution 7 (10) (sol7)**.

4 Select Edge 12 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `W0`.

6 On the **Mapping: Comparison w** toolbar, click **Plot**.

The plot should reproduce Figure 4 (left).

Mapping: Comparison w 1

1 Right-click **Mapping: Comparison w** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Mapping: Comparison mu_eff` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Mapping: Comparison w 1** node, then click **Results>Mapping: Comparison mu_eff>Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `spf2.mu_eff`.

Line Graph 2

1 In the **Model Builder** window, under **Results>Mapping: Comparison mu_eff** click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `MU_eff0`.

4 On the **Mapping: Comparison mu_eff** toolbar, click **Plot**.

The plot should reproduce Figure 4 (right).

Acoustics Model

Finally, set up the acoustics model solving the Linearized Navier-Stokes equations for the no-flow situation and for the background mean flow with Ma = 0.05 and Ma = 0.1.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Acoustics>Aeroacoustics>Linearized Navier-Stokes, Frequency Domain (lnsf)**.

4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1 - CFD	
Study 2 - Mapping	

5 Click **Add to Component** in the window toolbar.

6 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

DEFINITIONS

In the **Model Builder** window, expand the **Definitions** node.

Variables 1

1 Right-click **Component 2 (comp2)>Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Variables: Plane Wave` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `helmholtz_resonator_with_flow_variables.txt`.

These variables define a plane wave moving in the positive z-direction including the effect of a uniform flow. This wave is used as the source of the model.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 6 only.

5 In the **Operator name** text field, type `intop_in`.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 18 only.

5 In the **Operator name** text field, type `intop_out`.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 6 only.

LINEARIZED NAVIER-STOKES, FREQUENCY DOMAIN (LNSF)

Linearized Navier-Stokes Model 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Linearized Navier-Stokes, Frequency Domain (lnsf)** click **Linearized Navier-Stokes Model 1**.

2 In the **Settings** window for **Linearized Navier-Stokes Model**, locate the **Model Input** section.

3 In the p_0 text field, type `P0[Pa]`.

4 Specify the $\mathbf{u}_0$ vector as

`U0[m/s]`	x
`V0[m/s]`	y
`W0[m/s]`	z

5 Locate the **Fluid Properties** section. From the μ list, choose **User defined**. In the associated text field, type `MU_eff0`.

Use the **From speed of sound** option for both the thermal expansion and the compressibility.

6 Locate the **Thermal Expansion and Compressibility** section. From the α_p list, choose **From speed of sound**.

7 From the β_T list, choose **From speed of sound**.

MATERIALS

Air (mat2)

1 In the **Model Builder** window, expand the **Component 2 (comp2)>Materials** node, then click **Air (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	0	Pa·s	Basic

LINEARIZED NAVIER-STOKES, FREQUENCY DOMAIN (LNSF)

Wall 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Linearized Navier-Stokes, Frequency Domain (lnsf)** click **Wall 1**.

2 In the **Settings** window for **Wall**, locate the **Mechanical** section.

3 From the **Mechanical condition** list, choose **Slip**.

4 Locate the **Thermal** section. From the **Thermal condition** list, choose **Adiabatic**.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

Background Acoustic Fields 1

1 On the **Physics** toolbar, click **Domains** and choose **Background Acoustic Fields**.

2 Select Domain 2 only.

3 In the **Settings** window for **Background Acoustic Fields**, locate the **Background Acoustic Fields** section.

4 In the p_b text field, type `pb`.

5 Specify the $\mathbf{u}_b$ vector as

ub	x
vb	y
wb	z

6 In the T_b text field, type `Tb`.

Linearized Navier-Stokes Model 2

1 On the **Physics** toolbar, click **Domains** and choose **Linearized Navier-Stokes Model**.

2 In the **Settings** window for **Linearized Navier-Stokes Model**, type `Linearized Navier-Stokes Model: No Flow` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

4 Locate the **Thermal Expansion and Compressibility** section. From the α_p list, choose **From speed of sound**.

5 From the β_T list, choose **From speed of sound**.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Turbulent Flow, SST (spf)	
Turbulent Flow, SST 2 (spf2)	
Weak Form PDE: Mapping (w)	

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(50,10,200)`.

3 In the **Model Builder** window, click **Study 3**.

4 In the **Settings** window for **Study**, type `Study 3 - Acoustics, Ma = 0` in the **Label** text field.

5 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

6 On the **Home** toolbar, click **Compute**.

Remember to change the value of the Mach number parameter Ma to be 0.05 for the next simulation (it's inital value is 0 as used for the previous study).

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
Ma	0.05	0.05	Flow Mach number

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Turbulent Flow, SST (spf)	
Turbulent Flow, SST 2 (spf2)	
Weak Form PDE: Mapping (w)	

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(50,10,200)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 2 (comp2)>Linearized Navier-Stokes, Frequency Domain (lnsf)>Linearized Navier-Stokes Model: No Flow**.

5 Click **Disable**.

6 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 2 - Mapping, Stationary**.

9 From the **Parameter value (Ma)** list, choose **0.05**.

10 In the **Model Builder** window, click **Study 4**.

11 In the **Settings** window for **Study**, type `Study 4 - Acoustics, Ma = 0.05` in the **Label** text field.

12 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

13 On the **Home** toolbar, click **Compute**.

Remember to change the value of the Mach number parameter Ma to be 0.1 for the next simulation (it's value is 0.05 as used for the previous study).

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
Ma	0.1	0.1	Flow Mach number

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Turbulent Flow, SST (spf)	
Turbulent Flow, SST 2 (spf2)	
Weak Form PDE: Mapping (w)	

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 5

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(50,10,200)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 2 (comp2)>Linearized Navier-Stokes, Frequency Domain (lnsf)>Linearized Navier-Stokes Model: No Flow**.

5 Click **Disable**.

6 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 2 - Mapping, Stationary**.

9 From the **Parameter value (Ma)** list, choose **0.1**.

Under the **Mesh Selection** section you can check that the acoustics mesh is used.

10 Click to expand the **Mesh selection** section. In the **Model Builder** window, click **Study 5**.

11 In the **Settings** window for **Study**, type `Study 5 - Acoustics, Ma = 0.1` in the **Label** text field.

12 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

13 On the **Home** toolbar, click **Compute**.

RESULTS

3D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustics: Pressure Ma = 0` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 3 - Acoustics, Ma = 0/ Solution 8 (12) (sol8)**.

Surface 1

1 Right-click **Acoustics: Pressure Ma = 0** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `lnsf.p_t`.

Selection 1

1 Right-click **Results>Acoustics: Pressure Ma = 0>Surface 1** and choose **Selection**.

2 Select Boundaries 4, 5, 7–15, 21–24, and 26–35 only.

3 On the **Acoustics: Pressure Ma = 0** toolbar, click **Plot**.

The plot should reproduce Figure 5.

Acoustics: Pressure Ma = 0.1

1 In the **Model Builder** window, under **Results** right-click **Acoustics: Pressure Ma = 0** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustics: Pressure Ma = 0.05` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 4 - Acoustics, Ma = 0.05/ Solution 9 (14) (sol9)**.

4 On the **Acoustics: Pressure Ma = 0.05** toolbar, click **Plot**.

The plot should reproduce Figure 6.

Acoustics: Pressure Ma = 0.05.1

1 Right-click **Results>Acoustics: Pressure Ma = 0.05** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustics: Pressure Ma = 0.1` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 5 - Acoustics, Ma = 0.1/ Solution 10 (16) (sol10)**.

4 On the **Acoustics: Pressure Ma = 0.1** toolbar, click **Plot**.

The plot should reproduce Figure 7.

1D Plot Group 9

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Transmission Loss` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `f (Hz)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `TL (dB)`.

8 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

9 In the **Title** text area, type `Transmission Loss`.

10 Locate the **Axis** section. Select the **Manual axis limits** check box.

11 In the **y minimum** text field, type `0`.

12 In the **y maximum** text field, type 40.

13 In the **x minimum** text field, type 50.

14 In the **x maximum** text field, type 200.

Global 1

1 Right-click **Transmission Loss** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3 - Acoustics, Ma = 0/Solution 8 (12) (sol8)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(intop_in(pb)/ intop_out(lnsf.p_t)))`		`Ma = 0`

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

6 From the **Positioning** list, choose **In data points**.

Global 2

1 Right-click **Results>Transmission Loss>Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 4 - Acoustics, Ma = 0.05/Solution 9 (14) (sol9)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(intop_in(pb)/ intop_out(lnsf.p_t)))`		`Ma = 0.05`

Global 3

1 Right-click **Results>Transmission Loss>Global 2** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 5 - Acoustics, Ma = 0.1/Solution 10 (16) (sol10)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(intop_in(pb)/ intop_out(lnsf.p_t)))`		`Ma = 0.1`

5 On the **Transmission Loss** toolbar, click **Plot**.

The plot should reproduce Figure 8.

The last thing to do is to disable the PMLs in the two non-acoustic studies. This is to make sure that, if the two studies are solved again then the results will be correct. The CFD interface is not supported inside domains with PMLs.

STUDY 1 - CFD

Step 3: Stationary 2

1 In the **Model Builder** window, under **Study 1 - CFD** click **Step 3: Stationary 2**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 2 (comp2)>Definitions> Perfectly Matched Layer 1 (pml1)**.

5 Click **Disable**.

STUDY 2 - MAPPING

Step 1: Stationary

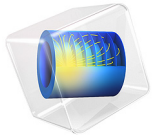
1 In the **Model Builder** window, expand the **Study 2 - Mapping** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 2 (comp2)>Definitions> Perfectly Matched Layer 1 (pml1)**.

5 Click **Disable**.



Hollow Cylinder

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Fluid acoustics coupled to structural objects, such as membranes or plates, represents an important application area in many engineering fields. Some examples are, loudspeakers, acoustic sensors, nondestructive impedance testing, and medical ultrasound diagnostics of the human body.

Model Definition

This model provides a general demonstration of an acoustic fluid phenomenon in 3D coupled to a solid object. In this study, the solid object is a capped, hollow aluminum cylinder filled with and immersed in water.

The acoustic waves created by a source inside the cylinder impact on the cylinder walls. In the model, you first calculate the frequency response from the solid object and then feed the information back to the acoustics domain so that you can analyze the wave pattern.

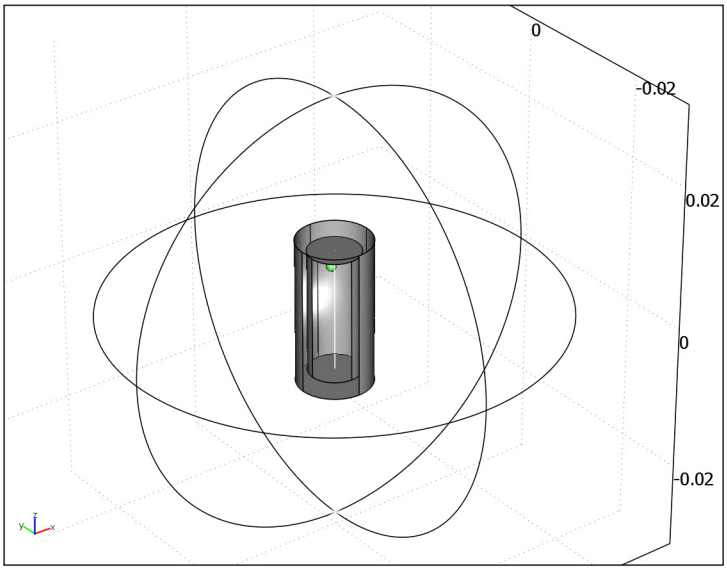


Figure 1: A hollow aluminum cylinder is immersed in water. The white line inside the cylinder indicates the line source, and the tiny sphere next to the line shows the position of the point source. The simulation domain is bounded by a large sphere.

Figure 1 illustrates the aluminum cylinder immersed in water. The cylinder is 2 cm in height and has an outer diameter of 1 cm. The thickness of its walls is 1.5 mm.

The water-filled acoustic domain outside the cylinder is truncated to a sphere with a reasonably large diameter. In two different versions of the model, the system is driven either by a line source coinciding with the axis of the cylinder and located entirely within the cylinder, or by a point source in the interior of the cylinder. The frequency is 60 kHz, that is, in the ultrasound region. The harmonic acoustic pressure in the water at the surface of the cylinder acts as a boundary load on the 3D solid to ensure continuity in pressure. In solving the model, the harmonic displacements and stresses in the solid cylinder are calculated, using the normal acceleration of the solid surface at the acoustics domain boundary to ensure continuity in acceleration.

DOMAIN EQUATIONS

Water Domain

For harmonic sound waves, in the frequency domain, we solve the Helmholtz equation defined in the Pressure Acoustics, Frequency Domain interface.

Solid Domain

You calculate the harmonic stresses and strains inside the solid cylinder walls using the same equation formulation as a frequency response analysis in the 3D Solid Mechanics interface. The material data comes from the built-in database for Aluminum 3003-H18.

BOUNDARY CONDITIONS

Outer Perimeter

On the outer spherical perimeter of the water domain (Figure 1), use the predefined *Radiation* condition with the *Spherical Wave* option. This boundary condition allows a spherical wave to travel out of the system, giving only slight reflections for the non-spherical components of the wave. The radiation boundary condition is useful when the surroundings are only a continuation of the domain.

For mathematical details on the radiation boundary condition, see the *Acoustics Module User's Guide*.

Cylinder-Water Interface

The model uses the Acoustic-Solid Interaction, Frequency Domain multiphysics interface, in which the Acoustic-Structure Boundary multiphysics coupling condition automatically apply to all acoustic-structure boundaries. The condition ensures that the acoustic pressure is added as a load on the solid and that the solid normal acceleration is added as a source on the acoustic domain.

EDGE AND POINT SETTINGS

In the two cases considered, the sound waves are generated by either a point monopole source or a line source. A line source along the z-axis defines the following right-hand-side source term to the Helmholtz equation:

$$\frac{4\pi}{\rho}\frac{1}{L_{\text{edge}}}\sqrt{\frac{P_{\text{rms}}c\rho}{2\pi}}\,\delta^{(2)}(\mathbf{r})$$

Here P_{rms} is the free space reference rms power per unit length of a line source of length L_{edge} placed in a homogeneous medium extending to infinity. Furthermore, $\delta^{(2)}(\mathbf{r})$ is the Dirac delta function in two dimensions, $\mathbf{r}$ denoting the projection of the position vector onto the xy-plane.

A monopole point source of rms power P_{rms} located at the point $\mathbf{R} = \mathbf{R}_0$ in an infinite homogeneous space defined the following source term

$$\frac{4\pi}{\rho}\sqrt{\frac{P_{\text{rms}}c\rho}{2\pi}}\,\delta^{(3)}(\mathbf{R}-\mathbf{R}_0)$$

where $\delta^{(3)}(\mathbf{R})$ is the Dirac delta function in three dimensions. Any type of confinement results in higher power usage.

Results

Figure 2 shows the resulting deformations and local sound pressure level from a computation with a line source.

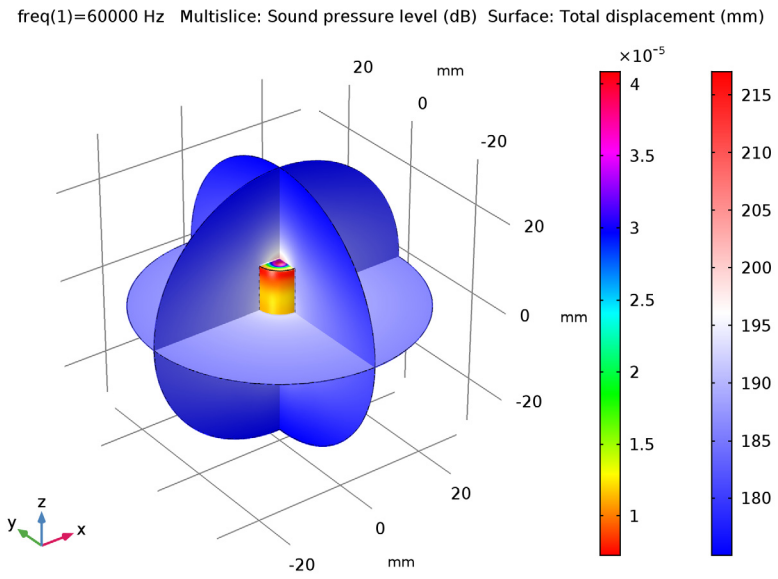


Figure 2: Sound-pressure level plot (dB) of the acoustic waves in the coupled problem, using a line source inside the cylinder. The surface of the cylinder shows its deformation (mm).

In Figure 3, you see the solution from an off-centered point. Note that the deformation color scale is different, implying that this source gives deformations that at least locally are greater than those from the line source.

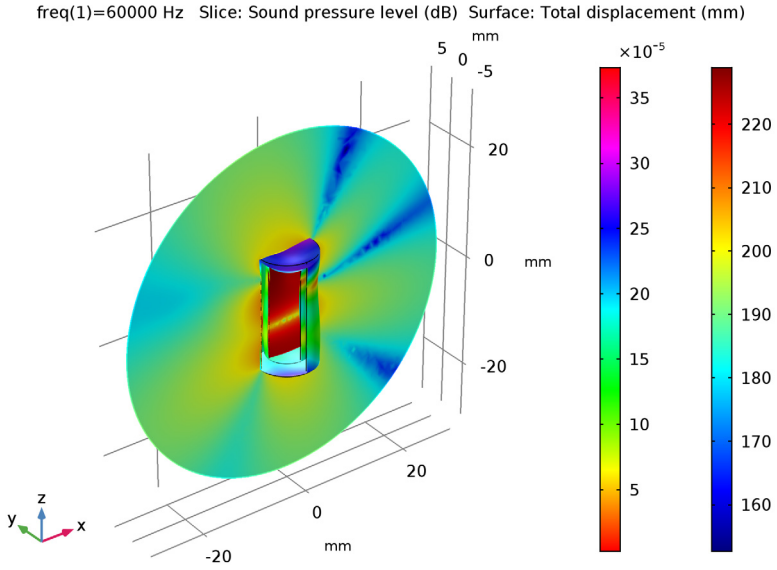


Figure 3: Sound-pressure level (dB) and structural deformations (mm) with a point source inside the cylinder. Some of the surfaces are hidden to reveal the pressure distribution inside the cylinder.

Notes About the COMSOL Implementation

The implementation of this model does not require any special tricks, but relies on standard equations and conditions in COMSOL Multiphysics and the Acoustics Module. Because of an internal scaling of the equations, the system of equations is symmetric. This means that you can use a solver designed for problems that generate symmetric stiffness matrices, thereby saving a considerable amount of system memory and shortening the calculation time.

Application Library path: `Acoustics_Module/Vibrations_and_FSI/hollow_cylinder`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Solid Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
edgeL	1.7[cm]	0.017 m	Length of line source

It is possible to fully parameterize the geometry, but in this model the length of the line source is the only parameter you define. The reason is that it appears both in the geometry and in the physics settings.

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**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `5`.

4 In the **Height** text field, type `20`.

5 Locate the **Position** section. In the **z** text field, type `-10`.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `3.5`.

4 In the **Height** text field, type `17`.

5 Locate the **Position** section. In the **z** text field, type `-8.5`.

6 Click **Build All Objects**.

7 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

With wireframe rendering, you now see the contours of both cylinders.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **z** to `-edgeL/2`.

5 In row **2**, set **z** to `edgeL/2`.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Point**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `1`.

4 In the **y** text field, type `2`.

5 In the **z** text field, type `5`.

Sphere 1 (sph1)

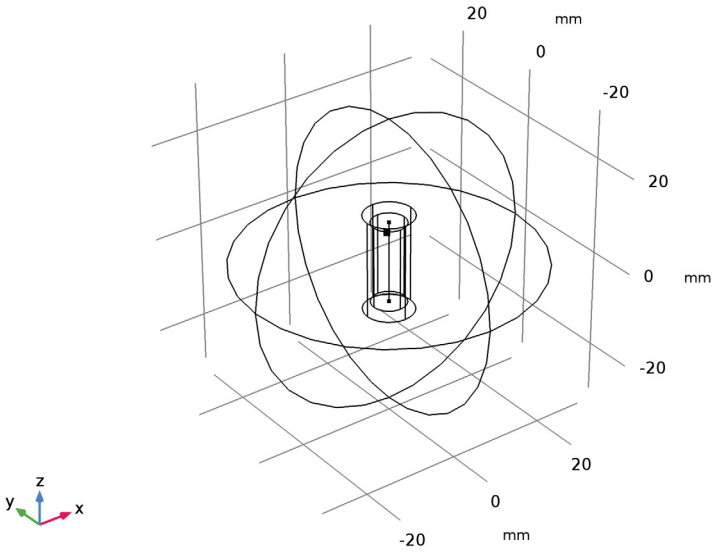
1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `30`.

4 Click **Build All Objects**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.



DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Solid` in the **Label** text field.

3 Select Domain 2 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Water` in the **Label** text field.

3 Select Domains 1 and 3 only.

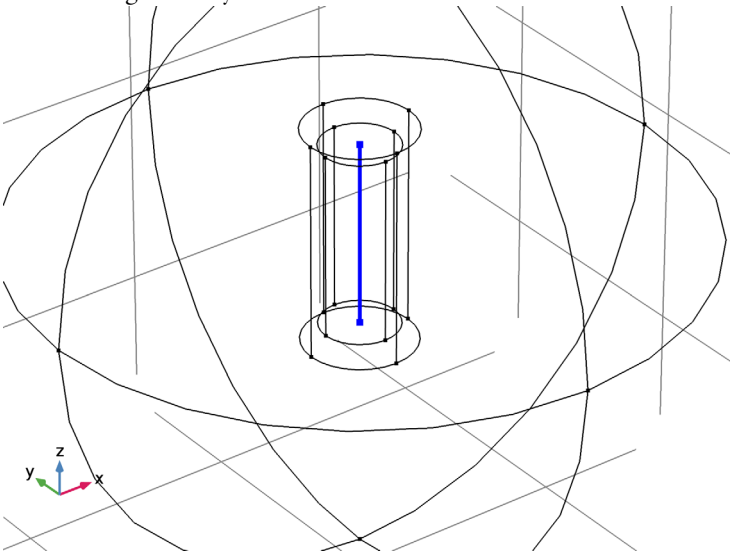
Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Line Source` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 26 only.



Explicit 4

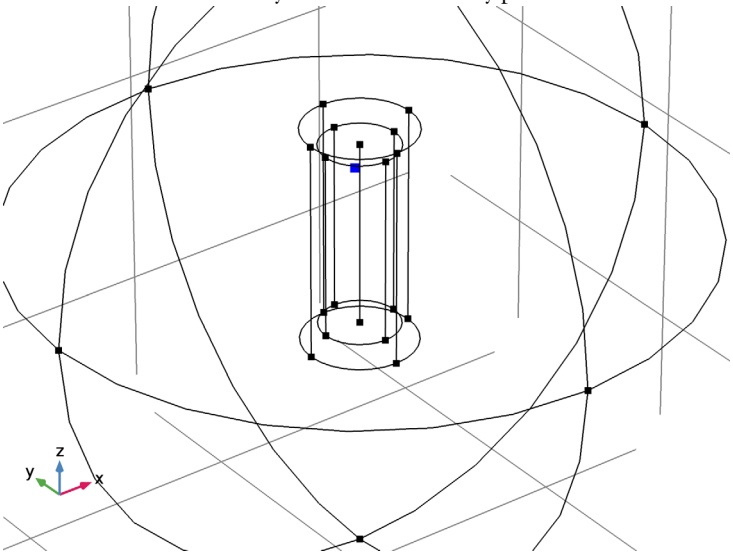
1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Point Source` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Point**.

4 Select Point 20 only.

Point 20 lies inside the cylinder and is the only point that is not connected to any edges.



ADD MATERIAL

1 On 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>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Water, liquid (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Water**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Aluminum 3003-H18**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum 3003-H18 (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Solid**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

By default, all domains are using the Pressure Acoustics model, which holds for fluids. Use the Water selection to specify where Pressure Acoustics is applied.

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 **Water**.

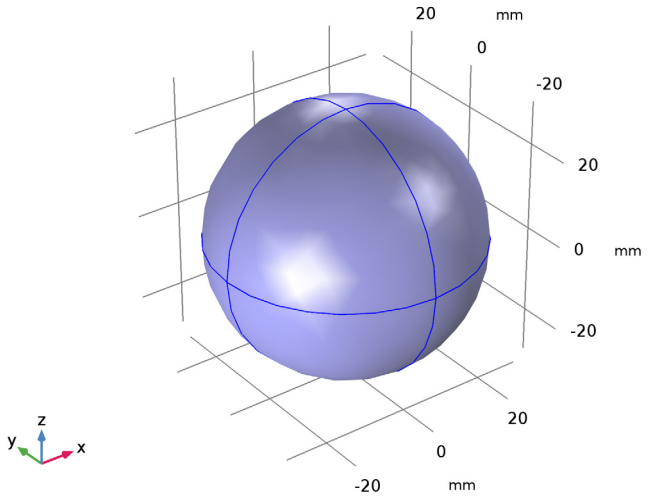
4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

The water reference pressure is 1 µPa. This does not affect the physics, but only serves as a reference in the definition of the sound pressure level.

Spherical Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Spherical Wave Radiation**.

2 Select Boundaries 1–4, 13, 14, 17, and 18 only.



Line Source 1

1 On the **Physics** toolbar, click **Edges** and choose **Line Source**.

2 In the **Settings** window for **Line Source**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Line Source**.

4 Locate the **Line Source** section. From the **Type** list, choose **Power**.

5 In the P_{rms} text field, type `0.5`.

Monopole Point Source 1

1 On the **Physics** toolbar, click **Points** and choose **Monopole Point Source**.

2 In the **Settings** window for **Monopole Point Source**, locate the **Point Selection** section.

3 From the **Selection** list, choose **Point Source**.

4 Locate the **Monopole Point Source** section. From the **Type** list, choose **Power**.

5 In the P_{rms} text field, type `0.5`.

You will select either the line source or the point source under the study by enabling **Modify physics tree and variables for study step**.

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 **Solid**.

MULTIPHYSICS

Now is a good time to inspect the **Multiphysics** node. Notice that the Acoustic-Structure Boundary feature is automatically added to all boundaries, but only applied to the boundaries that have water on one side and the solid on the other.

MESH 1

In the **Model Builder** window, expand the **Multiphysics** node.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Free Tetrahedral**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 From the **Selection** list, choose **Solid**.

6 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

9 In the associated text field, type `1.6`.

Size 2

1 Right-click **Free Tetrahedral 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 **Edge**.

4 From the **Selection** list, choose **Line Source**.

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 `2.5`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `5`.

This value corresponds to $\lambda/5$, where λ is the wavelength of the sound waves in the water. Combined with the default choice of second-order elements, it follows that the rule-of-thumb minimum of 10-12 degrees of freedom per wavelength for the solution to be reliable is satisfied. The structure and edge source use a finer mesh.

5 Click **Build All**.

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 **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `60[kHz]`.

Now, disable the point source by modifying the physics tree.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)>Monopole Point Source 1**.

6 Click **Disable**.

7 On the **Home** toolbar, click **Compute**.

Four default plots are created after running this model. You can inspect them and modify them.

Create a new plot of the deformation and sound pressure level. Then proceed to solving the system again with a monopole point source.

RESULTS

3D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement/SPL` in the **Label** text field.

Multislice 1

1 On the **Displacement/SPL** toolbar, click **More Plots** and choose **Multislice**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.Lp`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

Surface 1

1 In the **Model Builder** window, under **Results** right-click **Displacement/SPL** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.disp`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Cyclic**.

Deformation 1

1 Right-click **Results>Displacement/SPL>Surface 1** and choose **Deformation**.

2 On the **Displacement/SPL** toolbar, click **Plot**.

Your plot should now look like Figure 2. The remaining instructions show you how to set up a second study and replace the line source with a point source, while disabling the line source.

ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On 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 In the **Frequencies** text field, type `60[kHz]`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Pressure Acoustics, Frequency Domain (acpr)>Line Source 1**.

5 Click **Disable**.

6 In the **Model Builder** window, click **Study 2**.

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

9 On the **Home** toolbar, click **Compute**.

RESULTS

Study 2/Solution 2 (sol2)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 2/Solution 2 (sol2)**.

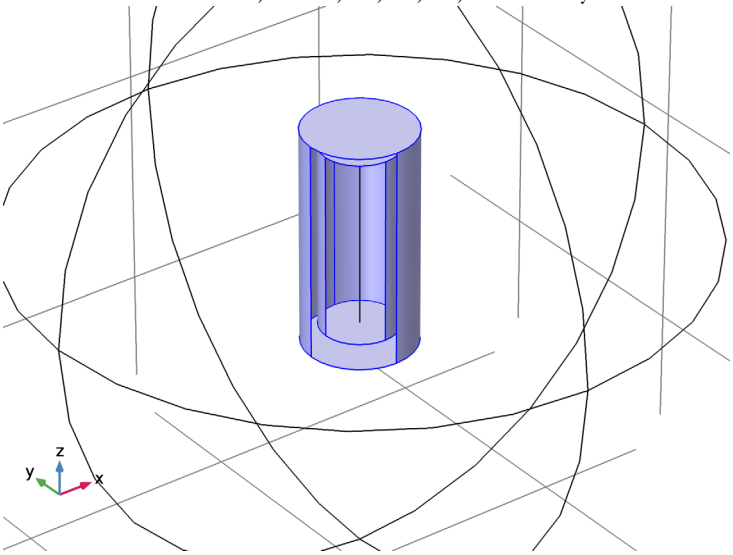
Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 6–8, 10–12, 15, 16, 19, and 20 only.



3D Plot Group 6

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement/SPL for Study 2` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

Slice 1

1 Right-click **Displacement/SPL for Study 2** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **ZX-planes**.

4 In the **Planes** text field, type `1`.

5 Locate the **Expression** section. In the **Expression** text field, type `acpr.Lp`.

Surface 1

1 In the **Model Builder** window, under **Results** right-click **Displacement/SPL for Study 2** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.disp`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Cyclic**.

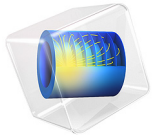
Deformation 1

1 Right-click **Results>Displacement/SPL for Study 2>Surface 1** and choose **Deformation**.

2 On the **Displacement/SPL for Study 2** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

You can see the resulting plot in Figure 3.



Optimizing the Shape of a Horn

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example shows how to apply boundary shape optimization to a simple axisymmetric horn. For the sake of simplicity, the far-field sound pressure level is maximized for a single frequency and in a single direction. The focus is on the optimization procedure, which involves parametrization of the geometry, choice of objective function and optimization solver settings.

The model was inspired by the work of Erik Bängtsson, Daniel Noreland, and Martin Berggren (Ref. 1).

Note: This application requires the Optimization Module.

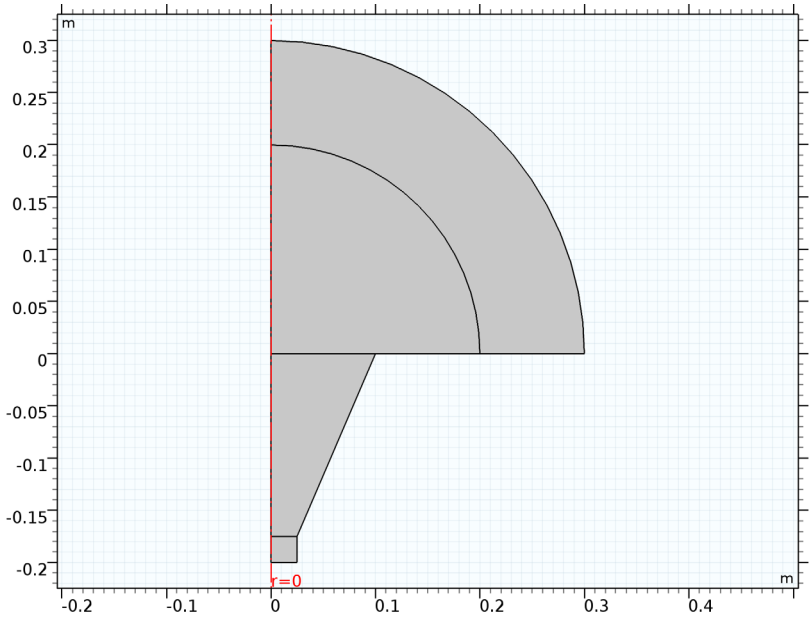


Figure 1: The initial configuration is a simple cone (the z<0 part of the gray area) in an infinite baffle.

Model Definition

A plane-wave mode feeds an axisymmetric horn radiating from an infinite baffle towards an open half space, see Figure 1. The radius of the feeding waveguide is assumed to be fixed, as well as the depth of the horn and the size of the hole where the horn is attached

to the baffle. By varying the curvature of the initially conical surface of the horn, its directivity and impedance can be changed.

The surface is parameterized assuming that the radius of the horn (as a function of the distance from the baffle) deviates from the simple cone by a function of the form

$$d_r = \sum_{i=1}^{N} q_i d_i \sin(i\pi s) \tag{1}$$

where s is a parameter varying between 0 and 1 along the edge of the cone, d_i are scale factors and the q_i are the *optimization variables* to be optimized. Note that $d_r(0) = d_r(1) = 0$, and that the function is smooth. The number of optimization variables can be varied. Using more variables gives more freedom and potentially a better final value of the objective function, but it also makes the optimization process more sensitive and can generate a shape that is less suitable for production.

Optimization can only be applied to real-valued functions, because the minimum of a complex-valued function is not well-defined. But the raw result from a frequency-domain acoustics simulation is a complex-valued pressure field. From this you generate a scalar, real-valued quantity to be used as the *objective function* in the optimization process. However, any operation which converts a complex number to a real value is necessarily non-analytical, which means that its derivative is not uniquely defined.

The gradient-based optimization solver in the Optimization Module by default evaluates derivatives of the objective function via the solution of an *adjoint equation*. This procedure requires that the symbolic derivative of any non-analytic function is selected in a special way. The default behavior of the composite functions `abs`(z) and `conj`(z), which are most commonly used to obtain a real-valued objective function, is to return a derivative parallel to the real axis. However, this behavior is not appropriate for the adjoint method, where you instead need the definitions

$$\begin{aligned} \frac{d}{dz}|z| &= \frac{\bar{z}}{|z|} \\ \frac{d}{dz_1}(z_1\bar{z}_2) &= \bar{z}_2 \\ \frac{d}{dz_2}(z_1\bar{z}_2) &= \bar{z}_1 \end{aligned} \tag{2}$$

It is indeed possible to redefine the symbolic derivatives of built-in functions in COMSOL Multiphysics, but in this case it is more convenient to use the special function `realdot`(z_1,

z_2), which evaluates as $\mathrm{real}(z_1 \cdot \mathrm{conj}(z_2))$ but differentiates according to Equation 2. In particular, as a measure of the transmission properties of the horn, use an expression of the form $\mathrm{realdot}(p_\mathrm{m}, p_\mathrm{m})/{p_0}^2$, where p_m is the pressure measured at a specific point in front of the horn and p_0 is the (real-valued and constant) amplitude of the incoming wave.

If you choose to evaluate p_m in the near-field, or can afford to include a sufficiently large domain in front of the horn to effectively measure a far-field value at a point in the model, you can simply measure p_m as the local pressure in a geometry vertex. However, in order to optimize the far-field directivity pattern in an efficient way, p_m should be defined using an integral representation of the far-field pressure as a function of the angle from the axis.

COMSOL Multiphysics contains optimized code for evaluating such far-field integrals. This is, however, a pure postprocessing feature that does not support the automatic differentiation required by the adjoint method. Therefore, return to the definition of the Helmholtz-Kirchhoff integral as given in its asymptotic axisymmetric form by Equation 2-17 in the *Acoustics Module User's Guide*:

$$p_{\mathrm{far}}(\mathbf{R}) \cong -\frac{1}{2}\int_S re^{ik\frac{zZ}{|\mathbf{R}|}}\left[J_0\left(\frac{krR}{|\mathbf{R}|}\right)\nabla p(\mathbf{r})\cdot\mathbf{n} - \frac{ikp(\mathbf{r})}{|\mathbf{R}|}\left(in_r RJ_1\left(\frac{krR}{|\mathbf{R}|}\right) + n_z ZJ_0\left(\frac{krR}{|\mathbf{R}|}\right)\right)\right]dS \tag{3}$$

If the infinite baffle is placed at $z = 0$, its effect is the same as that of adding a mirror image of the horn and at the same time removing the baffle. If, in addition, the integration surface is taken to be the wide end of the horn, in the plane of the baffle, most of the terms in Equation 3 cancel out, and all that is left is

$$p_\mathrm{m}(\Theta) = \int_S rJ_0(kr\sin(\theta))\frac{\partial p}{\partial z}dr \tag{4}$$

where J_0 is the Bessel function of the first kind of order 0, and the angle Θ from the axis has been introduced as a parameter. This integral is easily implemented in COMSOL Multiphysics using an integration operator.

Optimization as a rule implies many evaluations of the model for different designs, which can be very time consuming. In addition, the solver can be asked to evaluate each design at a number of frequencies and optimize with respect to the sum of the objective function values for each frequency. In this tutorial, a single frequency of 5000 Hz has been selected in order to make it possible to experiment with other aspects of the model. For example,

changing the parameter Θ, you can easily study the effect on the horn shape of optimizing the output at a specified angle from the axis.

Results and Discussion

By changing the shape of the horn within the limits of the selected parametrization, the on-axis sound pressure level can be raised by about 1 dB compared to the simple cone in Figure 1.

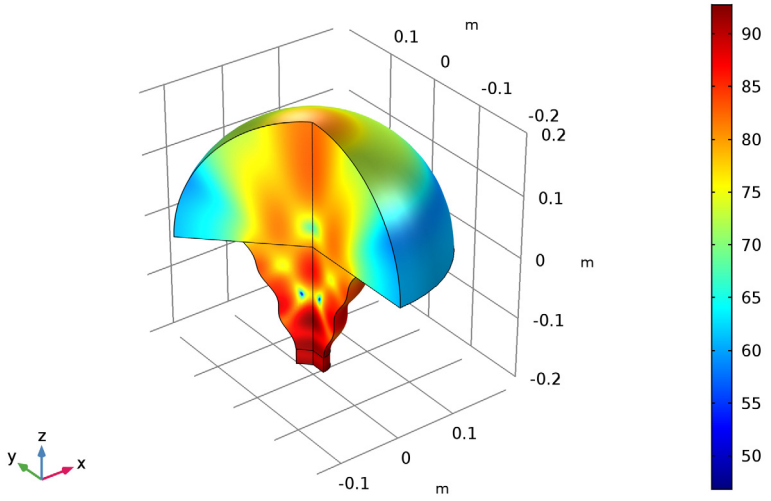


Figure 2: The final shape of the horn, optimized for on-axis SPL at 5000 Hz.

The improvement is rather small, because the initial design also shows a marked directivity, as can be seen from Figure 3. Obviously, the optimal shape with respect to on-axis SPL leads to deep undesirable minima in other directions.

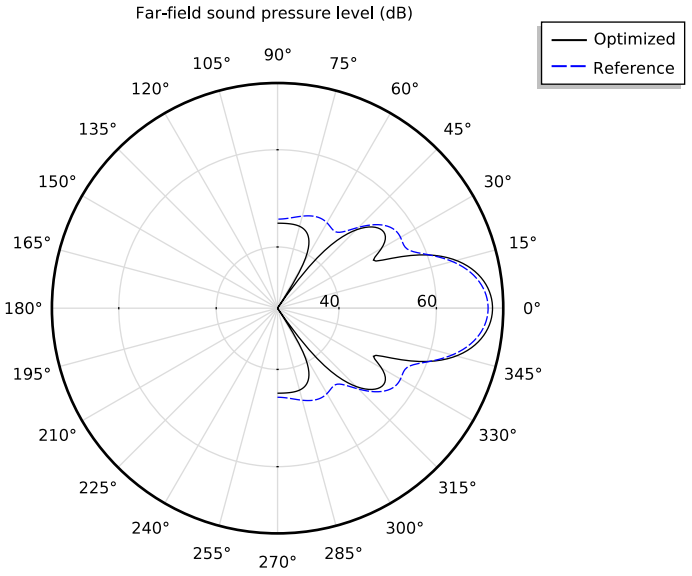


Figure 3: Radiation plot of the original (dashed blue) and final (solid black) designs.

Optimizing with respect to a slight off-axis direction can give you a more uniform far-field pattern, but may also result in a deep minimum on the axis. Try for example to set the off-axis angle Θ to 22°.

To search for a stable and practically useful horn design, you might instead create a composite objective function as a weighted sum of transmission values evaluated for a number of discrete directions, or choose to minimize the deviation from the mean SPL over a range of angles. In addition, you would also want to optimize with respect to more than one frequency, and experiment with different parameterizations.

Notes About the COMSOL Implementation

COMSOL Multiphysics implements the parametrization as a prescribed boundary displacement in a Deformed Geometry interface. The mesh is allowed to move freely in the conical part of the horn, but otherwise kept fix. Some measures must be taken to avoid inverted elements when the shape of the cone is changed. Firstly, a quad mesh is used, since quads are less likely to become inverted, compared to triangles.

Secondly, the amplitude of the boundary displacement is restricted by limits on the optimization variables. These artificial constraints are intended to keep the mesh element volumes positive at all times and must not be active at the optimum point. You perform this sanity check as a final postprocessing step.

A time-harmonic Pressure Acoustics, Frequency Domain interface solves for the pressure field inside the horn and in a small spherical domain surrounding its opening. The air domain is terminated by a spherical PML layer which absorbs outgoing waves in such a way that the artificial termination of the domain has no influence on the near field. An accurate near field is sufficient, since the far-field result is based on an integral representation evaluated in the plane of the baffle. A plane wave radiation boundary condition on the waveguide attached to the narrow end feeds the horn with a plane wave of amplitude 1 Pa.

Set up the optimization problem in the Optimization study node. The five scalar variables, q_1 to q_5 are added as control variables. These are constrained to vary in the interval [–1, 1]. The maximum effect of each variable on the boundary displacement is controlled by the scale factors d_i in Equation 1. The pressure measured by an integration coupling variable, according to Equation 4, is inserted into a scalar objective function equal to the SPL value, or $10\cdot\log_{10}(0.5\cdot\text{realdot}(p_\text{m}, p_\text{m})/(20\cdot10^{-6})^2)$.

Reference

1. E. Bängtsson, D. Noreland, and M. Berggren, "Shape Optimization of an Acoustic Horn," *Technical Report 2002-019*, Department of Information Technology, Uppsala University, May 2002.

Application Library path: `Acoustics_Module/Optimization/horn_shape_optimization`

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 **Mathematics>Deformed Mesh>Deformed Geometry (dg)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

8 Click **Done**.

GEOMETRY 1

Square 1 (sq1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `0.025`.

4 Locate the **Position** section. In the **z** text field, type `-0.2`.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.3`.

4 In the **Sector angle** text field, type `90`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.1

6 Click **Build Selected**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **2**, set **r** to `0.1`.

5 Find the **Added segments** subsection. Click **Add Linear**.

6 Find the **Control points** subsection. In row **2**, set **r** to `0.025` and **z** to `-0.175`.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **2**, set **r** to `0`.

9 Click **Build All Objects**.

10 Click the **Zoom Extents** button on the **Graphics** toolbar.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `horn_shape_optimization_parameters.txt`.

DEFINITIONS

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 6 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
pm	intop1(r*besselj(0,acpr.k*r*sin(theta))*pz)	N/m	Measured pressure
dr	q1*d1*sin(1*pi*s)+q2*d2*sin(2*pi*s)+q3*d3*sin(3*pi*s)+q4*d4*sin(4*pi*s)+q5*d5*sin(5*pi*s)	m	Radial displacement

In the definition of pm, intop1() is the name of your integration operator, r is the radial coordinate, acpr.k is the local wave number, theta is the observation angle, and pz is the derivative of the pressure with respect to the z-coordinate. dr contains the optimization parameters q1 - q5, the scale factors d1 - d5, and the local curve parameter s. For details, see Model Definition.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 4 only.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Plane Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundary 2 only.

Incident Pressure Field 1

1 On the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 In the **Settings** window for **Incident Pressure Field**, locate the **Incident Pressure Field** section.

3 In the p_0 text field, type 1.

4 From the *c* list, choose **From material**.

5 From the **Material** list, choose **Air (mat1)**.

This gives you a plane wave with the amplitude 1 Pa propagating in the positive z direction

The default boundary condition is sound hard, which is appropriate for the horn surface and the baffle, and does no harm at the PML domain's outer boundary. The PML and the incident plane-wave condition therefore fully specify the physics of the model. In order to prepare for the results processing, add a Far-Field Calculation node.

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 Select Boundary 6 only.

3 In the **Settings** window for **Far-Field Calculation**, locate the **Far-Field Calculation** section.

4 Select the **Symmetry in the z=0 plane** check box.

DEFORMED GEOMETRY (DG)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Deformed Geometry (dg)**.

2 Select Domain 2 only.

Only the domain where the shape of the horn is allowed to change needs to deform.

Free Deformation 1

1 On the **Physics** toolbar, click **Domains** and choose **Free Deformation**.

2 Select Domain 2 only.

Prescribed Mesh Displacement 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Prescribed Mesh Displacement**.

2 Select Boundary 9 only.

3 In the **Settings** window for **Prescribed Mesh Displacement**, locate the **Prescribed Mesh Displacement** section.

4 In the d_R text field, type `dr`.

MESH 1

Run the model at a frequency of 5000 Hz, corresponding to a wavelength of just under 7 cm. Using the standard at-least-six-elements-per-wavelength rule, a maximum element size of 1 cm seems like a good choice. However, the shape wavelength along the edge of the cone should also be resolved, and the maximum wavelength is therefore set to 0.5 cm. A quad mesh is in general more resistant to element warping when the mesh is deformed.

Therefore, use an unstructured quad mesh everywhere except in the PMLs, which perform better with a mapped mesh aligned with the radial and tangential directions.

Free Quad 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Quad**.

2 In the **Settings** window for **Free Quad**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1–3 only.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `0.005`.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundary 7 only.

4 In the **Settings** window for **Distribution**, locate the **Distribution** section.

5 In the **Number of elements** text field, type `8`.

6 Click **Build All**.

STUDY 1

Before starting the actual optimization, it is good practice to check the model set-up by solving once with the default parameters. This way, you can also study the reference state on which you intend to improve.

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then 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 `5000`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

3 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.

4 In the **Settings** window for **Stationary Solver**, locate the **General** section.

5 From the **Linearity** list, choose **Nonlinear**.

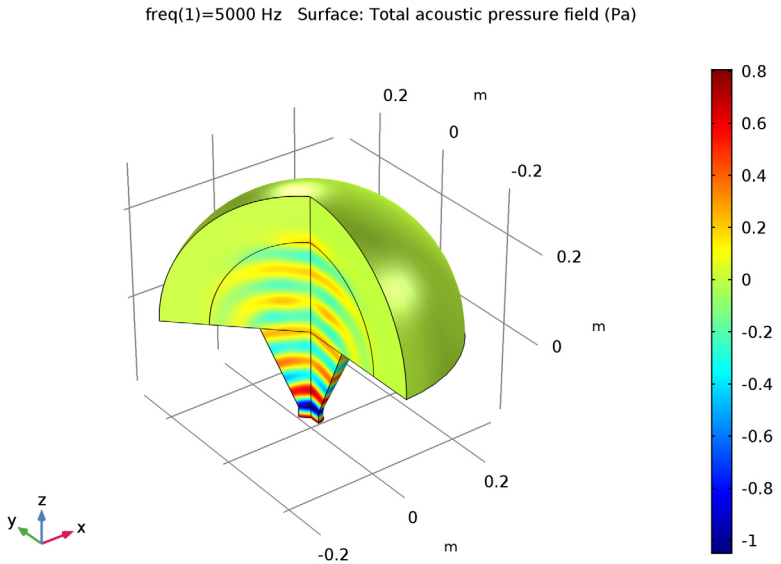
6 On the **Study** toolbar, click **Compute**.

RESULTS

Acoustic Pressure, 3D (acpr)

The default plot in the main window shows the distribution of the instantaneous pressure in the physical domain and the PML. Note that the pressure near the outer boundary of the PML is practically zero. This has no physical relevance, but indicates that the PML is doing a good job absorbing the sound.

1 Click the **Zoom Extents** button on the **Graphics** toolbar.



ROOT

Next, add a new study for the optimization.

ADD STUDY

1 On the **Study** 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Study** 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 In the **Frequencies** text field, type `5000`.

Optimization

1 On the **Study** toolbar, click **Optimization**.

2 In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.

3 From the **Method** list, choose **SNOPT**.

4 In the **Optimality tolerance** text field, type `1e-3`.

5 In the **Maximum number of model evaluations** text field, type `10000`.

6 Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description	Evaluate for
`10*log10(0.5*realdot(comp1.pm, comp1.pm)/2e-5^2)`	`Sound pressure level`	freq

7 From the **Type** list, choose **Maximization**.

8 Locate the **Control Variables and Parameters** section. Click **Load from File**.

9 Browse to the model's Application Libraries folder and double-click the file `horn_shape_optimization_control_parameters.txt`.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations> Solution 2 (sol2)>Optimization Solver 1** node, then click **Stationary 1**.

4 In the **Settings** window for **Stationary**, locate the **General** section.

5 In the **Relative tolerance** text field, type `1e-6`.

6 From the **Linearity** list, choose **Nonlinear**.

By making the nonlinear tolerance stricter than that of the optimization solver, you ensure that the optimization does not fail because each solution is not sufficiently converged. An optimality tolerance of 1e-4 is still stricter than the accuracy of this low-resolution finite element model.

7 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Optimization Solver 1>Stationary 1** node, then click **Fully Coupled 1**.

8 In the **Settings** window for **Fully Coupled**, click to expand the **Method and termination** section.

9 Locate the **Method and Termination** section. In the **Minimum damping factor** text field, type `1e-4`.

10 Right-click **Study 2>Solver Configurations>Solution 2 (sol2)>Optimization Solver 1>Stationary 1>Fully Coupled 1** and choose **Compute**.

RESULTS

Acoustic Pressure (acpr) 1

When the solution is ready, you can immediately see the optimal (with the constraints imposed by the parameterization) shape of the horn. The PML does not add any physical information and can be excluded for clarity.

Study 2/Solution 2 (2) (sol2)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Study 2/Solution 2 (2) (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1–3 only.

Sound Pressure Level, 3D (acpr) 1

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level, 3D (acpr) 1**.

2 On the **Sound Pressure Level, 3D (acpr) 1** toolbar, click **Plot**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Your plot of the sound pressure level should now look like Figure 2.

To see a direct comparison of the far-field polar pattern before and after optimization, modify the second to last default plot. This is a far-field plot of the sound pressure level in the rz-plane. Modify it to only plot the results in the positive half plane ($z > 0$), increase the resolution, and change some Coloring and Style options. The resulting plot of the far field should look like Figure 3. Note that 0 deg on the polar graph corresponds to the vertical z-axis.

Far-Field Sound Pressure Level (acpr) 1

1. In the **Model Builder** window, under **Results** click **Far-Field Sound Pressure Level (acpr) 1**.
2. In the **Settings** window for **Polar Plot Group**, click to expand the **Title** section.
3. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `Far-field sound pressure level (dB)`.

Far Field 1

1. In the **Model Builder** window, expand the **Far-Field Sound Pressure Level (acpr) 1** node, then click **Far Field 1**.
2. In the **Settings** window for **Far Field**, locate the **Evaluation** section.
3. Find the **Angles** subsection. In the **Number of angles** text field, type `180`.
4. From the **Restriction** list, choose **Manual**.
5. In the ϕ **start** text field, type `-90`.
6. In the ϕ **range** text field, type `180`.
7. Click to expand the **Legends** section. Select the **Show legends** check box.
8. From the **Legends** list, choose **Manual**.
9. In the table, enter the following settings:

Legends
Optimized

10. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Far Field 2

1. Right-click **Results>Far-Field Sound Pressure Level (acpr) 1>Far Field 1** and choose **Duplicate**.
2. In the **Settings** window for **Far Field**, locate the **Data** section.
3. From the **Data set** list, choose **Study 1/Solution 1 (1) (sol1)**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Reference

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

6 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 On the **Far-Field Sound Pressure Level (acpr) 1** toolbar, click **Plot**.

Derived Values

Optionally, evaluate the values of the optimization parameters.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (2) (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Solver>Control parameters>q1 - Optimization parameter, q1**.

5 Click **New Table**.

Global Evaluation 2

1 Right-click **Global Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
q2		Optimization parameter, q2

4 Click **Table 2 - Global Evaluation 1 (q1)**.

5 Right-click **Results>Derived Values>Global Evaluation 2** and choose **Duplicate**.

6 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

7 In the table, enter the following settings:

Expression	Unit	Description
q3		Optimization parameter, q3

8 Click **Table 2 - Global Evaluation 1 (q1)**.

9 Right-click **Global Evaluation 2** and choose **Duplicate**.

10 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

11 In the table, enter the following settings:

Expression	Unit	Description
q4		Optimization parameter, q4

12 Click **Table 2 - Global Evaluation 1 (q1)**.

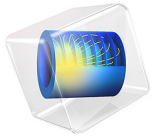
13 Right-click **Global Evaluation 2** and choose **Duplicate**.

14 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

15 In the table, enter the following settings:

Expression	Unit	Description
q5		Optimization parameter, q5

16 Click **Table 2 - Global Evaluation 1 (q1)**.



Jet Pipe

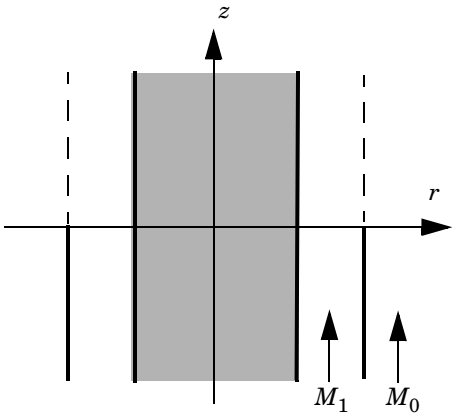
This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example models the radiation of fan noise from the annular duct of a turbofan aeroengine. When the jet stream excites the duct, a vortex sheet appears along the extension of the duct wall. In the model you calculate the near field on both sides of the vortex sheet. The background mean-flow is assumed to be well described by a potential flow, in this model a uniform flow. This means that, the acoustic field can be modeled by solving the linearized potentiality flow equations in the frequency domain.

Model Definition

The model is axisymmetric with the symmetry axis coinciding with the engine's centerline (gray area in the figure below). The flows both inside and outside the duct are uniform mean flows, they have a magnitude of M_1 and M_0, respectively. Because the flow velocities differ, a vortex sheet separates them (dashed line in the figure below).



Sketch of the turbofan motor.

The Linearized Potential Flow, Frequency Domain interface in the Acoustics Module describes acoustic waves in a moving fluid with the potential ϕ, for the local particle velocity as the basic dependent variable; see the chapter about aeroacoustics in the *Acoustics Module User's Guide* for details. The field equations are only valid when the background velocity field is irrotational, a condition that is not satisfied across a vortex sheet. As a consequence, the velocity potential is discontinuous across this sheet. To model this discontinuity you use the Vortex Sheet boundary condition which is available on

interior boundaries. The boundary conditions on the two sides of the vortex sheet are defined as follows:

$$\left[\mathbf{n}\cdot\left(\rho_0\nabla\phi - \mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\right)\right]_i = [\rho_0(i\omega + \mathbf{V}\cdot\nabla)w]_i \qquad i = \text{up, down}$$

$$p_{\text{up}} = p_{\text{down}} \qquad w_{\text{up}} = -w_{\text{down}}$$

In these equations, ω is the angular velocity, $\mathbf{V}$ is the mean-flow velocity, w is the outward normal displacement, ϕ is the velocity potential, and p is the pressure. The subscripts "up" and "down" refer to the two sides of the boundary.

The velocity normal to the vortex sheet is zero, which implies that the last two terms on the left-hand side of the condition vanishes. In the model the variables are made dimensionless. The velocities are divided by the speed of sound in air and the densities are divided by the density for air. For example, the model uses the Mach number $M = V/c_0$ as the mean flow velocity. This leads to the boundary conditions

$$(i\omega + M_{\text{up}}\nabla_T)w = \frac{\partial\phi_{\text{up}}}{\partial n}$$

$$(i\omega + M_{\text{down}}\nabla_T)w = \frac{\partial\phi_{\text{down}}}{\partial n}$$

$$p_{\text{up}} = p_{\text{down}} \qquad w_{\text{up}} = -w_{\text{down}}$$

where M denotes the transverse Mach number.

The duct has a hard wall, which you also model using an interior boundary condition.

The acoustic field inside the duct can be described as a sum of eigenmodes propagating in the duct and then radiating in the free space. This is discussed in section 2.1 in Ref. 1. In this example you study the radiated acoustic waves produced by a single eigenmode at a time. First you calculate the eigenmodes with the circumferential mode order 4 on the inlet boundary. From these eigenmodes, the one with radial mode order 0 is used as incident wave. You then calculate the velocity fields with circumferential mode numbers m = 17 and 24 and with radial mode order n = 1.

Results and Discussion

The inlet sources are found using a boundary mode analysis. The analysis is made with the circumferential wave number m = 4, 17, and 24 and gives several eigenmodes corresponding to different radial mode numbers. This example, like Ref. 1, uses the

eigenmodes $(m,n) = (4,0)$, $(17,1)$, and $(24,1)$ as incident waves in the duct. The modes are depicted in Figure 1. The radial mode $n = 0$ corresponds to the largest eigenvalue for a given m, while $n = 1$ corresponds to the smallest eigenvalue.

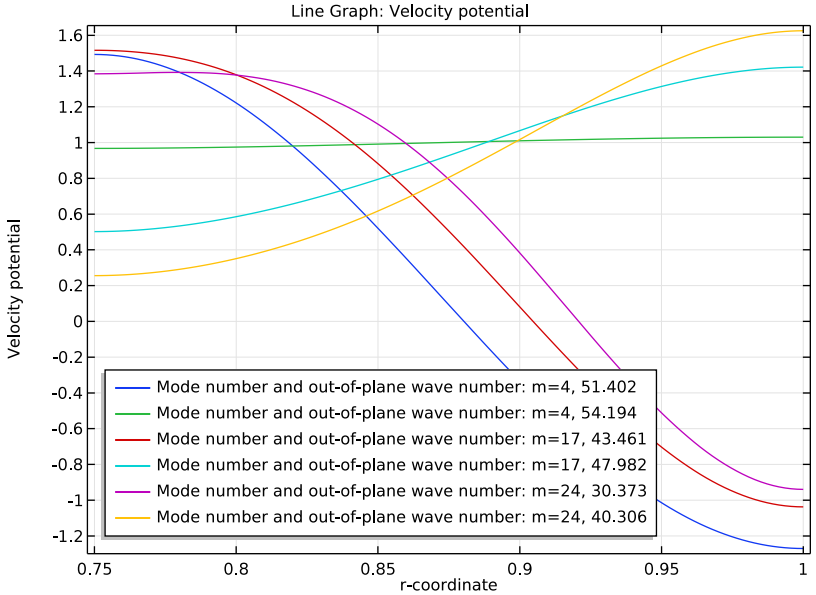


Figure 1: Mode shape for m = 4, 17, and 24 for both n = 0 and n=1

In Figure 2 the source velocity potential is depicted in the revolved geometry including the circumferential mode contribution, given as

$$\phi \cdot e^{-im\varphi}$$

where m is the mode number and φ is the azimuthal angle.

The near field pressure around the duct obtained by COMSOL Multiphysics can be compared to the results for the near field in Ref. 1. Figure 3 through Figure 7 show the near-field solution for a Mach number equal to $M_1 = 0.45$ in the pipe and $M_0 = 0.25$ on the outside. The figures show the pressure field for the different source eigenmodes shown in Figure 1.

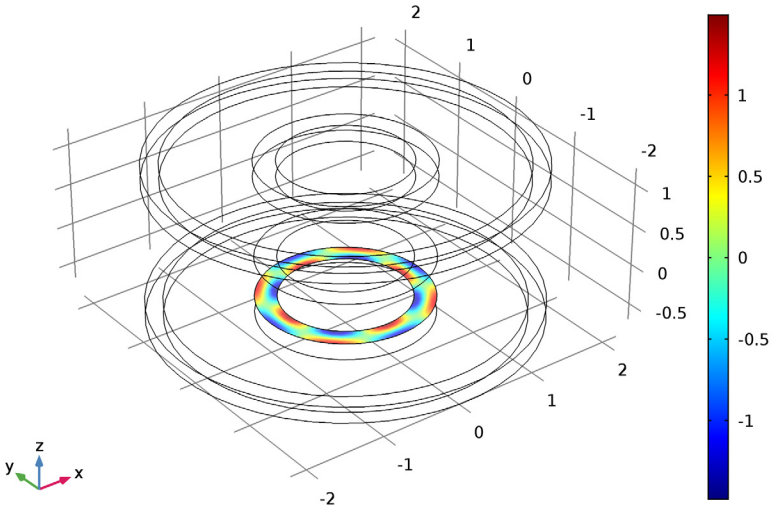


Figure 2: The boundary mode (m,n) = (4,0) depicted in the revolved geometry including the circumferential wave number contribution.

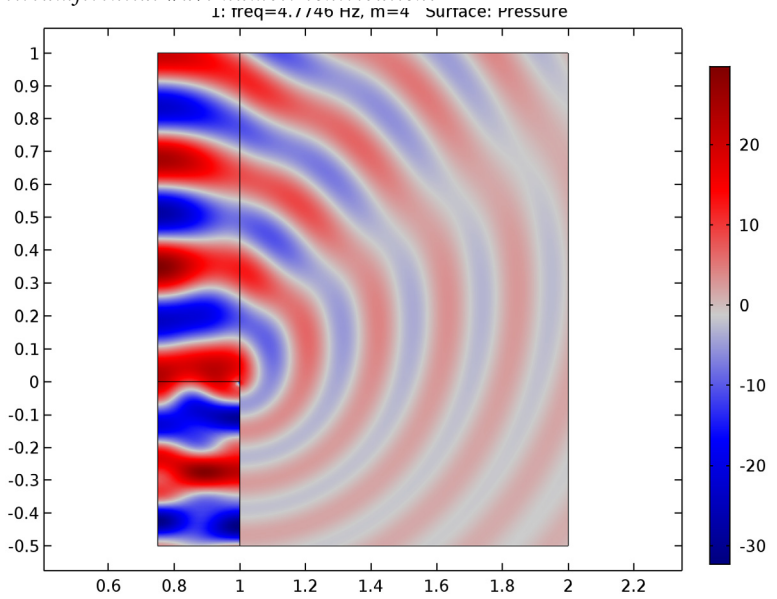


Figure 3: The near-field solution for m = 4 and n = 0.

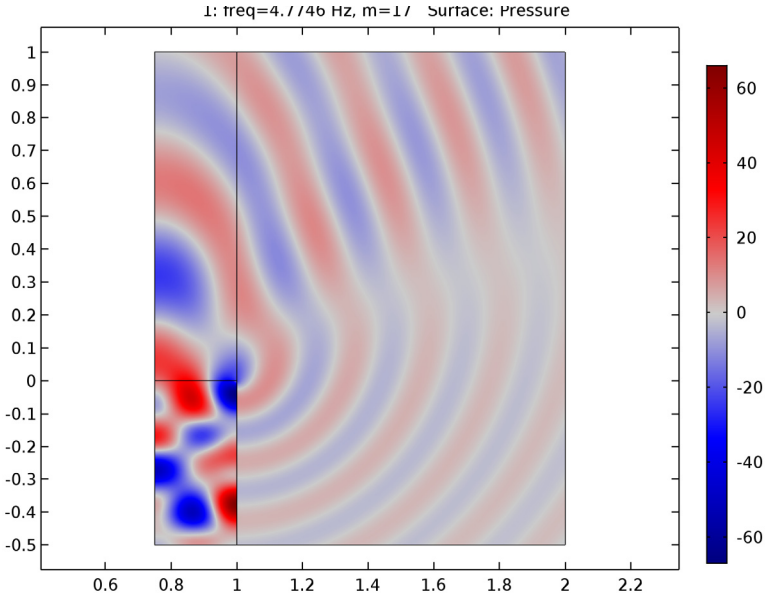


Figure 4: The near-field solution for m = 17 and n = 1.

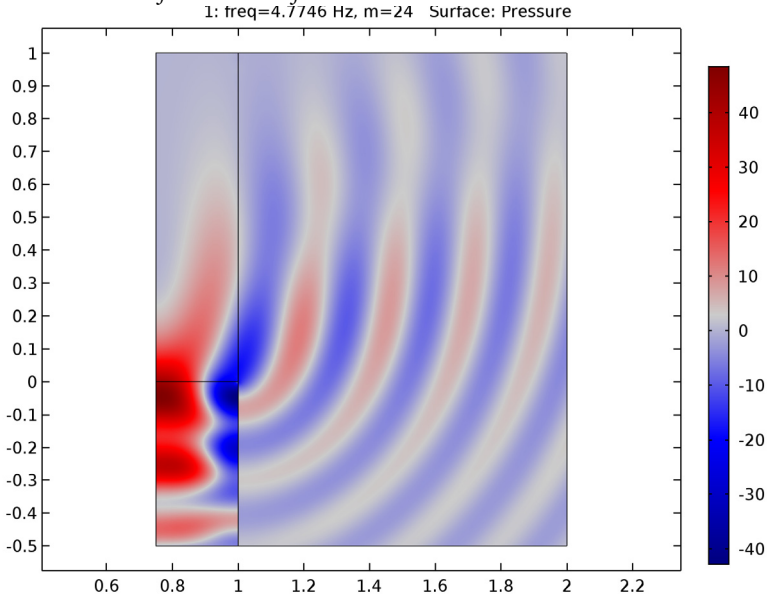


Figure 5: The near-field solution for m = 24 and n = 1.

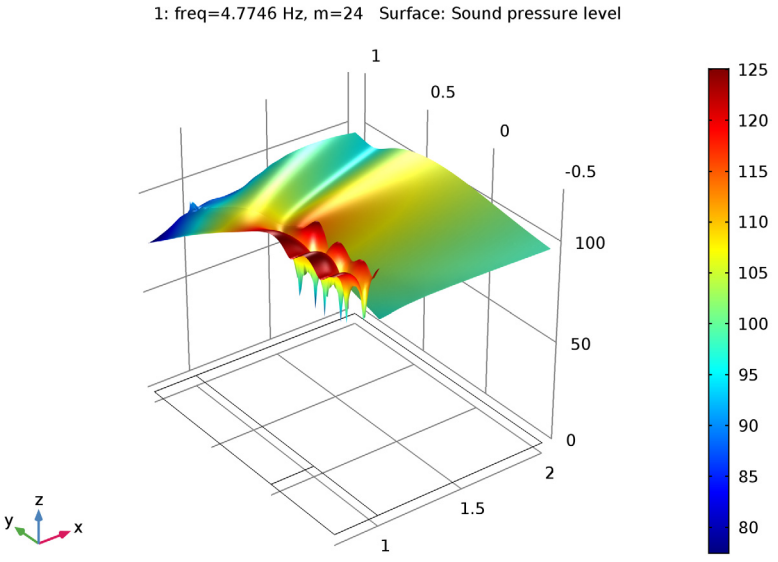


Figure 6: The near-field sound pressure level for m = 24 and n = 1.

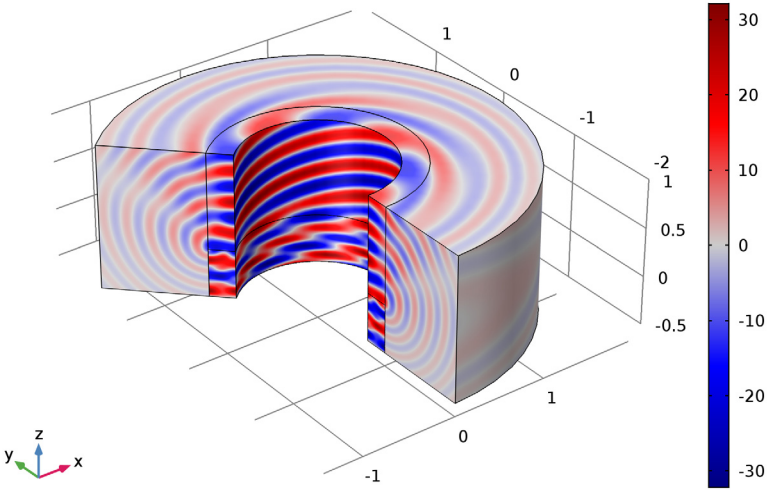


Figure 7: The near-field pressure plotted in the revolved geometry for m = 4 and n = 0.

Reference

1. G. Gabard and R.J. Astley, "Theoretical Model for Sound Radiations from Annular Jet Pipes: Far- and Near-field Solution," *J. Fluid Mech.*, vol. 549, pp. 315–341, 2006.

Application Library path: `Acoustics_Module/Aeroacoustics_and_Noise/jet_pipe`

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>Aeroacoustics>Linearized Potential Flow, Boundary Mode (aebm)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Acoustics>Aeroacoustics>Linearized Potential Flow, Frequency Domain (ae)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Mode Analysis**.

8 Click **Done**.

ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Unit System** section.

3 From the **Unit system** list, choose **None**.

This setting turns off all unit support in the model.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.25`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. In the **r** text field, type `0.75`.

6 In the **z** text field, type `-0.5`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.25`.

4 Locate the **Position** section. In the **r** text field, type `0.75`.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.25`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **r** text field, type `0.75`.

6 In the **z** text field, type `1`.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Rectangle 4 (r4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type `1.5`.

4 Locate the **Position** section. In the **r** text field, type `1`.

5 In the **z** text field, type `-0.5`.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Rectangle 5 (r5)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1.2`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **r** text field, type `1`.

6 In the **z** text field, type `-0.7`.

Rectangle 6 (r6)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.2`.

4 In the **Height** text field, type `1.9`.

5 Locate the **Position** section. In the **r** text field, type `2`.

6 In the **z** text field, type `-0.7`.

Rectangle 7 (r7)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1.2`.

4 In the **Height** text field, type `0.2`.

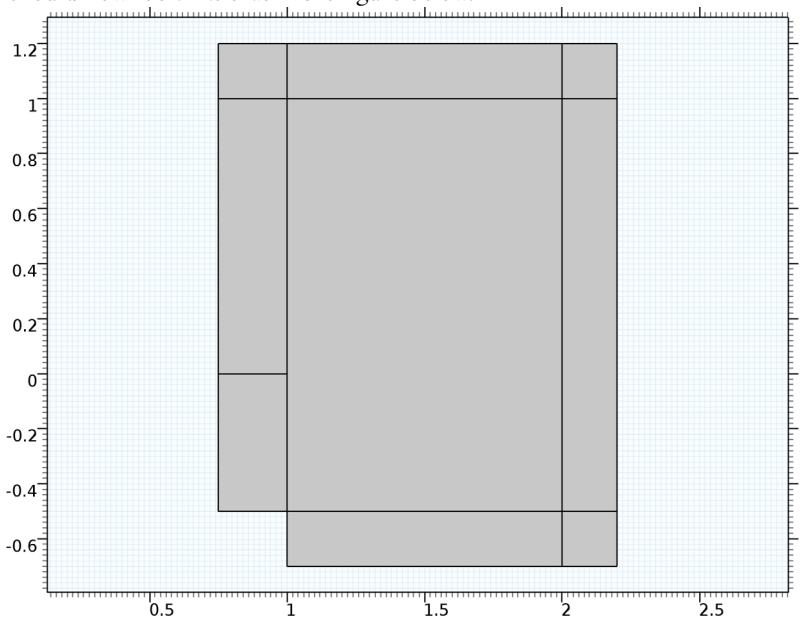
5 Locate the **Position** section. In the **r** text field, type `1`.

6 In the **z** text field, type `1`.

7 Click **Build All Objects**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

This completes the geometry-modeling state. The geometry in the Graphics window should now look like that in the figure below.



GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
M0	0.25	0.25	Mach number outside the duct
M1	0.45	0.45	Mach number inside the duct
m	4	4	Circumferential wave number
f	30/(2*pi)	4.7746	Frequency
k0	2*pi*f/(1-M1)	54.545	Largest wave number

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Duct Cross Section` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 2 only.

MATERIALS

Specify the density and speed of sound, both normalized to 1, as material parameters. You need to add a separate material node for the duct cross section because it is a boundary and not a domain.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1		Basic
Speed of sound	c	1		Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Duct Cross Section**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1		Basic
Speed of sound	c	1		Basic

LINEARIZED POTENTIAL FLOW, BOUNDARY MODE (AEBM)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Boundary Mode (aebm)**.

2 In the **Settings** window for **Linearized Potential Flow, Boundary Mode**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Duct Cross Section**.

4 Click to expand the **Equation** section. Locate the **Linearized Potential Flow Equation Settings** section. In the m text field, type `m`.

Linearized Potential Flow Model 1

1 In the **Model Builder** window, expand the **Linearized Potential Flow, Boundary Mode (aebm)** node, then click **Linearized Potential Flow Model 1**.

2 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.

3 Specify the **V** vector as

0	r
M1	z

LINEARIZED POTENTIAL FLOW, FREQUENCY DOMAIN (AE)

On the **Physics** toolbar, click **Linearized Potential Flow, Boundary Mode (aebm)** and choose **Linearized Potential Flow, Frequency Domain (ae)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Linearized Potential Flow, Frequency Domain (ae)**.

2 In the **Settings** window for **Linearized Potential Flow, Frequency Domain**, locate the **Linearized Potential Flow Equation Settings** section.

3 In the m text field, type `m`.

4 Locate the **Typical Wave Speed for Perfectly Matched Layers** section. In the c_{ref} text field, type `1[m/s]`.

5 In the **Model Builder** window, click **Linearized Potential Flow Model 1**.

6 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.

7 Specify the **V** vector as

0	r
M0	z

8 In the **Model Builder** window, click **Linearized Potential Flow, Frequency Domain (ae)**.

Vortex Sheet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Vortex Sheet**.

2 Select Boundaries 12 and 13 only.

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 Select Boundary 10 only.

Linearized Potential Flow Model 2

1 On the **Physics** toolbar, click **Domains** and choose **Linearized Potential Flow Model**.

2 Select Domains 1–3 only.

3 In the **Settings** window for **Linearized Potential Flow Model**, locate the **Linearized Potential Flow Model** section.

4 Specify the $\mathbf{V}$ vector as

0	r
M1	z

Velocity Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Velocity Potential**.

The value of the velocity potential is given by the boundary mode solutions. The dependent variable is `phi`.

2 In the **Settings** window for **Velocity Potential**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Duct Cross Section**.

4 Locate the **Velocity Potential** section. In the ϕ_0 text field, type `phi`.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

3 From the **Coordinate stretching type** list, choose **Rational**.

4 Locate the **Geometry** section. From the **Type** list, choose **Cylindrical**.

5 Select Domains 3, 4, and 6–9 only.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Extremely fine**.

4 Click **Build All**.

Start by solving the boundary mode problem to find the source potentials that will be applied in the full model.

STUDY 1

1 In the **Settings** window for **Study**, type `Study 1 - Mode Analysis` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Parametric Sweep

On the **Study** toolbar, click **Parametric Sweep**.

STUDY 1 - MODE ANALYSIS

Parametric Sweep

1 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

2 Click **Add**.

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
m (Circumferential wave number)	4 17 24	

Step 1: Mode Analysis

1 In the **Model Builder** window, under **Study 1 - Mode Analysis** click **Step 1: Mode Analysis**.

2 In the **Settings** window for **Mode Analysis**, locate the **Study Settings** section.

3 In the **Mode analysis frequency** text field, type `f`.

4 From the **Mode search method around shift** list, choose **Smaller real part**.

5 Select the **Desired number of modes** check box.

6 In the associated text field, type `2`.

7 Select the **Search for modes around** check box.

8 In the associated text field, type `k0`.

9 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Aeroacoustics, Frequency Domain** interface.

10 On the **Study** toolbar, click **Compute**.

RESULTS

1D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Boundary Modes` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1 - Mode Analysis/ Parametric Solutions 1 (sol2)**.

4 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

Line Graph 1

1 Right-click **Boundary Modes** and choose **Line Graph**.

2 Select Boundary 2 only.

3 On the **Boundary Modes** toolbar, click **Plot**.

4 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

5 From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `r`.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 Find the **Include** subsection. In the **Prefix** text field, type `Mode number and out-of-plane wave number:` .

9 On the **Boundary Modes** toolbar, click **Plot**.

The graph that appears should be the same as that in Figure 1. This shows the two radial modes (n=0 and 1) for m = 4, 17, and 24.

Revolution 2D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1 - Mode Analysis/Parametric Solutions 1 (sol2)**.

4 Click to expand the **Advanced** section. Select the **Define variables** check box.

3D Plot Group 2

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Boundary Modes 2D Revolved` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Revolution 2D 1**.

4 From the **Parameter value (m)** list, choose **4**.

Surface 1

1 Right-click **Boundary Modes 2D Revolved** and choose **Surface**.

Add the circumferential behavior by using the rev1phi coordinate that exists in the revolved solution data set.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `phi*exp(-i*rev1phi*m)`.

4 On the **Boundary Modes 2D Revolved** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The graph that appears should be the same as that in Figure 2.

Now, add three studies to solve three cases with different sources: (m=4,n=0), (m=17, n=1), and (m=24,n=1), respectively. After setting up three studies and solving then plot the results.

First, solve the case with the circumferential wave number m=4 and the first radial mode (n=0). That is the mode with the highest eigenvalue.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Linearized Potential Flow, Boundary Mode (aebm)** interface.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Settings** window for **Study**, type `Study 2 - m=4 and n=0` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 2 - M=4 AND N=0

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - m=4 and n=0** 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 `f`.

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - Mode Analysis, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol2)**.

8 From the **Use** list, choose **m=4 (sol3)**.

9 From the **Out-of-plane wave number** list, choose **54.194**.

The value of m is not stored in the mode analysis data set so you need to set it manually as follows.

10 In the **Settings** window for **Frequency Domain**, click to expand the **Study extensions** section.

11 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

12 Click **Add**.

13 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
m (Circumferential wave number)	4	

14 On the **Home** toolbar, click **Compute**.

Secondly, solve the case with the circumferential wave number m=17 and the second radial mode (n=1). That is the mode with the second highest eigenvalue. The steps are the same as for Study 2.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Linearized Potential Flow, Boundary Mode (aebm)** interface.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

1 In the **Settings** window for **Study**, type `Study 3 - m=17 and n=1` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 3 - M=17 AND N=1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3 - m=17 and n=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 `f`.

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - Mode Analysis, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol2)**.

8 From the **Use** list, choose **m=17 (sol4)**.

9 From the **Out-of-plane wave number** list, choose **43.461**.

The value of m is not stored in the mode analysis data set so you need to set it manually as follows.

10 In the **Settings** window for **Frequency Domain**, click to expand the **Study extensions** section.

11 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

12 Click **Add**.

13 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
m (Circumferential wave number)	17	

14 On the **Home** toolbar, click **Compute**.

Finally, solve the case with the circumferential wave number m=27 again with the second radial mode (n=1). That is the mode with the second highest eigenvalue. The steps are the same as for Study 2 and Study 3.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Linearized Potential Flow, Boundary Mode (aebm)** interface.

4 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

5 Click **Add Study** in the window toolbar.

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

1 In the **Settings** window for **Study**, type `Study 4 - m=24 and n=1` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 4 - M=24 AND N=1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 4 - m=24 and n=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 `f`.

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - Mode Analysis, Mode Analysis**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol2)**.

8 From the **Use** list, choose **m=24 (sol5)**.

9 From the **Out-of-plane wave number** list, choose **30.373**.

The value of m is not stored in the mode analysis data set so you need to set it manually as follows.

10 In the **Settings** window for **Frequency Domain**, click to expand the **Study extensions** section.

11 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

12 Click **Add**.

13 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
m (Circumferential wave number)	24	

14 On the **Home** toolbar, click **Compute**.

Proceed to plotting the pressure near field solution shown in Figure 3, Figure 4, and Figure 5.

First, add selections to the data sets such that the results are not shown in the PML domains. In here the solutions is not physical.

RESULTS

Study 2 - m=4 and n=0/Solution 6 (sol6)

In the **Model Builder** window, under **Results>Data Sets** click **Study 2 - m=4 and n=0/Solution 6 (sol6)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 1, 2, and 5 only.

Study 3 - m=17 and n=1/Solution 7 (sol7)

In the **Model Builder** window, under **Results>Data Sets** click **Study 3 - m=17 and n=1/Solution 7 (sol7)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 1, 2, and 5 only.

Study 4 - m=24 and n=1/Solution 8 (sol8)

In the **Model Builder** window, under **Results>Data Sets** click **Study 4 - m=24 and n=1/Solution 8 (sol8)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1, 2, and 5 only.

Now, create the plots.

2D Plot Group 3

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Near Field Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 - m=4 and n=0/Solution 6 (sol6)**.

Surface 1

1 Right-click **Near Field Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `ae.p`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

5 On the **Near Field Pressure** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

The graph that appears should be the same as that in Figure 3. Change the evaluation data set to generate the graphics in Figure 4 and Figure 5.

Near Field Pressure

1 In the **Model Builder** window, under **Results** click **Near Field Pressure**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3 - m=17 and n=1/Solution 7 (sol7)**.

4 On the **Near Field Pressure** toolbar, click **Plot**.

The graph that appears should be the same as that in Figure 4.

5 In the **Model Builder** window, click **Near Field Pressure**.

6 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

7 From the **Data set** list, choose **Study 4 - m=24 and n=1/Solution 8 (sol8)**.

8 On the **Near Field Pressure** toolbar, click **Plot**.

The graph that appears should be the same as that in Figure 5.

Create a plot of the sound pressure level as the one shown in Figure 6. Just as for the pressure plot you can change the evaluation data set to see the other solutions.

2D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Near Field SPL` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 4 - m=24 and n=1/ Solution 8 (sol8)**.

Surface 1

1 Right-click **Near Field SPL** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `ae.Lp`.

Height Expression 1

1 Right-click **Results>Near Field SPL>Surface 1** and choose **Height Expression**.

2 On the **Near Field SPL** toolbar, click **Plot**.

Finally, plot the pressure in the revolved geometry including the circumferential wave number and create Figure 7.

Revolution 2D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - m=4 and n=0/Solution 6 (sol6)**.

4 Click to expand the **Revolution layers** section. Locate the **Revolution Layers** section. In the **Start angle** text field, type `-90`.

5 In the **Revolution angle** text field, type `225`.

6 Click to expand the **Advanced** section. Select the **Define variables** check box.

3D Plot Group 5

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Near Field Pressure 2D Revolved` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Revolution 2D 2**.

Surface 1

1 Right-click **Near Field Pressure 2D Revolved** and choose **Surface**.

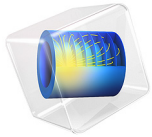
Add the circumferential behavior by using the `rev2phi` coordinate that exists in the revolved solution data set.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `ae.p*exp(-i*m*rev2phi)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

5 On the **Near Field Pressure 2D Revolved** toolbar, click **Plot**.



Loudspeaker Driver

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of a loudspeaker of the dynamic cone driver type, common for low and medium frequencies. The instructions walk you through modeling its electromagnetic, structural, and acoustic properties. The output from the model includes the total electric impedance and the sensitivity (the on-axis sound pressure level at a nominal driving voltage) as functions of the frequency. The spatial characteristics of the speaker are depicted in a directivity plot.

When performing the acoustic measurements in this model, the driver is set up in an infinite baffle—a wide reflective surface acting to shut out the sound produced on the backside of the cone. An extended 3D version of the model, Loudspeaker Driver in a Vented Enclosure, uses the electromechanical properties modeled here and adds a vented enclosure.

The model is set up with a combination of the Magnetic Fields interface from the AC/DC Module and the Acoustic-Structure Interaction multiphysics interface from the Acoustics Module. A first, optional analysis solves only the electromagnetic part of the problem, with the driver in stand-still. From here, a driving force factor and the blocked voice coil impedance can be extracted and exported. The second and final analysis is of the full model, including the relevant multiphysics interactions all the way from the driving voltage to the computed sound pressure level.

Note: This application requires the AC/DC Module.

Model Definition

Figure 1 shows the geometry of the baffled driver with its functional parts. The field from the *magnet* is supported and focused by the iron *pole piece* and *top plate* to the thin gap where the *voice coil* is wound around a former extending from the apex of the *cone*. Although the voice coil consists of many turns of wire, it is for simplicity drawn and modeled as a homogenized domain. When a driving AC voltage is applied to the voice coil, the resulting force causes it to vibrate, and the cone to create sound.

The *dust cap* protects the magnetic motor. In this design, it is made of the same stiff and light composite material as the cone and also contributes to the sound. A centered hole in the pole piece counteracts pressure buildup beneath the dust cap. The *suspension*,

consisting of the *surround*, made of a light foam material, and the *spider*, a flexible cloth, keep the cone in place and provide damping and spring forces.

The outer perimeters of the magnet and suspension are normally attached to a *basket*, a hollow supporting metal structure. The basket is not included in this model, but the magnet assembly and outer rims of the spider and surround are considered to be fixed. The omission of the basket means that the considered geometry is rotationally symmetric and can be modeled in the *rz*-plane.

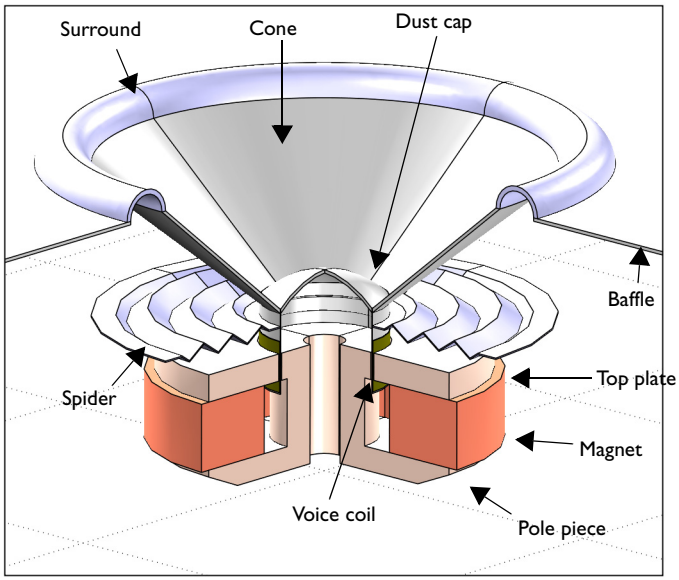


Figure 1: Geometry of the modeled loudspeaker driver.

The loudspeaker is driven by a time-harmonic voltage, $V = V_0\exp(i\omega t)$, applied to the voice coil. The following theory section first describes the electromagnetic analysis of the current in the voice coil and the driving force that this current gives rise to. The relation between the driving voltage and the force on the voice coil as well as the so-called back EMF are easily set up in COMSOL using built-in functionality. This force is then applied in an acoustic-structure interaction analysis to compute the sound generation. More details are given in Electromagnetic Interactions.

The structural equation is solved in the moving parts of the driver, and a pressure acoustics equation in the surrounding air. The pressure acoustics equation is automatically excited by the structural vibrations, and feeds back the pressure load onto the structure, using the built in *Multiphysics* coupling.

The air domains and the baffle should ideally extend to infinity. To avoid unphysical reflections where you truncate the geometry, you use a perfectly matched layer (PML), as seen in Figure 2. For more information about PMLs, see the *COMSOL Multiphysics Reference Manual*.

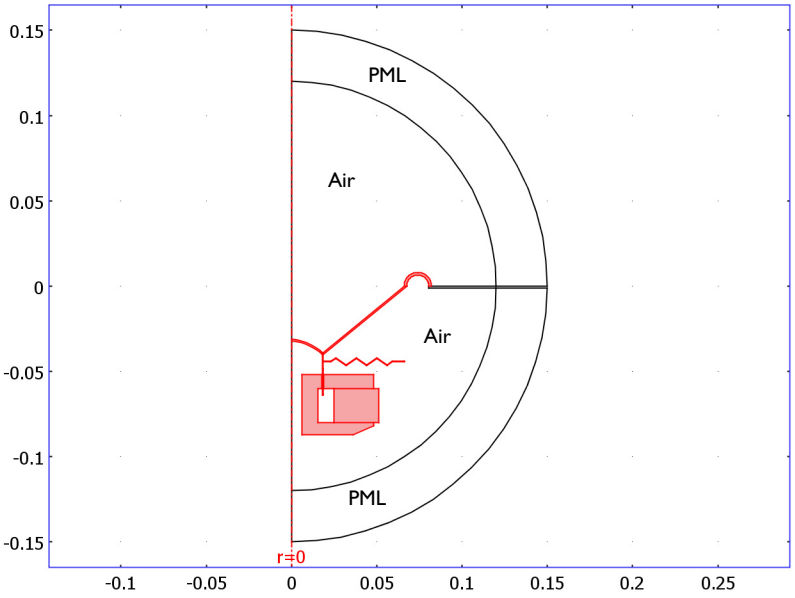


Figure 2: Overview of the model geometry with the structures in pink.

Although the modeled air domain has a radius of only 0.12 m, the local acoustic pressure and phase can be extracted anywhere outside the computational domain by using the far-field pressure computations. The sensitivity is calculated as the sound pressure level on the axis at a radius of 1 m, for the applied voltage $V_0 = 4$ V. This functionality also allows the computation of the Directivity plot for the speaker, again evaluated at 1 m.

ELECTROMAGNETIC INTERACTIONS

The Lorentz force on a wire of length L and with the current $\mathbf{I}$ in an externally generated magnetic flux density $\mathbf{B}$ perpendicular to the wire is given by $F = L\mathbf{I}\times\mathbf{B}$. The voice coil consists of a single copper wire making $N_0 = 100$ turns. The coil is homogenized so that

$$N_0 I = \int_A J_\varphi dA$$

where J_φ is the azimuthally directed current density through a cross-section of the coil, and the integral is taken over its area in the rz-plane. The total driving force on the coil hence becomes

$$F_e = -\int_V J_\phi B_r \mathrm{d}V \tag{1}$$

with B_r being the r-component of the magnetic flux density, and the integral evaluated over the volume occupied by the coil domain.

The Lorentz force is applied to the voice coil domain as a *Body Load* with the input set to *Lorentz force contribution (mf)*.

The current through the voice coil relates to the applied voltage as

$$I = (V_0 + V_{be})/Z_b \tag{2}$$

where Z_b is the *blocked electric impedance* (the electric impedance of the voice coil measured while the speaker's moving parts are stationary) and $-V_{be}$ denotes the *back EMF* (the voltage induced in the coil due to its motion through the permanent magnetic field in the gap).

The back EMF is automatically added to the voice coil using the *Velocity (Lorentz Term)* feature available in the AC/DC Module. The velocity is given directly from the Solid Mechanics interface by the variables: `solid.u_tR, solid.u_tZ`.

EXPORT TO LOUDSPEAKER DRIVER IN A VENTED ENCLOSURE

If you write Equation 1 in terms of the coil current I rather than the cross-sectional current density, you get

$$F_e = -\frac{2\pi I N_0}{A}\int r B_r \mathrm{dA} \tag{3}$$

as it is assumed that $J_\phi = I{\cdot}N_0/A$ and is constant in the coil cross-section of area A. The common factor in F_e and V_{be} is known in the loudspeaker community as the force factor, BL:

$$\mathrm{BL} = -\frac{2\pi N_0}{A}\int r B_r \mathrm{dA} \tag{4}$$

Note that if $A \to 0$, the integral becomes equal to a magnetic flux density times the length of the coil; hence the name.

With knowledge of BL and the frequency-dependent Z_b, Equation 2, Equation 3, and Equation 4 can be rearranged to form a relationship between V_0 and F_e:

$$F_e = \frac{BLV_0}{Z_b} - v\frac{(BL)^2}{Z_b}$$

This is the expression used for the driving force in the Loudspeaker Driver in a Vented Enclosure model. Note the dependence on the velocity of the moving coil, which is unknown prior to the acoustic-structure interaction computation.

Results and Discussion

The magnetic field in and around the magnetic motor appears in Figure 3. The maximum field in the air arises in the gap between the pole piece and the top plate. Performing the integral in Equation 4 over the voice coil domain gives a force factor BL = 7.63 N/A.

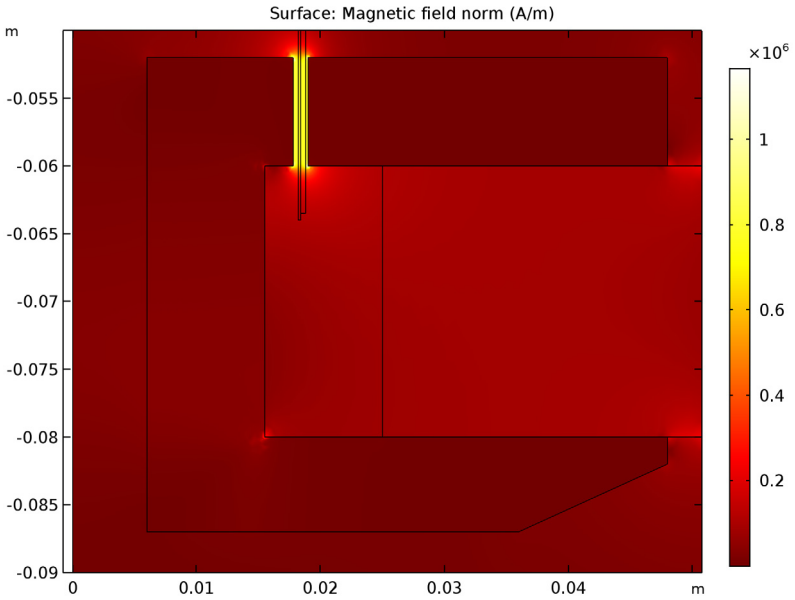


Figure 3: Magnetic field in and around the magnetic motor.

The iron in the pole piece and top plate is modeled as a nonlinear magnetic material, with the relationship between the B and H fields described by interpolation from measured data. Figure 4 shows the local effective relative permeability $\mu_r = B/(\mu_0 H)$. The plot shows that the iron is close to saturation in the center of the pole piece, but remains in the linear

regime above and below the magnet. This indicates that if you want to use less material, you can likely decrease the radius of the pole piece and top plate with very little effect on the magnetic field in the gap.

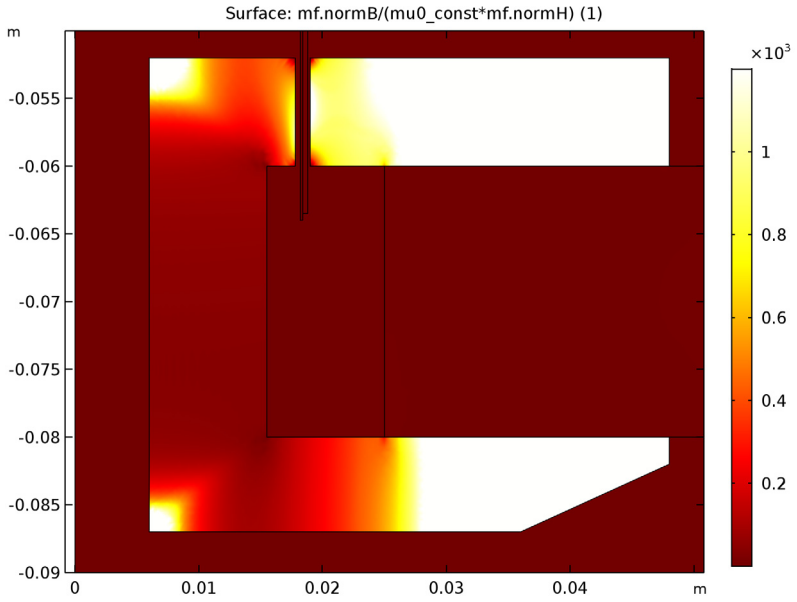


Figure 4: The local relative permeability in the pole piece and top plate, when subjected to the field from the magnet.

In computing the blocked coil impedance, the AC equation is linearized around the local permeability resulting from the static solution. Figure 5 shows the induced currents at two different frequencies.

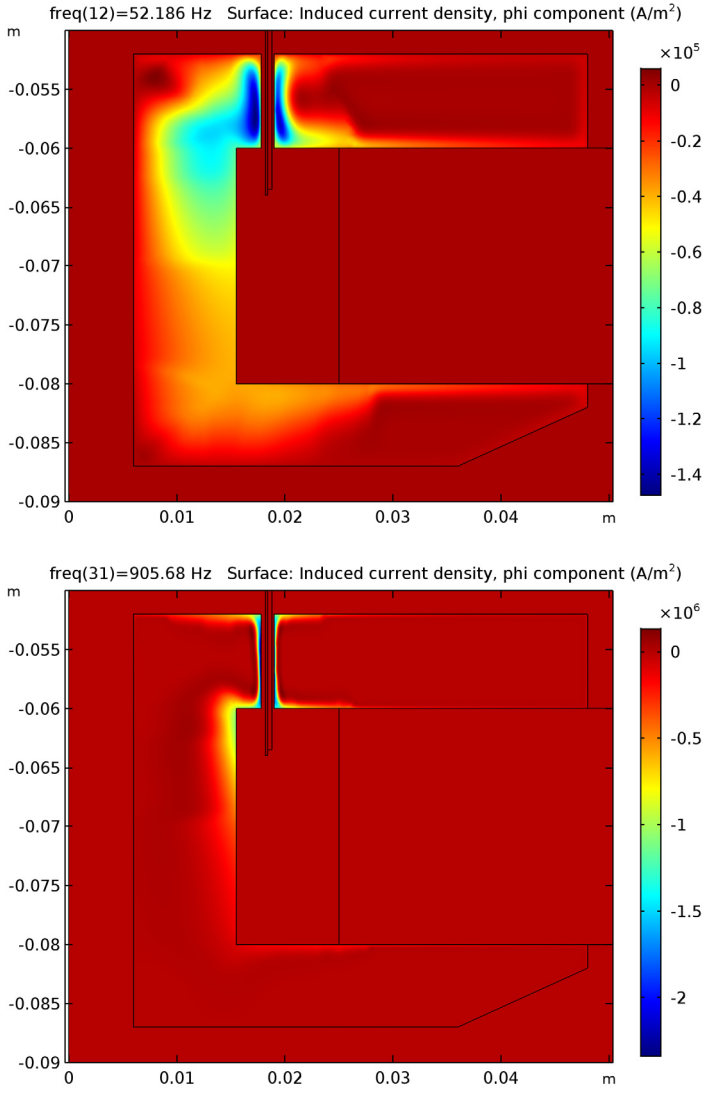


Figure 5: Induced currents in the pole piece and top plate at 52 Hz (top) and 905 Hz (bottom).

At the higher frequency, the skin effect brings the currents closer to the surfaces. This causes the inductance as well as the resistive part of the impedance to change with the

frequency. Figure 6 shows a plot of the blocked coil inductance versus frequency.

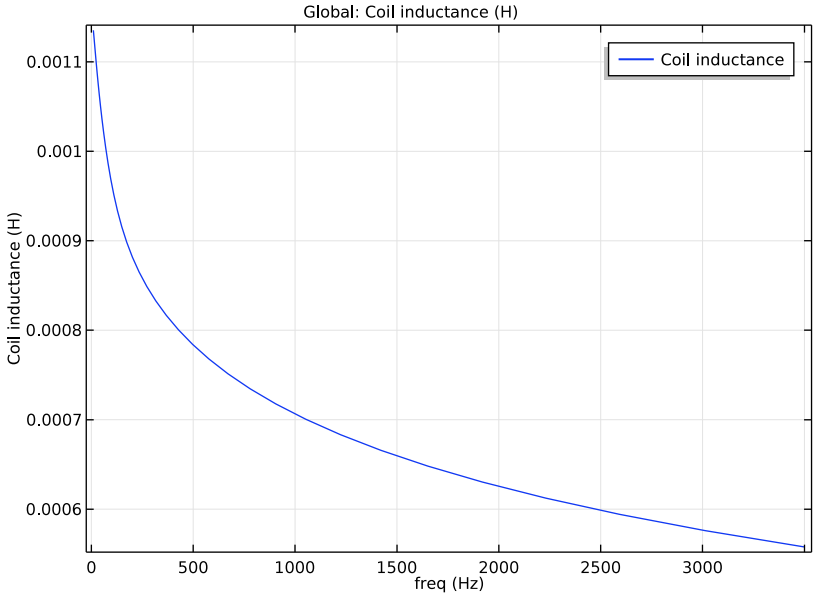


Figure 6: The inductance of the blocked coil as a function of frequency.

From the acoustic-structure interaction analysis, Figure 7 shows the sound pressure level distribution at 3500 Hz. A minimum has formed in a direction about 45 degrees above the baffle. At lower frequencies, the sound pressure level is rather evenly distributed but peaks in the on-axis direction.

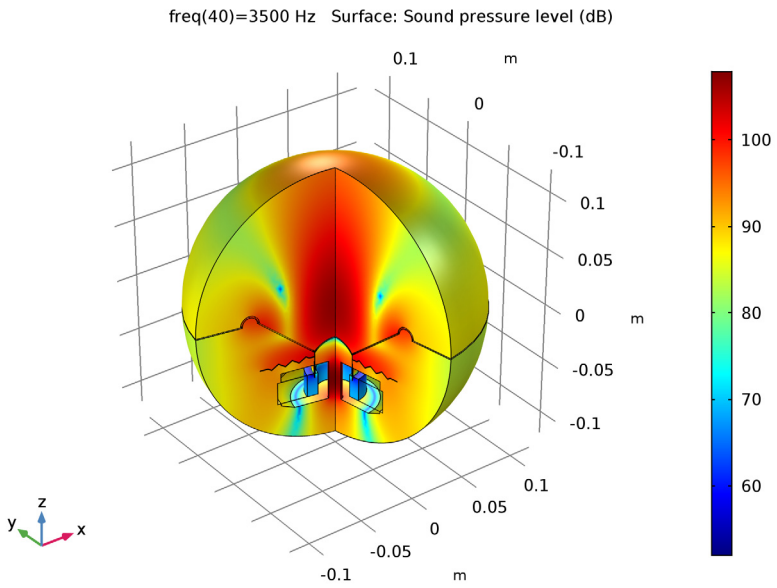


Figure 7: Sound pressure level distribution in dB at 3500 Hz.

Figure 8 presents the loudspeaker's sensitivity depicted both in 1/3 octave bands and as a continuous curve. The plot is realized using the specialized *Octave Band* plot available in the Acoustics Module. The preferred operating range is where the response is rather even, that is, roughly in the range 100 Hz–1500 Hz. A vented enclosure can extend the range to lower frequencies, as shown in the model Loudspeaker Driver in a Vented Enclosure.

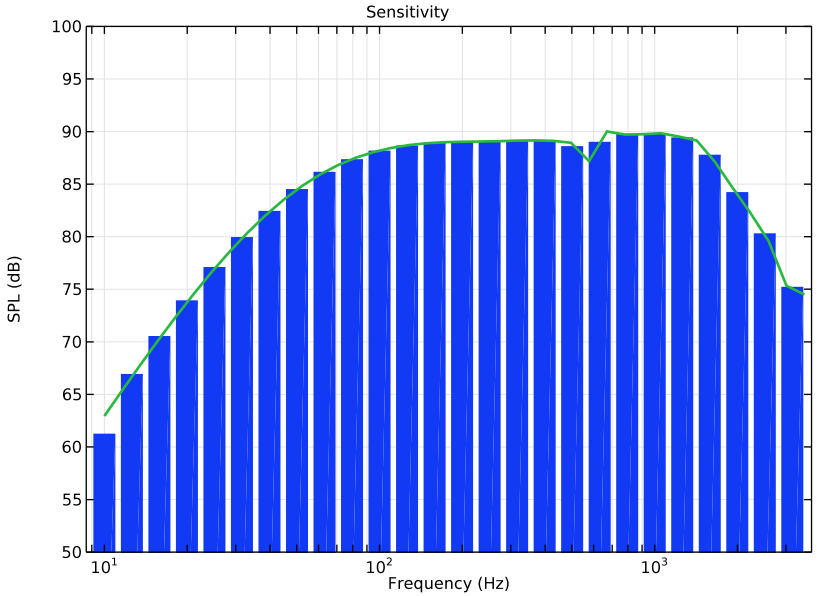


Figure 8: Loudspeaker sensitivity, measured as the on-axis sound pressure level (dB) at a distance of 1 m from the unit. The pressure is evaluated using an input signal of 4 V, or 2.83 V RMS, which corresponds to a power of 1 W at an 8 Ω nominal impedance. Note the logarithmic frequency scale.

The total electric impedance, defined as $Z = V_0/I$, appears in Figure 9 (absolute, real, and imaginary parts are plotted). The features of this plot are very characteristic of loudspeaker drivers. The peak at approximately 40 Hz coincides with the mechanical resonance; at this frequency the reactive part of the impedance switches sign from inductive to capacitive. In most of the operational range the impedance is largely resistive. Between 100 Hz and 1 kHz it varies only between 6.1 Ω and 8.3 Ω. These are typical values for speakers with a nominal impedance of 8 Ω, as the nominal impedance is usually taken to represent a mean value over the usable frequency range, which for this driver extends between somewhat below 100 Hz and above 1 kHz. At frequencies higher than 1 kHz, the impedance continues to increase as the inductance of the voice coil starts playing a more important part.

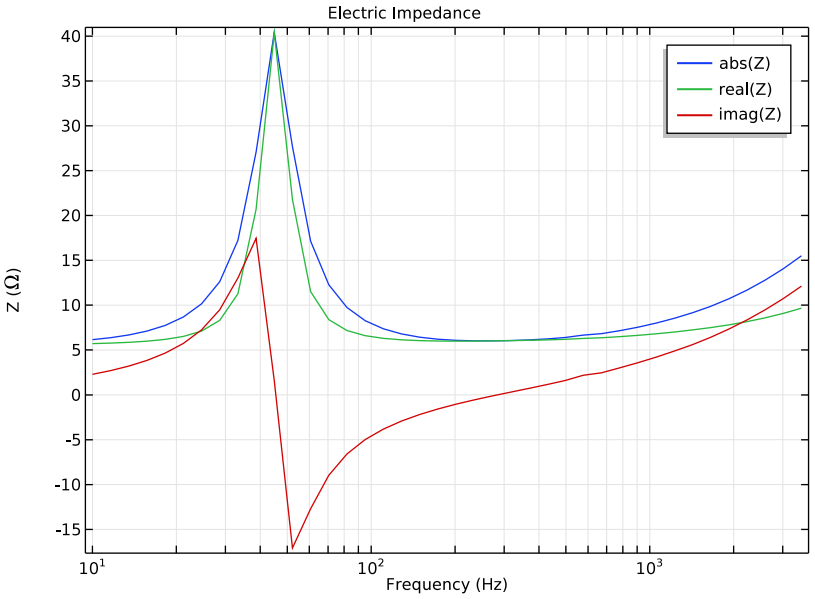


Figure 9: Electric impedance (Ω) of the loudspeaker as a function of frequency (Hz).

The deformation of the speaker cone, surround, spiders, and voice coil is depicted in Figure 10. The figure is evaluated at 3500 Hz and shows, for example, that the motion of the spider is out of phase with the motion of the cone. Typically, this kind of plot can be used to detect and pinpoint where breakups will occur. In the present case only rotationally symmetric breakups can be modeled (the speaker is analyzed in 3D in the *Loudspeaker Enclosure* model). Using an eigenfrequency study can pinpoint at which frequencies the breakup occurs.

Figure 11, finally, shows a directivity plot of the spatial speaker response. This is created using the dedicated Directivity plot available with the Acoustics Module. The plot shows a contour representation of the spatial response (measured on a half sphere in front of the speaker) versus the frequency. Directivity plots help analyze when side lobes occur and how they fall off. Several options, for example for the normalization and for switching the axes, are included. The plot in Figure 11 is normalized with respect to the level at 0^{o}.

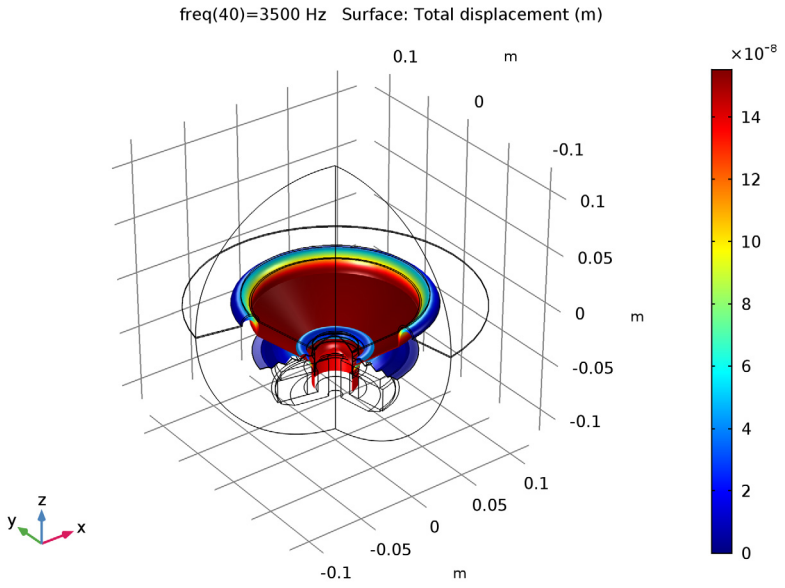


Figure 10: Deformation of the speaker cone, surround, spiders and voice-coil.

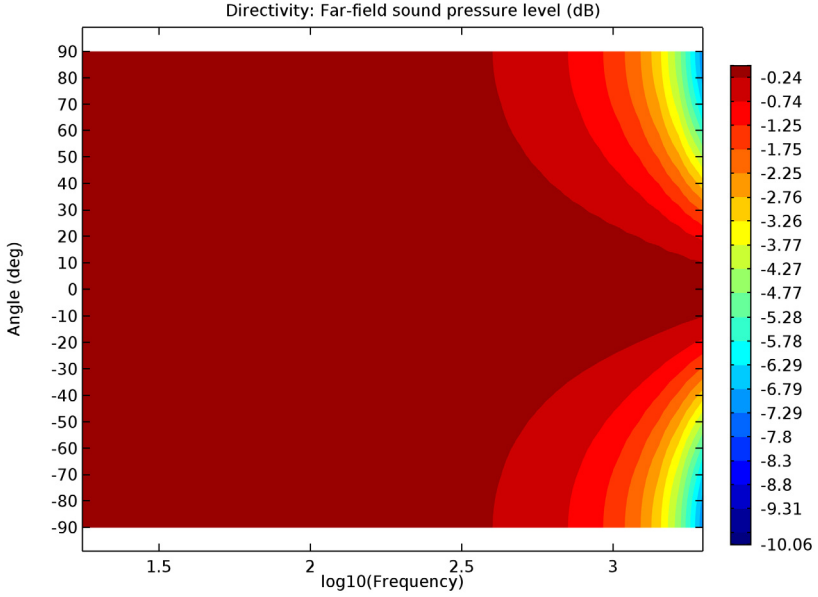


Figure 11: Directivity plot of the spatial speaker response.

Notes About the COMSOL Implementation

The step-by-step instructions take you through the following steps:

- Import the geometry and enter model parameters.
- Apply material settings.
- Set up the physics.
- Create a study computing first the static magnetic field from the permanent magnet, then the time-harmonic fields created by the voice coil over a range of frequencies.
- (Optional) Run the study to extract the force factor, BL, and the blocked coil impedance, Z_b.
- Copy and extend the study to include the acoustic-structure interaction.
- Solve to compute the sound pressure level and the total electric impedance of the driver over the same frequency range.

The iron used in the pole piece and top plate is a nonlinear magnetic material, with interpolation data describing the relationship between the B and H fields. Among other output, the static solution provides the local permeability, as shown in Figure 4.

The studies in this model have a Stationary study step followed by a Frequency Domain, Perturbation step. This automatically makes the stationary solution the linearization point for the subsequent frequency domain solution. This means that the Magnetic Fields interface derives and uses a differential permeability inherited from the one computed by the stationary study. For the frequency domain assumption to be strictly valid, the applied AC voltage must be so small that the resulting current creates a magnetic field which does not significantly alter this permeability. Even though this is not quite the situation here, linearizing around a local biased permeability should still be a better approximation than assuming a constant permeability. The most accurate way to compute the impedance would be in a fully transient analysis, which is outside the scope of this model.

The Acoustic-Structure Interaction multiphysics interface sets up the pressure acoustics and the solid mechanics interfaces together with the Acoustic-Structure Boundary multiphysics coupling. The multiphysics coupling (under the **Multiphysics** node) automatically provides and assigns the boundary conditions for the two-way acoustic-structural coupling between the air and the structures. The acoustic-structure interaction is solved for only in the Frequency Domain, Perturbation step.

The coil is driven with a voltage set to `linper(V0)`. The `linper()` operator ensures that the driving voltage `V0` is applied only in the Frequency Domain, Perturbation study step.

In most loudspeaker specifications, the suspension is characterized by a mechanical compliance C_s and resistance R_s. In order to keep the resistance constant over a range of frequencies, the material needs to have a damping factor that increases linearly with the frequency or, equivalently, Rayleigh damping with $\alpha_{\delta M} = 0$ and a constant $\beta_{dK} = \eta_0/\omega_0$, where η_0 is the loss factor measured at the angular frequency ω_0. In this model, the frequency where the loss factor is measured is chosen to be near the lowest mechanical resonance of the driver.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/loudspeaker_driver`

Note: This application also requires the file `Acoustics_Module/Electroacoustic_Transducers/loudspeaker_driver_materials` as it contains the material definitions for Materials.

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 **AC/DC>Magnetic Fields (mf)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Acoustics>Acoustic-Structure Interaction>Acoustic-Solid Interaction, Frequency Domain**.
5. Click **Add**.
6. Click **Study**.

   The Model Wizard lets you select the first one of the study steps you plan to use in the model. Select a stationary study used for solving the static magnetic fields.

7 In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Stationary**.

8 Click **Done**.

GEOMETRY 1

When working with your own modeling project of an acoustic driver, you will typically either draw the geometry in COMSOL Multiphysics, or import a CAD file of the driver itself and add the surrounding air and PML domains. Here, the entire geometry is imported as a sequence from the geometry file. The instructions to the geometry can be found in the appendix at the end of this document.

1 On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `loudspeaker_driver_geom_sequence.mph`.

3 On the **Geometry** toolbar, click **Build All**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

GLOBAL DEFINITIONS

Parameters

Enter the model parameters or load them from the file `loudspeaker_driver_parameters.txt`. Here, as well as in all following sections, the Description field helps you keep track of what you are doing, but is completely optional.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
B0	0.4[T]	0.4 T	Remanent flux density in permanent magnet
N0	100	100	Number of turns in coil
V0	4[V]	4 V	Peak driving voltage
f0	40[Hz]	40 Hz	Frequency at which loss factor is given
omega0	2*pi*f0	251.33 Hz	Angular frequency at which loss factor is given

The loss factor frequency definition will be used when setting up the structural properties.

DEFINITIONS

Create a selection for the coil, this will simplify setting up the physics.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Coil` in the **Label** text field.

3 Select Domain 8 only.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

Use the Perfectly Matched Layers (PMLs) to model a nonreflecting condition-like behavior and avoid unphysical reflections (spurious reflections) where the sound leaves the model. This feature is also set up under **Definitions**. This makes them available for any physics interface that needs them.

2 Select Domains 1 and 5 only.

MATERIALS

While the material properties used in this model are partly made up, they resemble those used in a real driver. The diaphragm and dust cap both consist of a HexaCone®-like material; a light and very stiff composite. The apex has properties representative of glass fiber materials. The spider, acting as a spring, is made of a phenolic cloth with a much lower stiffness. The material used in the coil is taken to be lighter than copper, as the wire is insulated and does not completely fill the coil domain. The surround, finally, is a light resistive foam.

Except for air and soft Iron, the materials you will use all come from a material library created especially for this model (to be loaded from the file `loudspeaker_driver_materials.mph`). You may notice that some of the materials will report missing properties. For example, the composite does not include any electromagnetic properties. This is fine, as you will not model the magnetic fields in the domains where the composite is used.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

First, add air which will be present everywhere in your geometry. Next, switch to using nonlinear Iron in the pole piece and top plate.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **AC/DC>Soft Iron (with losses)**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Soft Iron (with losses) (mat2)
Select Domains 6 and 11 only.

Add Material

1 Right-click **Component 1 (comp1)>Materials>Soft Iron (with losses) (mat2)** and choose **Browse Materials**.

In the ribbon make sure to select the **Materials** tab and then click the **Browse Materials** icon.

MATERIAL BROWSER

1 In the **Material Browser** window, The **Import Material Library** functionality is activated by clicking the small icon at the bottom right, below the Material Browser tree.

2 Click **Import Material Library**.

3 Browse to the model's Application Libraries folder and double-click the file `loudspeaker_driver_materials.mph`.

4 click **Done**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **loudspeaker driver materials>Composite**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Composite (mat3)
Select Domains 3 and 10 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **loudspeaker driver materials>Cloth**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Cloth (mat4)
Select Domain 9 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **loudspeaker driver materials>Foam**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Foam (mat5)
Select Domain 13 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **loudspeaker driver materials>Coil**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Coil (mat6)
Select Domain 8 only.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **loudspeaker driver materials>Glass Fiber**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Glass Fiber (mat7)

Select Domain 7 only.

MAGNETIC FIELDS (MF)

The Magnetic Fields equation needs to be solved in and around the magnetic motor. To reduce simulation time, make this physics interface active only where it is needed. You can remove all domains where you expect the magnetic field to be negligible.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

2 Select Domains 2, 6–9, 11, and 12 only.

Ampère's Law is per default solved in all domains where the physics interface is active. Add a second instance of it to apply to the permanent magnet, where you need a different constitutive relation.

Ampère's Law 2

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domain 12 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **Remanent flux density**.

5 Specify the $\mathbf{B}_r$ vector as

0	r
0	phi
B0	z

This setting gives a static remanent flux density equal to 0.4 T in the z direction. This will create a static magnetic field distribution in the model, providing the linearization point for the frequency domain study.

Ampère's Law 3

1 On the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.

2 Select Domains 6 and 11 only.

3 In the **Settings** window for **Ampère's Law**, locate the **Magnetic Field** section.

4 From the **Constitutive relation** list, choose **BH curve**.

The HB curve is provided by the soft iron material.

Coil 1

1 On the **Physics** toolbar, click **Domains** and choose **Coil**.

2 In the **Settings** window for **Coil**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Coil**.

4 Locate the **Coil** section. From the **Conductor model** list, choose **Homogenized multi-turn**.

5 Locate the **Homogenized Multi-Turn Conductor** section. In the N text field, type `N0`.

6 In the a_{coil} text field, type `3.5e-8[m^2]`.

With `N0 = 100` turns, the total cross-sectional area covered by the wires will be 3.5e-6 m^2. The area of the coil domain is 6e-6 m^2, making the fill factor approximately 60 %.

7 Locate the **Coil** section. From the **Coil excitation** list, choose **Voltage**.

8 In the V_{coil} text field, type `linper(V0)`.

This is the driving voltage. Because the `linper()` operator is used it will kick in only in the **Frequency Domain, Perturbation** study.

Now, add the **Velocity (Lorent Term)** domain condition. This feature sets up the induced voltage caused by the motion of the coil in the static magnetic field.

Velocity (Lorentz Term) 1

1 On the **Physics** toolbar, click **Domains** and choose **Velocity (Lorentz Term)**.

2 In the **Settings** window for **Velocity (Lorentz Term)**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Coil**.

4 Locate the **Velocity (Lorentz Term)** section. From the **v** list, choose **Velocity (solid/lemm1)**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Select the air domains above and under the speaker as well as the PML regions.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 Select Domains 1, 2, 4, and 5 only.

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

The far-field calculation requires a source boundary encompassing all local sound sources, and with a symmetry plane to account for the infinite baffle. After computing the solution, you can evaluate the pressure in any point (r,z) outside the domain by entering `pfar(r,z)`.

2 Select Boundary 70 only.

3 In the **Settings** window for **Far-Field Calculation**, locate the **Far-Field Calculation** section.

4 From the **Type of integral** list, choose **Full integral**.

5 Select the **Symmetry in the z=0 plane** check box.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domains 3, 7–10, and 13 only.

With the above selection, you leave out the magnet, pole piece, and top plate. You will consider these domains as perfectly rigid, by using the default sound hard wall condition on their surfaces.

Add damping to some of the solid material.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 1

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 3, 7, and 10 only.

5 Locate the **Damping Settings** section. From the **Damping type** list, choose **Isotropic loss factor**.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 2

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 9 only.

5 Locate the **Damping Settings** section. In the β_{dK} text field, type `0.14/omega0`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 3

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 13 only.

5 Locate the **Damping Settings** section. In the β_{dK} text field, type `0.46/omega0`.

Body Load 1

1 On the **Physics** toolbar, click **Domains** and choose **Body Load**.

2 In the **Settings** window for **Body Load**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Coil**.

4 Locate the **Force** section. From the $\mathbf{F}_V$ list, choose **Lorentz force contribution (mf)**.

This sets up the driving Lorentz force on the coil (it represents the product of the time-harmonic current and the static magnetic field in which it is traveling). For details, see Notes About the COMSOL Implementation.

The spider and the surround are attached to the case.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundaries 58 and 62 only.

Now is a good time to inspect **Acoustic-Structure Boundary 1** multiphysics coupling under the **Multiphysics** node. When using a predefined multiphysics interface the coupling is automatically applied to all acoustic-solid boundaries.

MESH 1

The mesh used in computing the impedance needs to resolve the induced eddy currents in the pole piece and the top plate. For the results to be accurate, the skin depth needs to be resolved by at least 1, preferably 2 quadratic elements.

With a conductivity of 1.12e7 S/m and a peak relative permeability of 1200, the skin depth in the iron at the maximum frequency of 3.5 kHz does not go below 0.07 mm. In practice, most of the induced currents will run in regions of the pole piece where the biased relative permeability is much less than 1200, which makes the skin depth greater. In this

model, it is therefore sufficient to use a mesh size of 0.2 mm along the iron surfaces that are closest to the voice coil.

For the acoustic-structure interaction, the air domain and the thin moving structures also need to be well resolved. The Extra fine setting gives a maximum element size of 6 mm which is smaller than the suggested 6 elements per wavelength (this only requires a 16 mm mesh size). The PML is preferably meshed with mapped elements, use 8 elements for the default polynomial scaling.

Mapped 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.
2. In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 1 and 5 only.

Distribution 1

1. Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.
2. Select Boundaries 1 and 5 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `8`.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Extra fine**.

Size 1

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.
2. Right-click **Free Triangular 1** and choose **Size**.
3. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
4. From the **Geometric entity level** list, choose **Domain**.
5. Select Domains 6 and 11 only.
6. Locate the **Element Size** section. Click the **Custom** button.
7. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
8. In the associated text field, type `1[mm]`.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 6 and 11 only.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type `3`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `0.2[mm]`.

7 Click **Build All**.

STUDY 1

Your Study node already contains the Stationary study that you picked from the Model Wizard. Disable the Pressure Acoustics and Solid Mechanics interfaces.

Add a Frequency-Domain, Perturbation study.

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Pressure Acoustics, Frequency Domain (acpr)** and **Solid Mechanics (solid)**.

Step 2: Frequency Domain Perturbation

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain Perturbation**.

2 In the **Settings** window for **Frequency Domain Perturbation**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `10^(range(1,(log10(3500)-1)/39, log10(3500)))`.

This generates 40 exponentially distributed frequencies between 10 Hz and 3.5 kHz.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Pressure Acoustics, Frequency Domain (acpr)** and **Solid Mechanics (solid)**.

NOTE: The first solution of this model is only of the electromagnetic part of the problem, with the coil assumed to be fixed in order to extract the BL factor and the blocked coil impedance. If you would like to skip ahead to the solution of the full electroacoustic problem, you can do so by proceeding from here to the instructions starting at the section **Component 1 (comp1)**, further below. Otherwise, continue by computing the solution.

5 On the **Study** toolbar, click **Compute**.

RESULTS

First, create a 2D plot to view the magnetic field distribution from the permanent magnet.

2D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Static Magnetic Field` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution Store 1 (sol2)**.

Each step in a study creates its own data set. The numbering of the data sets begins from the last step. Hence in this model Solution 2 contains the stationary solution and Solution 1 the frequency domain perturbation.

Surface 1

1 Right-click **Static Magnetic Field** 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>Magnetic Fields>Magnetic> mf.normH - Magnetic field norm**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

4 On the **Static Magnetic Field** toolbar, click **Plot**.

You are now looking at the magnetic field norm created by the permanent magnet. Note that it has a distinct maximum in the gap where the voice coil is moving. If you zoom in a little, the plot should look like Figure 3.

Another interesting result is the permeability distribution in the iron. Begin by duplicating the existing plot.

Static Magnetic Field 1

1 In the **Model Builder** window, under **Results** right-click **Static Magnetic Field** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Effective Permeability` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Results>Effective Permeability** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `mf.normB/(mu0_const*mf.normH)`.

4 On the **Effective Permeability** toolbar, click **Plot**.

This is the effective relative permeability at the linearization point, as given by the ratio of the flux density and the field multiplied by the permeability of vacuum. The plot should look like Figure 4.

Derived Values

Next, evaluate the BL force factor.

Surface Average 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Average>Surface Average**.

2 In the **Settings** window for **Surface Average**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution Store 1 (sol2)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Coil**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`-mf.Br*N0*2*pi*r`	`Wb/m`	

6 Locate the **Integration Settings** section. Clear the **Compute volume integral** check box.

7 Click **Evaluate**.

The BL factor evaluates to 7.53 Wb/m.

Next, study the induced current density at a few different frequencies.

2D Plot Group 3

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Induced Current Density` in the **Label** text field.

Surface 1

1 Right-click **Induced Current Density** 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>Magnetic Fields> Currents and charge>Induced current density (spatial frame)>mf.Jiphi - Induced current density, phi component**.

The **Compute differential** check box, if turned on, wraps the expression in a `lindev()` operator. For more information, search for lindev in the COMSOL Multiphysics Documentation.

3 On the **Induced Current Density** toolbar, click **Plot**.

At 3500 Hz, the induced currents are highly localized to the surfaces of the top plate and pole piece.

Induced Current Density

1 In the **Model Builder** window, under **Results** click **Induced Current Density**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **10**.

4 On the **Induced Current Density** toolbar, click **Plot**.

At 10 Hz, the induced currents distribute throughout the material. The distribution depends on the vicinity to the voice coil, but also very much on the local effective permeability from the static study. To reproduce Figure 5, try two frequencies in between.

5 From the **Parameter value (freq (Hz))** list, choose **905.68**.

Before proceeding to the full electroacoustic analysis, plot the blocked coil inductance as a function of the frequency.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Blocked Coil Inductance` in the **Label** text field.

Global 1

1 Right-click **Blocked Coil Inductance** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Magnetic Fields> Coil parameters>mf.LCoil_1 - Coil inductance**.

3 On the **Blocked Coil Inductance** toolbar, click **Plot**.

The result should look like Figure 6.

Having extracted the force factor and the voice coil inductance, you now have most of the electromagnetic data required for a separate model of the acoustic parts of a boxed loudspeaker driver. See the `Loudspeaker Driver in a Vented Enclosure` model also in the Application Library. What remains is the real part of the blocked coil impedance. This is available as `mf.RCoil_1`, the Coil Resistance. If you would like to extract this too, you can set it up in a Global plot just like the inductance, and export the results as described in the following steps.

4 Right-click **Results>Blocked Coil Inductance>Global 1** and choose **Add Plot Data to Export**.

Plot 1

If you want to export the data, you can now enter a file name and click the Export button.

COMPONENT 1 (COMP1)

It is now time to compute and evaluate the solution of the entire model, including the acoustic-structure interaction. In order to this, you will set up a new study. Although you could technically reuse the static magnetic fields solution from the first study, it can be handy to include this step in the new study too. This allows you to make changes anywhere in the model, including such that affect the static magnetic fields, and run only the new study again to get correctly updated results.

ADD STUDY

1 On the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

Select an empty study so that you can copy and modify the study steps from your previous study.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

1 In the **Settings** window for **Study**, type `Magnetic Fields Study` in the **Label** text field.

2 In the **Model Builder** window, expand the **Magnetic Fields Study** node.

3 Use Ctrl-click to select both the nodes **Step 1: Stationary** and **Step 2: Frequency Domain Perturbation**, then right-click and choose **Copy**.

STUDY 2

1 In the **Model Builder** window, right-click **Study 2** and choose **Paste Multiple Items**.

2 In the **Settings** window for **Study**, type `Complete Study` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 2: Frequency Domain Perturbation

Make the Frequency Domain, Perturbed study solve for the acoustics and the solid physics too.

1 In the **Model Builder** window, under **Complete Study** click **Step 2: Frequency Domain Perturbation**.

2 In the **Settings** window for **Frequency Domain Perturbation**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Pressure Acoustics, Frequency Domain (acpr)	√	physics
Solid Mechanics (solid)	√	physics

4 On the **Home** toolbar, click **Compute**.

RESULTS

You will now have received two more data sets: Solution 4 containing the same static magnetic fields as Solution 2, and Solution 3 with the frequency domain electromagnetic and acoustic-structure interaction results. In order to get a good overview of the latter, begin by adding a selection of all domains except the PMLs. You will then create a Revolution data set in order to enable a 3D view of the model.

Complete Study/Solution 3 (sol3)

In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Complete Study/Solution 3 (sol3)**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **All domains**.

5 Select Domains 2–4 and 6–13 only.

Revolution 2D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Complete Study/Solution 3 (sol3)**.

4 Click to expand the **Revolution layers** section. Locate the **Revolution Layers** section. In the **Start angle** text field, type `-80`.

5 In the **Revolution angle** text field, type `250`.

Create a 3D plot of the instantaneous pressure distribution at zero phase in and around the speaker.

3D Plot Group 5

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Instantaneous Pressure` in the **Label** text field.

Surface 1

1 Right-click **Instantaneous Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_t`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

5 Select the **Symmetrize color range** check box.

Contour 1

1 In the **Model Builder** window, under **Results** right-click **Instantaneous Pressure** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_t`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

6 Clear the **Color legend** check box.

Next, reproduce Figure 7 with a plot of the local sound pressure level.

3D Plot Group 6

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level` in the **Label** text field.

Surface 1

1 Right-click **Sound Pressure Level** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.Lp`.

4 On the **Sound Pressure Level** toolbar, click **Plot**.

Create a 1D plot of the sensitivity versus the frequency, as in Figure 8.

1D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Sensitivity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Complete Study/Solution 3 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Sensitivity`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Frequency (Hz)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `SPL (dB)`.

Octave Band 1

1 On the **Sensitivity** toolbar, click **More Plots** and choose **Octave Band**.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

3 From the **Geometric entity level** list, choose **Global**.

Use the Octave Band Plot to depict the sensitivity. Evaluate the pressure 1 m in front using `pfar()` operator. The reference pressure is the default for an SPL evaluation. Plot the sensitivity both as a continuous curve and in 1/3 octave bands.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `pfar(0,1)`.

5 On the **Sensitivity** toolbar, click **Plot**.

6 Locate the **Plot** section. From the **Style** list, choose **1/3 octave bands**.

Octave Band 2

1 In the **Model Builder** window, right-click **Octave Band 1** and choose **Duplicate**.

2 In the **Settings** window for **Octave Band**, locate the **Plot** section.

3 From the **Style** list, choose **Continuous**.

4 On the **Sensitivity** toolbar, click **Plot**.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type 2.

Sensitivity

1 In the **Model Builder** window, under **Results** click **Sensitivity**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **y minimum** text field, type 50.

5 In the **y maximum** text field, type 100.

6 On the **Sensitivity** toolbar, click **Plot**.

Finally, plot the total electric impedance versus the frequency.

1D Plot Group 8

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Total Electric Impedance` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Complete Study/Solution 3 (sol3)**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Electric Impedance`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Frequency (Hz)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Z (\Omega)`.

Global 1

1 Right-click **Total Electric Impedance** and choose **Global**.

The automatic computation of the coil impedance uses the entire applied voltage. Because this now includes a contribution from the motion of the coil, you need to define it manually as only the driving voltage V0 divided by the time-harmonic current through the coil. The results are depicted in Figure 9.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(V0/mf.ICoil_1)`		`abs(Z)`

Expression	Unit	Description
real(V0/mf.ICoil_1)		real(Z)
imag(V0/mf.ICoil_1)		imag(Z)

4 On the **Total Electric Impedance** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Next create a plot of the deformation of the loudspeaker. This can in general be used to visualize breakups in the cone and surround.

3D Plot Group 9

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Deformation` in the **Label** text field.

Surface 1

1 Right-click **Deformation** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.disp`.

Deformation 1

1 Right-click **Results>Deformation>Surface 1** and choose **Deformation**.

2 On the **Deformation** toolbar, click **Plot**.

The plot should look like the one in Figure 10.

Finally, visualize the directivity of the loudspeaker using the built in Directivity plot. This is an important plot used to visualize and analyze the spatial response of the speaker as function of the frequency.

First, create a copy of the relevant data set and switch to spatial coordinates. This data set is automatically created when the Generate default plot is not de-selected. It is necessary for the Directivity plot and other plots using the far-field feature.

Complete Study/Solution 3 (5) (sol3)

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Complete Study/Solution 3 (sol3)** and choose **Duplicate**.

2 In the **Settings** window for **Solution**, locate the **Solution** section.

3 From the **Frame** list, choose **Spatial (r, phi, z)**.

2D Plot Group 10

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Directivity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

Directivity 1

1 On the **Directivity** toolbar, click **More Plots** and choose **Directivity**.

Select the data set you just created.

2 In the **Settings** window for **Directivity**, locate the **Data** section.

3 From the **Data set** list, choose **Complete Study/Solution 3 (5) (sol3)**.

Per default the reference direction is along the z-axis. This defines what 0 deg corresponds to. Change the angles to correspond to everything in front of the speaker, that is, from -90 to 90 deg.

4 Locate the **Evaluation** section. Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.

5 In the ϕ **start** text field, type `-90`.

6 In the ϕ **range** text field, type `180`.

7 On the **Directivity** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

The image should look like the once in Figure 11. If you are more familiar with having the frequency on the y-axis, you can just change that under the **Coloring and Style** tab and change the **Layout** option.

Appendix: Geometry Sequence Instructions

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>2D Axisymmetric**.

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 **m**.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `150[mm]`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	30[mm]

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type 8[mm].

4 In the **Sector angle** text field, type 180.

5 Locate the **Position** section. In the **r** text field, type 74[mm].

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	1.5[mm]

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 On the object **c2**, select Boundaries 2–4 only.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 70[mm].

4 In the **Height** text field, type 1[mm].

5 Locate the **Position** section. In the **r** text field, type 80.5[mm].

6 In the **z** text field, type -1[mm].

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **r1** only.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `42[mm]`.

4 In the **Height** text field, type `35[mm]`.

5 Locate the **Position** section. In the **r** text field, type `6[mm]`.

6 In the **z** text field, type `-87[mm]`.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `35.5[mm]`.

4 In the **Height** text field, type `20[mm]`.

5 Locate the **Position** section. In the **r** text field, type `15.5[mm]`.

6 In the **z** text field, type `-80[mm]`.

Rectangle 4 (r4)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1.2[mm]`.

4 In the **Height** text field, type `8[mm]`.

5 Locate the **Position** section. In the **r** text field, type `17.8[mm]`.

6 In the **z** text field, type `-60[mm]`.

Rectangle 5 (r5)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Locate the **Size and Shape** section. In the **Width** text field, type `26[mm]`.

5 In the **Height** text field, type `20[mm]`.

6 Locate the **Position** section. In the **r** text field, type `25[mm]`.

7 In the **z** text field, type `-80[mm]`.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **r** to `48[mm]`.

5 In row **1**, set **z** to `-82[mm]`.

6 In row **2**, set **r** to `36[mm]`.

7 In row **2**, set **z** to `-87[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **r** to `48[mm]`.

Difference 2 (dif2)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 In the **Settings** window for **Difference**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Select the object **r2** only.

5 Locate the **Difference** section. Find the **Objects to subtract** subsection. Select the **Active** toggle button.

6 Select the objects **r4**, **r3**, and **b1** only.

Rectangle 6 (r6)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Locate the **Size and Shape** section. In the **Width** text field, type `0.2[mm]`.

5 In the **Height** text field, type `25[mm]`.

6 Locate the **Position** section. In the **r** text field, type `18.2[mm]`.

7 In the **z** text field, type `-64[mm]`.

Rectangle 7 (r7)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Locate the **Size and Shape** section. In the **Width** text field, type `0.4[mm]`.

5 In the **Height** text field, type `15[mm]`.

6 Locate the **Position** section. In the **r** text field, type `18.4[mm]`.

7 In the **z** text field, type `-63.5[mm]`.

Rectangle 8 (r8)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `4.6[mm]`.

4 In the **Height** text field, type `0.4[mm]`.

5 Locate the **Position** section. In the **r** text field, type `18.4[mm]`.

6 In the **z** text field, type `-44.5[mm]`.

Rectangle 9 (r9)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `7[mm]`.

4 In the **Height** text field, type `0.4[mm]`.

5 Locate the **Position** section. In the **r** text field, type `59[mm]`.

6 In the **z** text field, type `-44.5[mm]`.

Bézier Polygon 2 (b2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **r** to `23[mm]`.

5 In row **1**, set **z** to `-44.1[mm]`.

6 In row **2**, set **r** to `26[mm]`.

7 In row **2**, set **z** to `-42.1[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **r** to `32[mm]`.

10 In row **2**, set **z** to `-46.1[mm]`.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **r** to `38[mm]`.

13 In row **2**, set **z** to `-42.1[mm]`.

14 Find the **Added segments** subsection. Click **Add Linear**.

15 Find the **Control points** subsection. In row **2**, set **r** to `34[mm]`.

16 In row **2**, set **z** to `-46.1[mm]`.

17 In row **2**, set **r** to `44[mm]`.

18 Find the **Added segments** subsection. Click **Add Linear**.

19 Find the **Control points** subsection. In row **2**, set **r** to `50[mm]`.

20 In row **2**, set **z** to `-42.1[mm]`.

21 Find the **Added segments** subsection. Click **Add Linear**.

22 Find the **Control points** subsection. In row **2**, set **z** to `-46.1[mm]`.

23 In row **2**, set **r** to `56[mm]`.

24 Find the **Added segments** subsection. Click **Add Linear**.

25 Find the **Control points** subsection. In row **2**, set **r** to `59[mm]`.

26 In row **2**, set **z** to `-44.1[mm]`.

27 Find the **Added segments** subsection. Click **Add Linear**.

28 Find the **Control points** subsection. In row **2**, set **z** to `-44.5[mm]`.

29 Find the **Added segments** subsection. Click **Add Linear**.

30 Find the **Control points** subsection. In row **2**, set **r** to `56[mm]`.

31 In row **2**, set **z** to `-46.5[mm]`.

32 Find the **Added segments** subsection. Click **Add Linear**.

33 Find the **Control points** subsection. In row **2**, set **z** to `-42.5[mm]`.

34 In row **2**, set **r** to `50[mm]`.

35 Find the **Added segments** subsection. Click **Add Linear**.

36 Find the **Control points** subsection. In row **2**, set **r** to `44[mm]`.

37 In row **2**, set **z** to `-46.5[mm]`.

38 Find the **Added segments** subsection. Click **Add Linear**.

39 Find the **Control points** subsection. In row **2**, set **r** to `38[mm]`.

40 In row **2**, set **z** to `-42.5[mm]`.

41 Find the **Added segments** subsection. Click **Add Linear**.

42 Find the **Control points** subsection. In row **2**, set **r** to `32[mm]`.

43 In row **2**, set **z** to `-46.5[mm]`.

44 Find the **Added segments** subsection. Click **Add Linear**.

45 Find the **Control points** subsection. In row **2**, set **r** to `26[mm]`.

46 In row **2**, set **z** to `-42.5[mm]`.

47 Find the **Added segments** subsection. Click **Add Linear**.

48 Find the **Control points** subsection. In row **2**, set **r** to `20[mm]`.

49 In row **2**, set **z** to `-46.5[mm]`.

50 In row **2**, set **r** to `23[mm]`.

51 In row **2**, set **z** to `-44.5[mm]`.

52 Find the **Added segments** subsection. Click **Add Linear**.

53 Find the **Control points** subsection. In row **2**, set **z** to `-44.1[mm]`.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Select the objects **r8**, **r9**, and **b2** only.

5 Locate the **Union** section. Clear the **Keep interior boundaries** check box.

Bézier Polygon 3 (b3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Selections of Resulting Entities** section.

3 Click **New**.

4 In the **New Cumulative Selection** dialog box, type `Composite` in the **Name** text field.

5 Click **OK**.

6 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

7 Find the **Added segments** subsection. Click **Add Linear**.

8 Find the **Control points** subsection. In row **1**, set **r** to `18.4[mm]`.

9 In row **1**, set **z** to `-39[mm]`.

10 In row **2**, set **r** to `66[mm]`.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **r** to `67.5[mm]`.

13 Find the **Added segments** subsection. Click **Add Linear**.

14 Find the **Control points** subsection. In row **2**, set **r** to `18.4[mm]`.

15 In row **2**, set **z** to `-40.26[mm]`.

16 Find the **Added segments** subsection. Click **Add Linear**.

17 Find the **Control points** subsection. In row **2**, set **z** to `-39[mm]`.

Circle 3 (c3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `23.5848[mm]`.

4 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	1[mm]

5 Locate the **Position** section. In the **z** text field, type `-54[mm]`.

6 Locate the **Size and Shape** section. In the **Sector angle** text field, type `90`.

Delete Entities 2 (del2)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **c3**, select Boundaries 1 and 2 only.

Rectangle 10 (r10)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.2[mm]`.

4 In the **Height** text field, type `25[mm]`.

5 Locate the **Position** section. In the **r** text field, type `18.2[mm]`.

6 In the **z** text field, type `-64[mm]`.

Union 2 (uni2)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **del2** and **r10** only.

Delete Entities 3 (del3)

1 Right-click **Geometry 1** and choose **Delete Entities**.

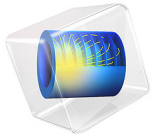
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **uni2**, select Domains 2–5 only.

5 Locate the **Selections of Resulting Entities** section. From the **Contribute to** list, choose **Composite**.

6 On the **Geometry** toolbar, click **Build All**.



Lumped Loudspeaker Driver

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of a moving-coil loudspeaker where a lumped parameter analogy represents the behavior of the electrical and mechanical speaker components. This lumped model is coupled to a 2D axisymmetric pressure acoustics model describing the surrounding air domain.

Electric circuit representations of transducers are well known and widely used. In the loudspeaker industry such models have been employed for a long time and with great success. The parameters that characterize the low-frequency performance of a loudspeaker are commonly known as the Thiele-Small or the small-signal parameters. It is common to use these parameters to design and simulate a desired speaker performance in terms of, for example, the on-axis speaker response, the electrical impedance, and the diaphragm velocity.

In the low-frequency regime the motion of the speaker cone can be approximated by the motion of a rigid piston in an infinite baffle. Analytical expressions exist for the sound field radiated from a piston. In the model at hand, the simple piston geometry is replaced by a more realistic speaker cone shape, and the sound field is solved using the finite element method. Of course this is still an approximation as the motion of the speaker cone is still assumed to be rigid; however, the spatial response of the speaker is more realistic. The methodology of lumping certain parts of a complex system is a general and powerful approach that can be applied to other systems. A first extension of the current model can be to only lump the electrical components and model the mechanical components fully.

The output from the model includes, among many things, the total electric impedance, the on-axis sound pressure level at a nominal driving voltage, and the mean speaker cone velocity. The results are compared with an analytical solution based on the flat piston approximation. Finally, it is also shown how to create a so-called directivity plot of the speaker.

Note: This application requires the Acoustics Module and the AC/DC Module.

Model Definition

A schematic representation of a moving coil loudspeaker is given in Figure 1. The figure shows a cross section of a loudspeaker. The speaker driver is placed in an infinite baffle with free space in front and on the back of the speaker. The speaker cone consists of the outer

suspension, the diaphragm, and a dust cap (not marked in the figure). The mechanical and electrical components of the speaker that are lumped are visualized inside the dotted box. On the electrical side it includes the voice-coil and magnetic system (permanent magnet and pole pieces), and on the mechanical side it includes the moving mass of the voice coil and speaker cone, the spring effect of the spider and outer suspension, as well as possible losses due to damping in these suspensions.

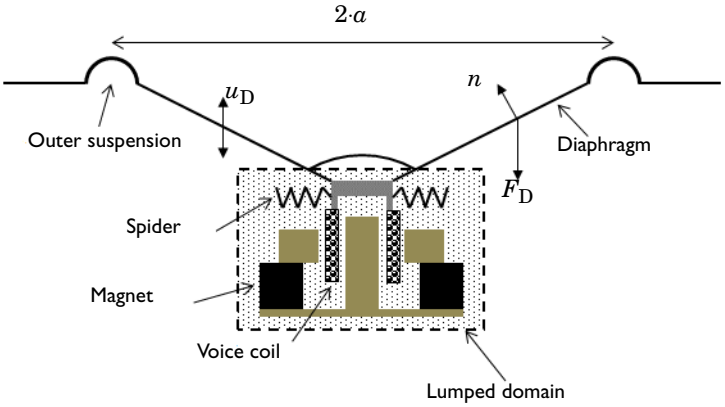


Figure 1: Schematic representation of a moving coil speaker unit.

ELECTROACOUSTIC ANALOGY

In an electroacoustic analogy, mechanical and acoustic physical properties such as force, velocity, pressure, and volume velocity are equated to voltages and currents in an analogous electrical circuit. In this model, the impedance analogy is used for the mechanical components only. This means that the current in the circuit represents the velocity of a moving part (SI unit: m/s) and the voltage represents a force (SI unit: N). In contrast, the acoustics is not represented by a circuit in this model. In general, however, in an acoustic analogous circuit, a voltage represents pressure (SI unit: Pa) and a current represents volume velocity (SI unit: m^3/s).

The passive elements in a circuit—that is, resistors, inductors, and capacitors—represent different processes, namely resistance, mass movement, and compliance, respectively. The

components and their analogs are listed in Table 1 (see also Ref. 1 for further details).

TABLE 1: PASSIVE ELEMENTS IN MECHANICAL AND ACOUSTIC ANALOGIES

ELECTRICAL COMPONENT	MECHANICAL (IMPEDANCE ANALOGY)	ACOUSTIC
• Resistor • SI unit: Ω	• Mechanical resistance • SI unit: N·s/m • Losses due to friction, as in a car suspension.	• Acoustic resistance • SI unit: kg/(m^4·s) • Dissipative losses due to viscosity and thermal conduction in the fluid.
• Inductor • SI unit: H	• Mass • SI unit: kg • Inertial force, acceleration of a mass.	• Acoustic mass • SI unit: kg/m^4 • Internal force of a volume of air that is accelerated but not compressed.
• Capacitor • SI unit: F	• Compliance • SI unit: m/N • The inverse of the spring constant.	• Acoustic compliance • SI unit: m^5/N • Compressibility effect of a volume of air that is not accelerated.

The active components in a circuit represent sources; they are external forces (pressures) or applied velocities (volume velocities). The sources are also used for couplings between the electrical, mechanical, and acoustic domains.

Note: To make the units fit in COMSOL, it is necessary to make unit transforms to fit the electrical units. When, for example, inserting a capacitor representing a mechanical compliance, C_{MS}, in the physics interface type: `C_MS[F*N/m]`

THE ANALOGOUS CIRCUIT

The analogous circuit for the electrical and mechanical parts of the system sketched in Figure 1 is shown in Figure 2. The upper figure represents the voice coil electrical system and the lower figure the mechanical analogue of the speaker cone, suspensions, and mass of the voice coil. In both figures the node numbers are also shown—they are very useful when setting up the circuit model in COMSOL.

In Figure 2 (top) the external voltage source is denoted by V_0 and the generator output resistance is R_g, in this model $R_g = 0\ \Omega$. The voice coil resistance is R_E, and the voice coil inductance is $L_E(\omega)$, which is frequency dependent. The losses in the magnetic circuit are modeled through the frequency dependent resistance $R'_E(\omega)$. The current controlled voltage source $BL \cdot u_D$, represents the back induced electromagnetic voltage generated when the voice coil (of length L) moves with velocity u_D in the magnetic field B. Here BL is the product of the magnetic field strength and the voice coil length L (see also Ref. 2 on how this can be modeled). In the electrical circuit the current is denoted i_c.

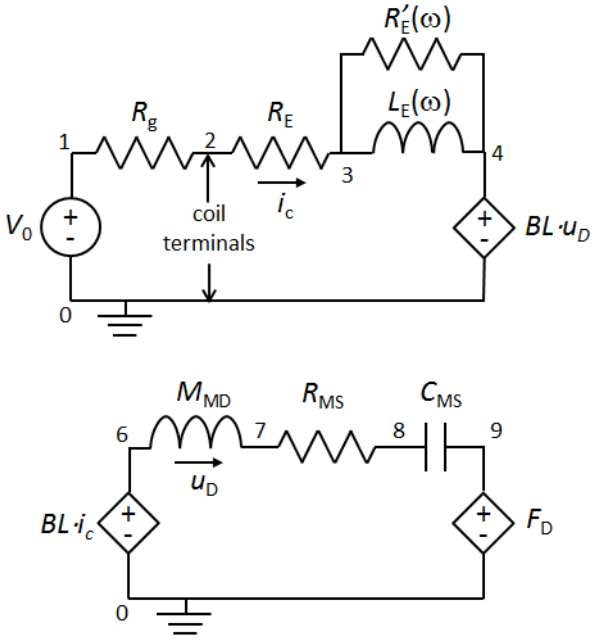


Figure 2: Analogous circuits for the electrical (top) and mechanical (bottom) properties of the speaker driver.

In the mechanical system given in Figure 2 (bottom) the current in the circuit is the velocity of the voice coil and speaker cone in the axial direction (see Figure 1). The force acting on the diaphragm is given by $BL \cdot i_c$. This is the Lorentz force on a voice coil of length L with current i_c, where B is the magnetic flux density. The force acts on a system of mass M_{MD} (voice coil and diaphragm assembly). The resistance R_{MS} models the damping and C_{MS} the mechanical compliance in the speaker suspensions (both spider and outer suspension). Finally, the voltage source F_D represents the pressure force acting on the speaker diaphragm (in the axial direction) and is given by

$$F_D = \int (\Delta p \cdot n_z) dA \tag{1}$$

where Δp is the pressure drop across the diaphragm and n_z is the axial component of the surface normal $\mathbf{n}$ (see Figure 1).This expression gives the couplings from the acoustic finite element model to the lumped circuit model. On the other hand, the coupling from the circuit model to the finite element model comes from specifying the acceleration $\mathbf{a}$ on the surface of the diaphragm, which is given by

$$\mathbf{a} = i\omega u_{\mathrm{D}} \mathbf{e}_z \tag{2}$$

where $\mathbf{e}_z$ is the unit vector in the axial direction (see Figure 3), u_{D} is the current in the mechanical circuit analog, and multiplication with $i\omega$ represents differentiation with respect to time.

Note: A version of this model using the *Lumped Mechanical System* interface also exists as a tutorial: **lumped_loudspeaker_driver_mechanical**. In that model the mechanical part of the system uses the mobility analogy where the present model is based entirely on the impedance analogy. The model additionally requires the Structural Mechanics and the Multibody Dynamics Modules.

SMALL-SIGNAL PARAMETERS

The fundamental small-signal parameters of the system (or Thiele-Small parameters) are the physical parameters of the loudspeaker driver. They are all constants, as they are given in the low-frequency limit of the model. See Table 2 below.

TABLE 2: FUNDAMENTAL SMALL-SIGNAL PARAMETERS

SYMBOL	VALUE	DESCRIPTION
M_{MD}	33.4 g	Moving mass (voice coil and diaphragm)
C_{MS}	$1.18 \cdot 10^{-3}$ m/N	Suspension compliance
R_{MS}	1.85 Ns/m	Suspension mechanical losses (damping)
L_{E}	6.89 mH	Voice coil inductance
R_{E}	7 Ω	DC resistance of voice coil
BL	11.4 T·m	Force factor
S_{D}	$a^2\pi$	Driver equivalent area
a	12 cm	Piston radius of driver (equivalent)

In Table 2 the radius a is the piston radius of the driver. It is typically taken as half the diaphragm aperture diameter, measured half-way into the outer suspension.

Note: The constants used in this model are all taken from Example 22 in Ref. 1.

Other small-signal parameters may be determined on basis of the fundamental small-signal parameters. Actually, these parameters are the ones measured when characterizing a driver; the fundamental parameters are inferred hereafter. The measured small-signal parameters are given in Table 3. In the table the speed of sound is denoted c_0 and the density of air is ρ_0. The values of these parameters are determined in the model using the Parameters feature under Global Definitions.

TABLE 3: SMALL-SIGNAL PARAMETERS (MEASURED)

Symbol	EXPRESSION	DESCRIPTION
M_{MS}	$M_{\mathrm{MD}} + 2S_{\mathrm{D}}^2 \frac{8\rho_0}{3\pi^2 a}$	Moving mass including acoustic load (low frequency approximation).
F_{s}	$\frac{1}{2\pi\sqrt{C_{\mathrm{MS}} M_{\mathrm{MS}}}}$	Fundamental resonant frequency.
Q_{ES}	$\frac{2\pi F_{\mathrm{s}} M_{\mathrm{MS}} R_{\mathrm{E}}}{(BL)^2}$	Electrical Q factor at F_{s}
Q_{MS}	$\frac{2\pi F_{\mathrm{s}} M_{\mathrm{MS}}}{R_{\mathrm{MS}}}$	Mechanical Q factor at F_{s}
Q_{TS}	$\frac{Q_{\mathrm{MS}} Q_{\mathrm{ES}}}{Q_{\mathrm{MS}} + Q_{\mathrm{ES}}}$	Total Q factor at F_{s}
V_{AS}	$\rho_0 c_0^2 S_{\mathrm{D}}^2 C_{\mathrm{MS}}$	Equivalent volume compliance (air volume having the same compliance as the suspension).
η_0	$\frac{4\pi^2 F_{\mathrm{s}}^3 V_{\mathrm{AS}}}{c_0^3 Q_{\mathrm{ES}}}$	Reference efficiency of the driver.

OTHER PARAMETERS

The model also uses other parameters than the small-signal parameters presented above. Two of them are the frequency dependent voice coil inductance $L_{\mathrm{E}}(\omega)$ and the resistance

$R'_E(\omega)$ associated with the losses in the magnetic system. A model for this is given in Ref. 1, for the higher audio frequencies, defining

$$L_E(\omega) = \left[\frac{L_E}{\sin\left(n_e\frac{\pi}{2}\right)}\right]\omega^{(n_e-1)} \qquad R'_E(\omega) = \left[\frac{L_E}{\cos\left(n_e\frac{\pi}{2}\right)}\right]\omega^{n_e}$$

where n_e is the so-called voice coil loss factor. For $n_e = 1$ and for in the low frequency limit the loss-less behavior is recovered where R'_E is an open circuit and L_E is constant. In this model $n_e = 0.7$.

Another expression used in the model is the acoustic radiated power which is given by

$$P_{AR} = \int(\mathbf{n}\cdot\mathbf{I})dA \tag{3}$$

where $\mathbf{I}$ is the intensity vector, whose components are defined in COMSOL as (`acpr.Ir`, `acpr.Iz`). The integral is taken over the front upper side of the speaker diaphragm. The electric input power (RMS value) is defined as

$$P_E = 0.5\mathrm{Re}(V_0\cdot i_c^*) \tag{4}$$

where * is the complex conjugate operator, and, finally, the speaker efficiency comparing the input electric power to the radiated acoustic power is

$$\eta = \frac{P_{AR}}{P_E}$$

A number of the analytical results derived in Ref. 1, which are based on the piston approximation, are also used for comparison with the current more realistic speaker cone shape. They are the low-frequency and high-frequency approximations to the speaker velocity, the on axis pressure at 1 m in front of the piston, and the acoustic radiated power from a piston in an infinite baffle. Expressions for these are given in the reference and under the variable node **Component 1>Definitions>Analytical approximations** in the model.

FINITE ELEMENT DOMAIN

The computational domain where the pressure acoustics model is solved is sketched in Figure 3. It represents the speaker cone, dust cap, and outer suspension, in an infinite baffle in a 2D axisymmetric model. On the speaker (red line) an internal acceleration boundary condition is applied (according to Equation 2) and the rest of the baffle is an internal sound hard wall. The domain is truncated with a perfectly matched layer (PML)

to mimic an infinite open domain. Note that the internal sound hard boundary should also be applied inside the PML, to avoid erroneous energy leaks between the two sides of the infinite baffle. The air domain has a radius of R_{air} while the thickness of the PML layer is R_{pml}. For more information about PMLs, see the *COMSOL Multiphysics Reference Manual*.

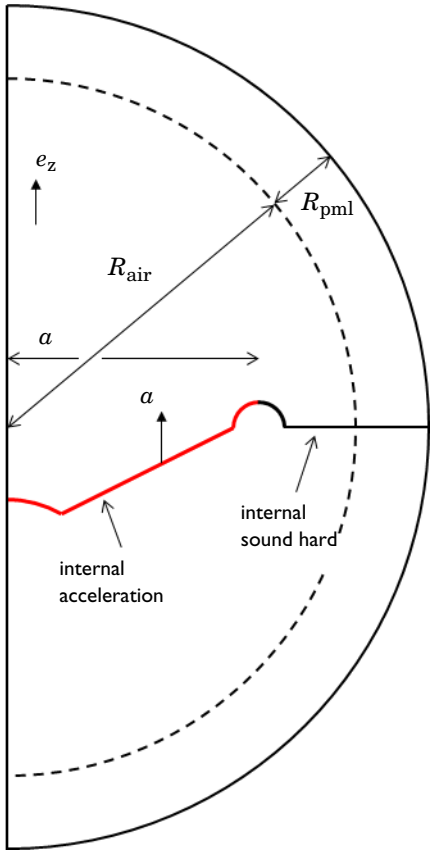


Figure 3: Computational domain and boundary conditions.

Results and Discussion

The generated pressure field is shown in Figure 4 for a frequency of 1000 Hz and 5000 Hz. This plot shows the directive characteristic of the speaker cone at increasing frequencies, this nature is discussed more at the end of this section when discussing the directivity plot Figure 11.

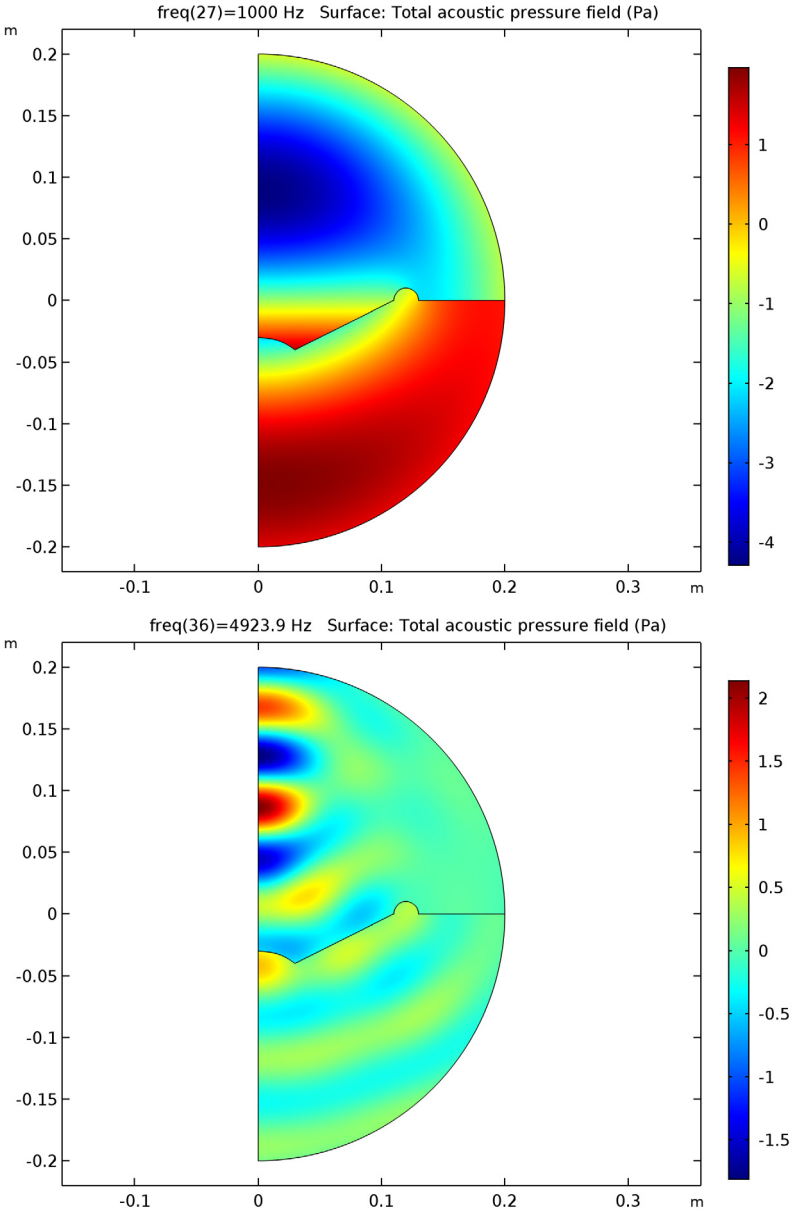


Figure 4: Acoustic pressure for a frequency of 1000 Hz (top) and 5000 Hz (bottom).

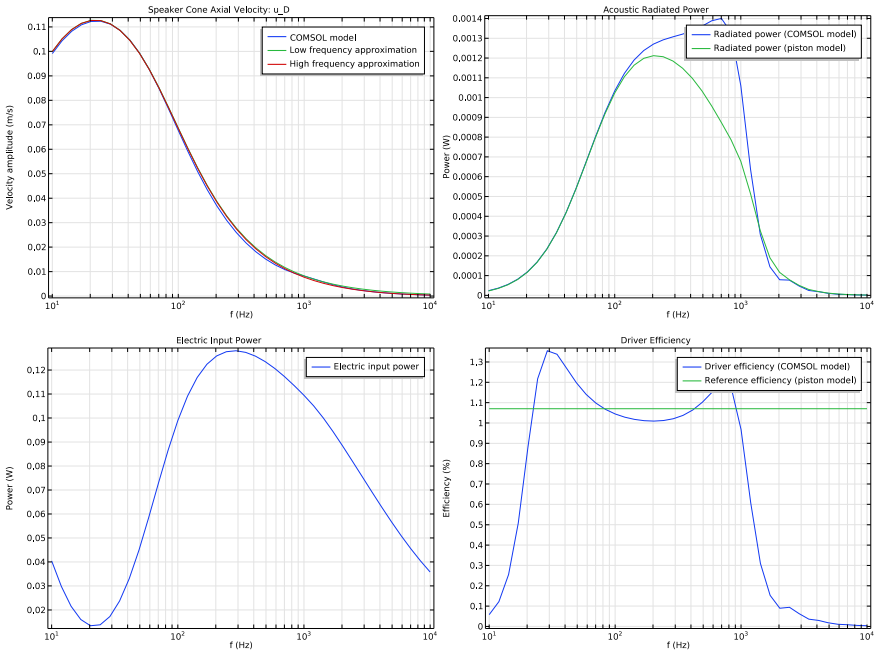


Figure 5: The frequency dependent diaphragm velocity amplitude (top left), acoustic radiated power (top right), electric input power (bottom left), and efficiency (bottom right).

In Figure 5 and Figure 6 a series of characteristic loudspeaker driver curves are shown. In some figures the results obtained with the hybrid lumped and finite element of COMSOL are compared with theoretical curves that are based on the piston approximation.

In the top left of Figure 5 the calculated speaker cone axial velocity u_D is shown together with two theoretical curves representing the approximate high-frequency and low-frequency behavior of a piston driver (see Ref. 1). The general trend is that the low-frequency curve fits well for most of the frequency range. The high-frequency curve is seen to converge toward the higher frequencies. If you increase the frequency range of the model the trend is that the high-frequency approximation is a better fit as expected.

The next two graphs in Figure 5 (top right and bottom left) represent the acoustically radiated power P_{AR} (see Equation 3) and the electric input power P_E (see Equation 4). At the low frequencies the behavior is just as the piston model, as expected. The transition to nonmatching models occurs when the speaker diameter becomes comparable to half the wavelength at around 700 Hz. The last graph of Figure 5 represents the driver efficiency given in percent (%), that is, the ratio of the input electric power to the acoustic radiated

power. The actual efficiency is seen to match well with the predicted reference efficiency η_0 of about 1 %.

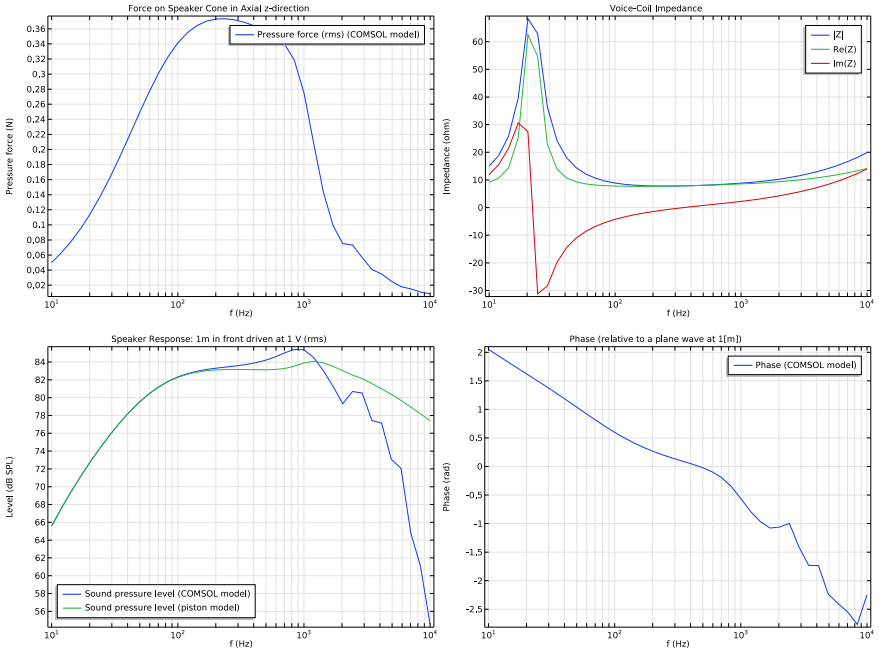


Figure 6: The frequency dependent pressure force on the speaker cone (rms value) (top left), voice coil impedance (top right), speaker sensitivity (bottom right), and phase response (bottom right).

In Figure 6 (top left) the RMS value of the acoustic pressure force F_D on the membrane (see Equation 1) is shown as function of frequency. The RMS value is obtained by the usual formula

$$\sqrt{0.5 \cdot F_D \cdot F_D^*}$$

where the * represents the complex conjugate.

Figure 6 (top right) represents the voice coil impedance (absolute value, real, and imaginary part) calculated as V_i/i_c. The resonance in the electric system is seen to coincide with the fundamental resonance frequency F_s = 22.0 Hz (see Table 3 and the **Global Definitions>Parameters** list in the model).

The two last figures in Figure 6 (bottom left and right) represent the speaker response (sound pressure level) measured 1 m in front and driven at 1 V RMS and the relative phase

measured in the same point. The sensitivity is seen to match the piston model at the low frequencies as expected and it has realistic values for a speaker unit. The phase is represented as the phase of the pressure $p(0,1\text{ m})$ relative to the phase a plane wave would have in the same point $\exp(-ik_0z)$ with $z = 1$ m. In this way the pure distance (phase lag) component of the phase has been removed.

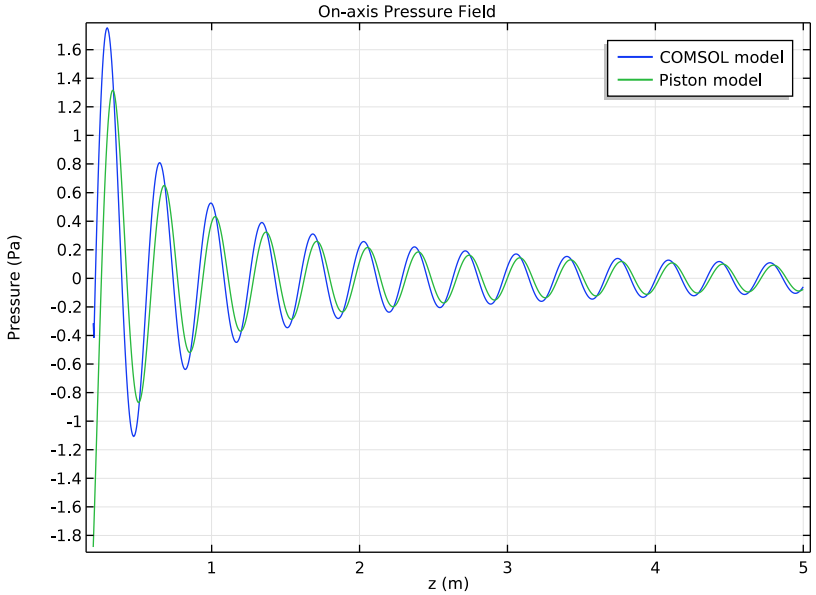


Figure 7: The pressure along the center z-axis from just outside the computational domain to a distance of 5 m. Evaluated for a frequency of 1000 Hz. The pressure is calculated using the far-field calculation feature.

The pressure field along the z-axis is shown in Figure 7 from $z = R_{\text{air}}$ to $z = 5$ m, evaluated at a frequency of 1000 Hz. In the model a *Parametrized Curve 2D* is used to evaluate the far-field pressure outside the computational mesh. The figure compares the modeled pressure with the analytical on axis pressure from a piston. The agreement is seen to be quite good at this frequency. For lower frequencies the agreement becomes much better while the two curves diverge at higher frequencies, as expected (change the evaluation frequency in the plot to see this trend).

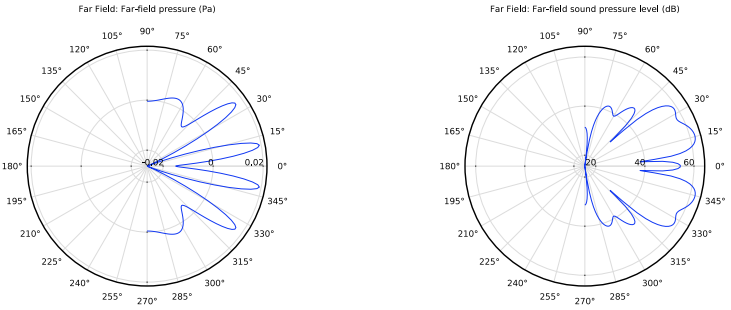


Figure 8: Far-field pressure and sound pressure level evaluated at a distance of 1 m the half sphere in front of the speaker and at 10 kHz.

The pressure and the sound pressure level evaluated at 1 m, using the dedicated far-field plots, are shown in Figure 8, here evaluated at 10 kHz. In the figure the 0° mark corresponds to the axial z direction. Both figures show a very strong directive pattern as expected at this high frequency.

In Figure 9 and Figure 10 the radiated intensity is illustrated and evaluated at the four frequencies 100 Hz, 1000 Hz, 5 kHz, and 10 kHz. The color plot represents the magnitude of the intensity vector $\mathbf{I}$, the domain vector field represents the components of the intensity vector, and finally the vectors plotted on the edges represent the surface normals. The normals are useful when setting up, for example, the expression for the total radiated power (Equation 3). In this case it is necessary to use the `up()` operator to get the intensity on the upper side of the speaker cone (seen relative to the normal direction). The four plots clearly show how the acoustic energy is focused for increasing frequencies.

The final plot of this model is shown in Figure 11, it is a so-called directivity plot of the speaker unit. The plot represents a contour plot of the sound pressure level L_p evaluated along a half circle in front of the speaker as function of the angle and the frequency, that is, $L_\mathrm{p}(\theta, f)$. As this plot is a more advanced and nonstandard plot in COMSOL the axis labels are a bit off. The x-axis represents the angle and actually runs from –90° to 90°. The y-axis is a logarithmic frequency axis running from 10^1 Hz to 10^4 Hz. The plot illustrates how the spatial response goes from a nearly omni-directional constant value at the low frequencies, through a single lobe response at intermediate frequencies, and ends up as a complex directive pattern at high frequencies. This type of plot is very often used, in industry, to characterize speakers and speaker units.

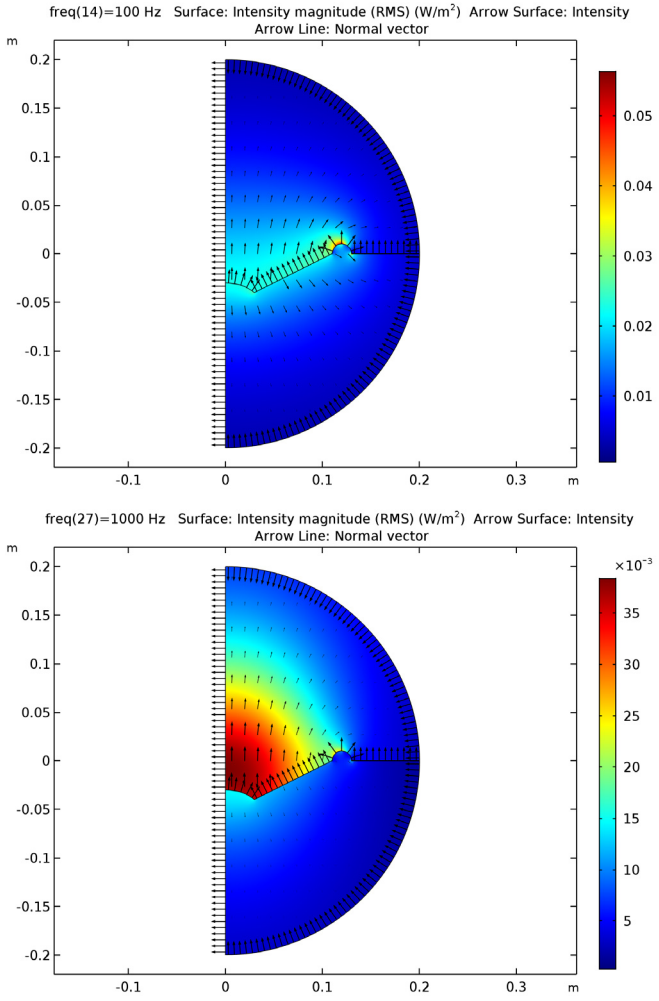


Figure 9: Intensity magnitude (color plot), intensity vector field (domain arrows), and surface normals (edge arrows) for 100 Hz and 1000 Hz.

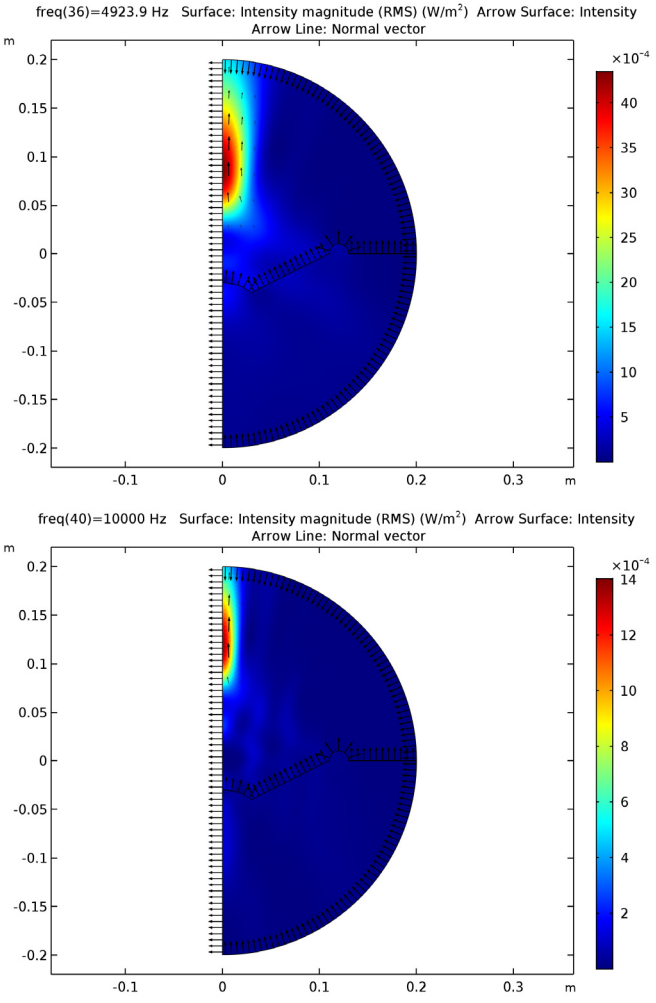


Figure 10: Intensity magnitude (color plot), intensity vector field (domain arrows), and surface normals (edge arrows) for 5 kHz and 10 kHz.

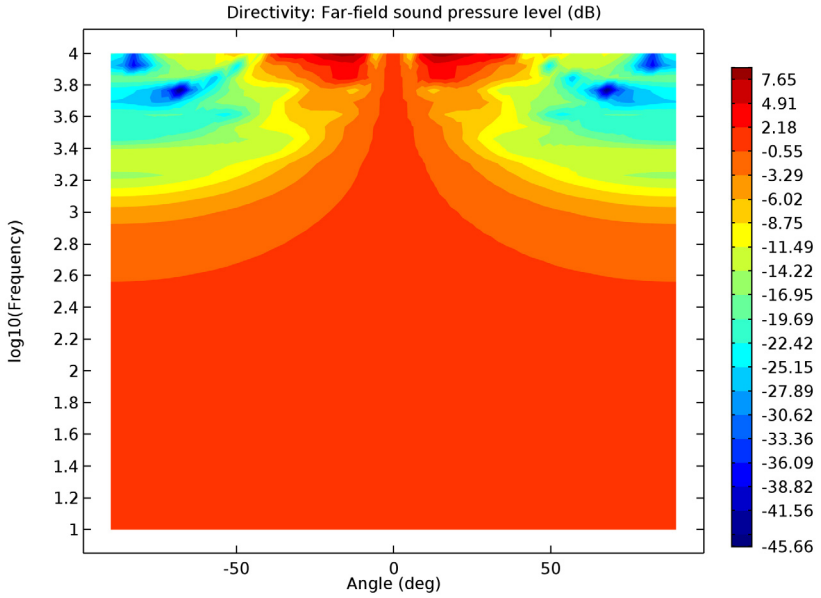


Figure 11: Directivity plot for the speaker. The x-axis is a scaled azimuthal angle that runs from -90° to 90° and the y axis is a logarithmic frequency axis that runs from 10^1 Hz to 10^4 Hz = 10 kHz.

References

1. W. Marshall Leach, Jr., *Introduction to Electroacoustics and Audio Amplifier Design*, Kendall Hunt, 2010.

2. *Loudspeaker Driver Model Documentation*, from the COMSOL Application Library.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver`

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>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.
8. Click **Done**.

GLOBAL DEFINITIONS

Load all the model parameters from a file; they include all the small signal parameters (Table 2 and Table 3) as well as geometry parameters.

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `lumped_loudspeaker_driver_parameters.txt`.

   Build the simple 2D axisymmetric geometry of the speaker driver by drawing some circles and lines.

GEOMETRY 1

Circle 1 (c1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `1[cm]`.
4. In the **Sector angle** text field, type `180`.
5. Locate the **Position** section. In the **r** text field, type `a`.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Rair+Rpml`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Rpml`

Polygon 1 (pol1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. In the **r** text field, type `3[cm] a-1[cm]`.

5 In the **z** text field, type `-4[cm] 0`.

Polygon 2 (pol2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. In the **r** text field, type `a+1[cm] Rair`.

5 In the **z** text field, type `0 0`.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Bézier Polygon 1 (b1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **General** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Polygon Segments** section. Find the **Added segments** subsection. Click **Add Quadratic**.

5 Find the **Control points** subsection. In row **1**, set **z** to `-3[cm]`.

6 In row **2**, set **r** to `1.8[cm]` and **z** to `-3.1[cm]`.

7 In row **3**, set **r** to `3[cm]` and **z** to `-4[cm]`.

8 Find the **Weights** subsection. In the **2** text field, type `1.5`.

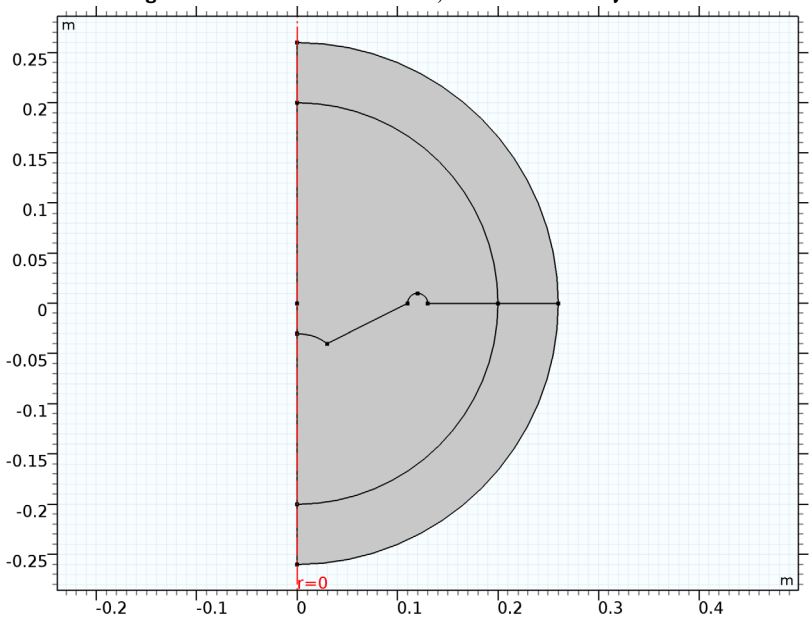
9 Click **Build All Objects**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **c1**, select Boundaries 3 and 4 only.

3 In the **Settings** window for **Delete Entities**, click **Build All Objects**.



Now, set up all the variables, selections, and component couplings under the **Definitions** node. Load the variables from the two variable files (one for model variables and one for the analytical piston expressions). The selections represent the speaker cone surface with an internal acceleration (red line in Figure 3) and the internal sound hard wall.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, type `Model variables` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `lumped_loudspeaker_driver_variables_1.txt`.

Variables 2

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, type `Analytical approximations` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `lumped_loudspeaker_driver_variables_2.txt`.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Speaker` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 6, 11, and 14 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Internal wall` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 7, 8, and 15 only.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Speaker**.

5 In the **Operator name** text field, type `intop`.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 4 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

5 In the **PML scaling factor** text field, type `0.5`.

6 In the **PML scaling curvature parameter** text field, type 5.

You have now changed the default settings for the perfectly matched layer (PML). The new settings will improve the performance of the PML at very low frequencies. First of all, the acoustic radiated power should be positive. If the default settings had been used you would see a negative radiated power at low frequencies (see Figure 5 (top right) and the instructions on how to create the plot, further down). The issue is that at low frequencies the evanescent waves created by the moving speaker cone extend into the PML layer. The interaction between the scaled coordinate system in the PML and these waves may create an erroneous energy contribution to the model (can be either positive or negative). Note that, the evanescent waves decay in only a fraction of a wavelength.

A good way to investigate the performance of the PML is to make a sensitivity analysis on some parameter (for example the total radiated acoustic power) with respect to changes in the PML parameters. In this model we do not perform such a sensitivity analysis.

In general, increasing the curvature factor effectively shifts the resolving power of the PML towards the physical domain, which is necessary in this case since the evanescent components decay in only a fraction of a wavelength. However, if you increase it too much, you may loose resolution in the other end, that is, of the free space wavelength. Assuming that the PMLs work properly for high frequencies and curvature parameter 1, you can in principle do a convergence study increasing a constant curvature parameter until the low-frequency result converges (for 10 Hz) while making sure that the high-frequency result (for 10 kHz) is not affected. It turns out that a value of 5 yields good results in this model. A scaling factor of 0.5 further improves the results, but only by a small amount. Decreasing the scaling factor corresponds to compressing the PML layer (shortening it), which in turn effectively increases the mesh resolution.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Now, set up the physics and the boundary conditions for the model. Use the interior conditions at the diaphragm and on the infinite baffle. This condition will allow for a discontinuous pressure field.

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 In the **Settings** window for **Interior Sound Hard Boundary (Wall)**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Internal wall**.

Interior Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Normal Acceleration**.

2 In the **Settings** window for **Interior Normal Acceleration**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Speaker**.

4 Locate the **Interior Normal Acceleration** section. Specify the $\mathbf{a}_0$ vector as

0	r
u_D*acpr.iomega	z

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 Select Boundary 12 only.

3 In the **Settings** window for **Far-Field Calculation**, locate the **Far-Field Calculation** section.

4 Select the **Symmetry in the z=0 plane** check box.

5 From the **Type of integral** list, choose **Full integral**.

Note that, you have applied the far-field calculation condition only to the front of the speaker. In reality the far-field condition should be applied to boundaries surrounding all sources and scatterers. This is of course not possible with an infinite baffle. However, in this specific case, a trick can be used, namely employing the fact that symmetry (in z = 0) is equal to a sound-hard wall in pressure acoustics. So in the infinite baffle configuration the far-field condition can still be used.

Proceed to set up the electric circuit system for the electric and mechanical model. When building this look at Figure 3 for the references to the node numbers used in the model.

ELECTRICAL CIRCUIT (CIR)

Voltage Source V1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrical Circuit (cir)** and choose **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. In the V_{src} text field, type `V0`.

Resistor R1

1 In the **Model Builder** window, right-click **Electrical Circuit (cir)** and choose **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the R text field, type `R_g`.

Resistor R2

1 Right-click **Electrical Circuit (cir)** and choose **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	3

4 Locate the **Device Parameters** section. In the R text field, type `R_E`.

Inductor L1

1 Right-click **Electrical Circuit (cir)** and choose **Inductor**.

2 In the **Settings** window for **Inductor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the L text field, type `L_E`.

Resistor R3

1 Right-click **Electrical Circuit (cir)** and choose **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the R text field, type `Rp_E`.

Current-Controlled Voltage Source H1

1 Right-click **Electrical Circuit (cir)** and choose **Dependent Sources>Current-Controlled Voltage Source**.

2 In the **Settings** window for **Current-Controlled Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	4
n	0

4 Locate the **Device Parameters** section. In the **Gain** text field, type `BL[m/Wb*ohm]`.

Remember to select the measured current when the mechanical circuit components have been set up.

Current-Controlled Voltage Source H2

1 Right-click **Electrical Circuit (cir)** and choose **Dependent Sources>Current-Controlled Voltage Source**.

2 In the **Settings** window for **Current-Controlled Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	6
n	0

4 Locate the **Device Parameters** section. In the **Gain** text field, type `BL[m/Wb*ohm]`.

5 From the **Measure current for device** list, choose **Resistor R2**.

Inductor L2

1 Right-click **Electrical Circuit (cir)** and choose **Inductor**.

2 In the **Settings** window for **Inductor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	6
n	7

4 Locate the **Device Parameters** section. In the L text field, type `M_MD[H/kg]`.

Current-Controlled Voltage Source H1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrical Circuit (cir)** click **Current-Controlled Voltage Source H1**.

2 In the **Settings** window for **Current-Controlled Voltage Source**, locate the **Device Parameters** section.

3 From the **Measure current for device** list, choose **Inductor L2**.

Resistor R4

1 In the **Model Builder** window, right-click **Electrical Circuit (cir)** and choose **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	7
n	8

4 Locate the **Device Parameters** section. In the R text field, type `R_MS[ohm/kg*s]`.

Capacitor C1

1 Right-click **Electrical Circuit (cir)** and choose **Capacitor**.

2 In the **Settings** window for **Capacitor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	8
n	9

4 Locate the **Device Parameters** section. In the *C* text field, type `C_MS[F*N/m]`.

Voltage Source V2

1 Right-click **Electrical Circuit (cir)** and choose **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	9
n	0

4 Locate the **Device Parameters** section. In the V_{src} text field, type `F_D[V/N]`.

Now, build the mesh. It consists of a triangular mesh around the speaker and an extruded mesh in the PML region. In the boundary where the far-field is calculated add a single boundary layer. For more details on this look at the model Acoustic Scattering off an Ellipsoid in the Application Library.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `343[m/s]/fmax/8`.

5 In the **Minimum element size** text field, type `343[m/s]/fmax/100`.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.

Boundary Layers 1

1 Right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 Select Boundaries 10 and 12 only.

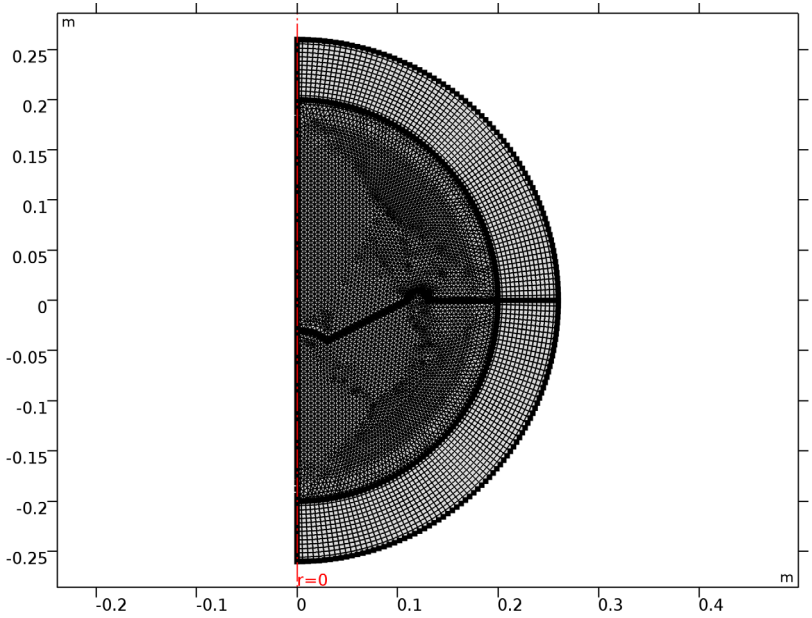
3 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.

4 In the **Number of boundary layers** text field, type `1`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `343[m/s]/fmax/100`.

7 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 `10^{range(1,3/39,4)}`.

4 On the **Home** toolbar, click **Compute**.

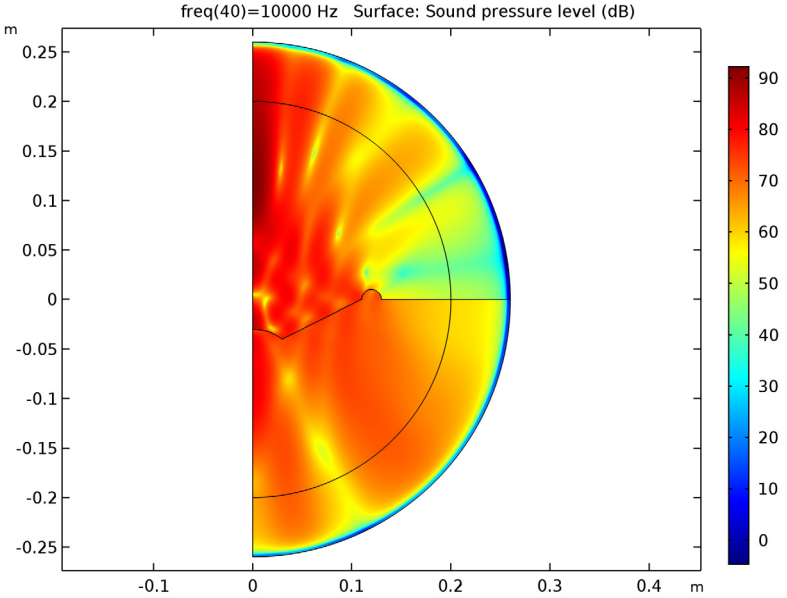
RESULTS

Acoustic Pressure (acpr)

First, look at the default plots. Investigate the 2D **Sound Pressure Level (acpr)** plot to verify the performance of the perfectly matched layer (PML). After doing this you can disable plotting in the PML region, which is non-physical. Secondly, look at the default far-field plots and make a few changes, before setting up a range of plots to investigate the loudspeaker driver performance.

Sound Pressure Level (acpr)

Look at the sound pressure level (SPL) plots at the frequencies of 10 kHz, 1 kHz, and 10 Hz. Note that the SPL decreases nearly 100 dB over the width of the thickness of the PML. This means that the outgoing waves are extremely damped.

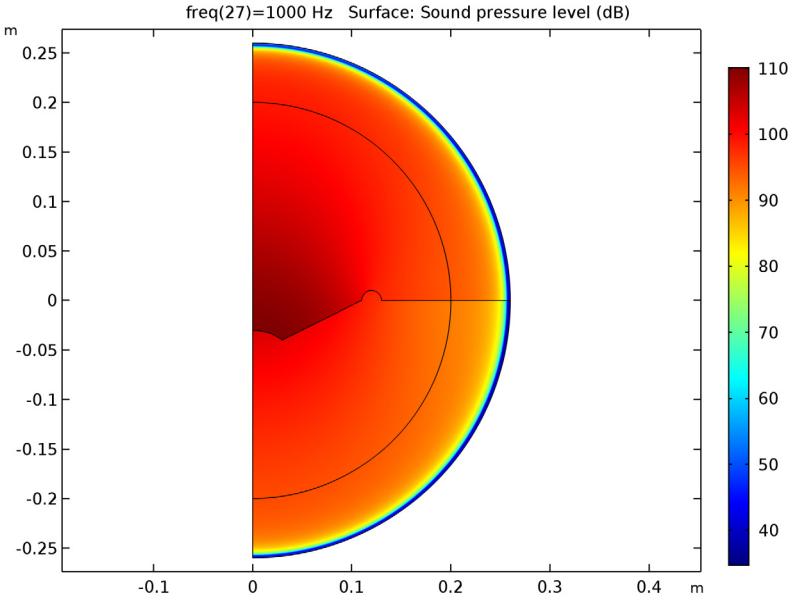


1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

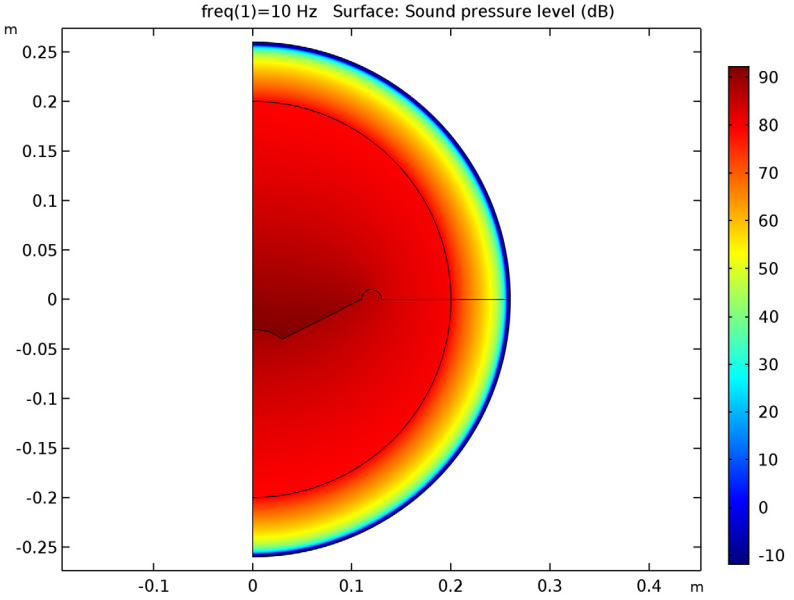
3 From the **Parameter value (freq (Hz))** list, choose **1000**.

4 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.



5 From the **Parameter value (freq (Hz))** list, choose **10**.

6 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.



Study 1/Solution 1 (1) (sol1)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/ Solution 1 (1) (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **1000**.

4 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

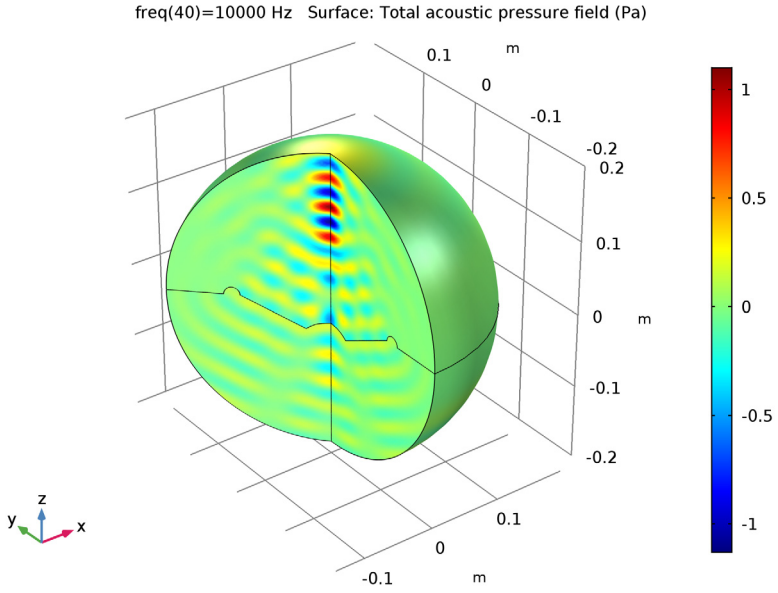
5 Click the **Zoom Extents** button on the **Graphics** toolbar.

6 From the **Parameter value (freq (Hz))** list, choose **4923.9**.

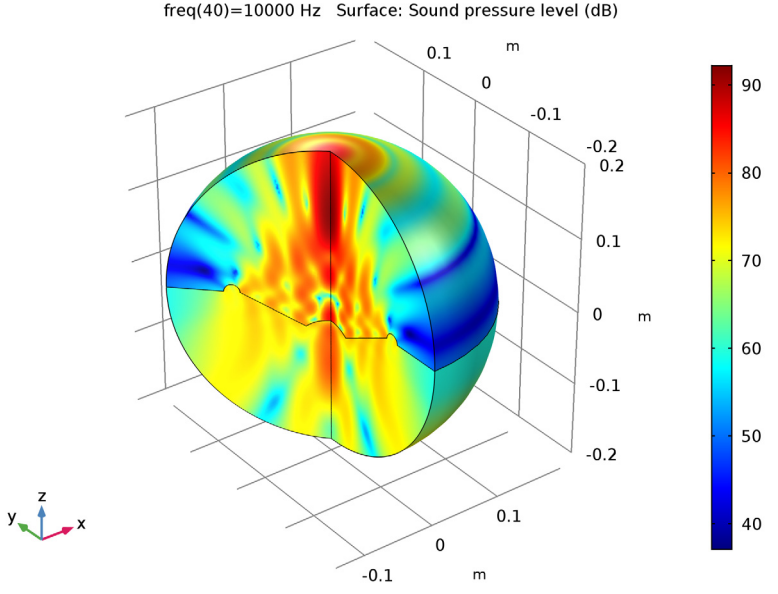
7 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

These two plots should reproduce Figure 4.

Acoustic Pressure, 3D (acpr)



Sound Pressure Level, 3D (acpr)



Far-Field Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Far-Field Sound Pressure Level (acpr)**.

2 In the **Settings** window for **Polar Plot Group**, locate the **Data** section.

3 From the **Parameter selection (freq)** list, choose **Last**.

Far Field 1

1 In the **Model Builder** window, expand the **Far-Field Sound Pressure Level (acpr)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type `180`.

4 From the **Restriction** list, choose **Manual**.

5 In the ϕ **start** text field, type `-90`.

6 In the ϕ **range** text field, type `180`.

7 On the **Far-Field Sound Pressure Level (acpr)** toolbar, click **Plot**.

Far-Field Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Far-Field Pressure (acpr)**.

2 In the **Settings** window for **Polar Plot Group**, locate the **Data** section.

3 From the **Parameter selection (freq)** list, choose **Last**.

Far Field 1

1 In the **Model Builder** window, expand the **Far-Field Pressure (acpr)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type `180`.

4 From the **Restriction** list, choose **Manual**.

5 In the ϕ **start** text field, type `-90`.

6 In the ϕ **range** text field, type `180`.

7 On the **Far-Field Pressure (acpr)** toolbar, click **Plot**.

These two polar plots should reproduce Figure 8. Note that, 0 deg. in the polar plot corresponds to the vertical z-direction.

Now, create nine 1D plots that depict various speaker characteristic plots. Each plot includes a number of steps to setup the title, the axis, the plot name etc., these are optional and can be skipped. You can also just create the specific plot that is of interest to you. The next steps reproduce the graphs in Figure 5, Figure 6, and Figure 7.

The plots are the following:

8 Membrane Velocity

9 Acoustic Radiated Power

10 Electric Input Power

11 Efficiency

12 Force on Speaker Cone

13 Voice-Coil Impedance

14 Sensitivity

15 Phase

16 On-axis Far-Field Pressure

1D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Diaphragm Velocity` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Speaker Cone Axial Velocity: u_D`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Velocity amplitude (m/s)`.

Global 1

1 Right-click **Diaphragm Velocity** 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
`abs(u_D)`	`m/s`	`COMSOL model`
`abs(Qlf_D/S_D)`	`m/s`	`Low frequency approximation`
`abs(Qhf_D/S_D)`	`m/s`	`High frequency approximation`

4 On the **Diaphragm Velocity** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 5 (top left).

1D Plot Group 8

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Acoustic Radiated Power` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Acoustic Radiated Power`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Power (W)`.

Global 1

1 Right-click **Acoustic Radiated Power** 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
`P_AR`	`W`	`Radiated power (COMSOL model)`
`P_AR_ana`	`W`	`Radiated power (piston model)`

4 On the **Acoustic Radiated Power** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 5 (top right).

1D Plot Group 9

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Electric Input Power` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Electric Input Power`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Power (W)`.

Global 1

1 Right-click **Electric Input Power** 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
P_E	W	Electric input power

4 On the **Electric Input Power** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 5 (bottom left).

1D Plot Group 10

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Driver Efficiency` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Driver Efficiency`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Efficiency (%)`.

Global 1

1 Right-click **Driver Efficiency** 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
eta*100	1	Driver efficiency (COMSOL model)
eta0*100		Reference efficiency (piston model)

4 On the **Driver Efficiency** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 5 (bottom right).

1D Plot Group 11

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Force on Speaker Cone` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Force on Speaker Cone in Axial z-direction`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Pressure force (N)`.

Global 1

1 Right-click **Force on Speaker Cone** 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
`sqrt(0.5*F_D*conj(F_D))`	`N`	`Pressure force (rms) (COMSOL model)`

4 On the **Force on Speaker Cone** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 6 (top left).

1D Plot Group 12

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Voice-Coil Impedance` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Voice-Coil Impedance`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Impedance (ohm)`.

Global 1

1 Right-click **Voice-Coil Impedance** 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
`abs(V0/cir.R1_i)`	`ohm`	`|Z|`
`real(V0/cir.R1_i)`	`ohm`	`Re(Z)`
`imag(V0/cir.R1_i)`	`ohm`	`Im(Z)`

4 On the **Voice-Coil Impedance** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 6 (top right).

1D Plot Group 13

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Sensitivity` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Speaker Response: 1m in front driven at 1 V (rms)`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Level (dB SPL)`.

9 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

Octave Band 1

1 On the **Sensitivity** toolbar, click **More Plots** and choose **Octave Band**.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

3 From the **Geometric entity level** list, choose **Global**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `pfar(0,1)`.

5 Locate the **Plot** section. From the **Style** list, choose **Continuous**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Sound pressure level (COMSOL model)`

9 On the **Sensitivity** toolbar, click **Plot**.

Octave Band 2

1 Right-click **Octave Band 1** and choose **Duplicate**.

2 In the **Settings** window for **Octave Band**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `sqrt(2)*prms`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Sound pressure level (piston model)`

5 On the **Sensitivity** toolbar, click **Plot**.

This plot should reproduce Figure 6 (bottom left).

1D Plot Group 14

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Phase` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Phase (relative to a plane wave at 1[m])`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Phase (rad)`.

Global 1

1 Right-click **Phase** 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
`arg(pfar(0,1)/exp(-i*k0*1[m]))`	`rad`	`Phase (COMSOL model)`

4 On the **Phase** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 6 (bottom right).

Set up a parametrized curve used to evaluate the far-field outside of the computational mesh.

Parameterized Curve 2D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 2D**.

2 In the **Settings** window for **Parameterized Curve 2D**, locate the **Expressions** section.

3 In the **R** text field, type `0`.

4 In the **Z** text field, type `s*5[m]+(1-s)*Rair`.

5 Select the **Only evaluate globally defined expressions** check box.

6 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (1) (sol1)**.

1D Plot Group 15

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `On-axis Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Parameterized Curve 2D 1**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `On-axis Pressure Field`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `z (m)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Pressure (Pa)`.

10 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **From list**.

11 In the **Parameter values (freq (Hz))** list, select **1000**.

Line Graph 1

1 Right-click **On-axis Pressure** and choose **Line Graph**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `pfar(r,z)`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `z`.

7 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
COMSOL model

Line Graph 2

1 Right-click **Results>On-axis Pressure>Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `acpr.iomega*rho0*u_D*S_D*exp(-i*k0*z)/(2*pi*z)`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Piston model

5 On the **On-axis Pressure** toolbar, click **Plot**.

This plot should reproduce Figure 7.

Create a 2D intensity plot that includes the magnitude of the intensity vector `acpr.I_rms` as well as an arrow surface (vector field plot) of the intensity vector, with the components (`acpr.Ir`,`acpr.Iz`).

2D Plot Group 16

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Intensity` in the **Label** text field.

Surface 1

1 Right-click **Intensity** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.I_rms`.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Intensity** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Pressure Acoustics, Frequency Domain>Intensity>acpr.Ir,acpr.Iz - Intensity**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Arrow Line 1

1 Right-click **Intensity** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Geometry>acpr.nr,acpr.nz - Normal vector**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

4 On the **Intensity** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

This plot should reproduce the last frame in Figure 10, now change the evaluation frequency to 5000 Hz, 1000 Hz, and 100 Hz. This will reproduce the remaining frames in Figure 9 and Figure 10.

Intensity

1 In the **Model Builder** window, under **Results** click **Intensity**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **4923.9**.

4 On the **Intensity** toolbar, click **Plot**.

5 From the **Parameter value (freq (Hz))** list, choose **1000**.

6 On the **Intensity** toolbar, click **Plot**.

7 From the **Parameter value (freq (Hz))** list, choose **100**.

8 On the **Intensity** toolbar, click **Plot**.

Next create the directivity plot of the speaker.

2D Plot Group 17

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Directivity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

Directivity 1

1 On the **Directivity** toolbar, click **More Plots** and choose **Directivity**.

2 In the **Settings** window for **Directivity**, 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 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Layout** list, choose **Frequency on y-axis**.

7 On the **Directivity** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

This should reproduce the directivity plot depicted in Figure 11. You can tailor the plot to your needs using the normalization options or defining the specific levels to use in the contour plot.

Create an additional plot that shows the sensitivity and the phase in one plot using two y-axis.

Global 1

In the **Model Builder** window, under **Results>Phase** right-click **Global 1** and choose **Copy**.

Sensitivity 1

1 In the **Model Builder** window, under **Results** right-click **Sensitivity** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Sensitivity and Phase` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Octave Band 2

In the **Model Builder** window, expand the **Results>Sensitivity and Phase** node.

Octave Band 1

1 Right-click **Octave Band 2** and choose **Delete**.

2 In the **Model Builder** window, under **Results>Sensitivity and Phase** click **Octave Band 1**.

3 In the **Settings** window for **Octave Band**, locate the **Plot** section.

4 From the **Style** list, choose **1/3 octave bands**.

Global 1

1 In the **Model Builder** window, under **Results** right-click **Sensitivity and Phase** and choose **Paste Global**.

2 In the **Settings** window for **Global**, click to expand the **Coloring and style** section.

3 Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

4 From the **Color** list, choose **Red**.

Sensitivity and Phase

1 In the **Model Builder** window, under **Results** click **Sensitivity and Phase**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **Two y-axes** check box.

4 In the table, enter the following settings:

Plot	Plot on secondary y-axis
Global 1	√

5 Locate the **Axis** section. Select the **Manual axis limits** check box.

6 In the **y minimum** text field, type `40`.

7 In the **y maximum** text field, type `90`.

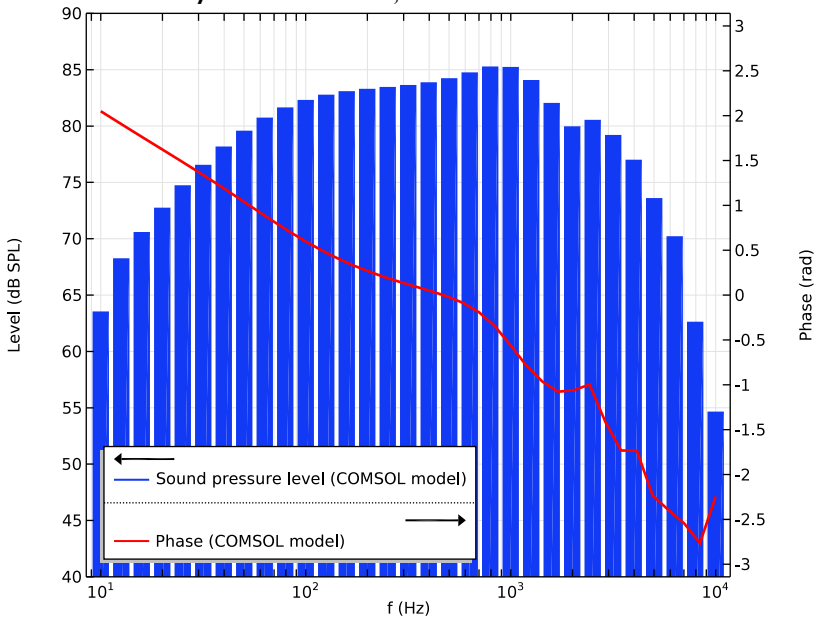
8 In the **Secondary y minimum** text field, type `-pi`.

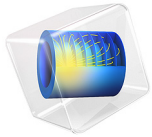
9 In the **Secondary y maximum** text field, type `pi`.

10 Locate the **Plot Settings** section. Select the **Secondary y-axis label** check box.

11 In the associated text field, type `Phase (rad)`.

12 On the **Sensitivity and Phase** toolbar, click **Plot**.





Lumped Loudspeaker Driver using Lumped Mechanical System

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of a moving-coil loudspeaker where a lumped parameter analogy represents the behavior of the electrical and mechanical speaker components. This lumped model is coupled to a 2D axisymmetric pressure acoustics model describing the surrounding air domain.

This example illustrates an alternate way of modeling mechanical components, i.e. mass, spring, damper using the Lumped Mechanical System interface in a lumped loudspeaker driver model. More information regarding the theory or a model set up can be found in the model Lumped Loudspeaker Driver.

Note: This application requires the Acoustics Module, the AC/DC Module, the Structural Mechanics Module, and the Multibody Dynamics Module.

Model Definition

A schematic representation of a moving coil loudspeaker is given in Figure 1. The figure shows a cross section of a loudspeaker. The speaker driver is placed in an infinite baffle with free space in front and on the back of the speaker. The speaker cone consists of the outer suspension, the diaphragm, and a dust cap (not marked in the figure).

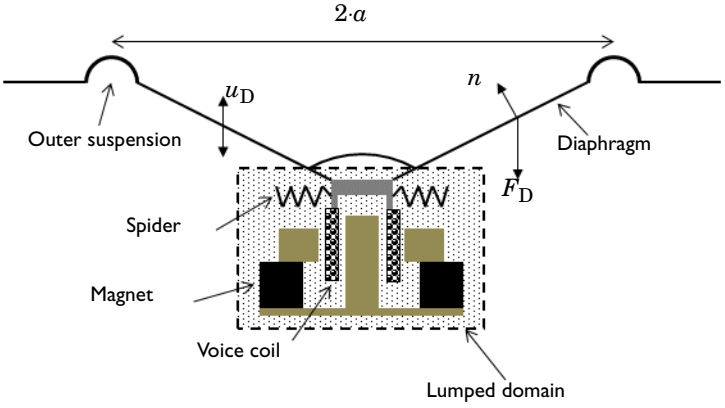


Figure 1: Schematic representation of a moving coil speaker unit.

The mechanical and electrical components of the speaker that are lumped are shown inside the dotted box. On the electrical side it includes the voice-coil and magnetic system (permanent magnet and pole pieces), and on the mechanical side it includes the moving mass of the voice coil and speaker cone, the spring effect of the spider and outer suspension, as well as possible losses due to damping in these suspensions.

LUMPED REPRESENTATION OF ELECTRICAL AND MECHANICAL COMPONENTS

The lumped or circuit representation for the electrical and mechanical parts of the system sketched in Figure 1 is shown in Figure 2. The upper figure represents the voice coil electrical system and the lower figure represents the lumped mechanical system of the speaker cone, suspensions, and mass of the voice coil. In both figures the node numbers are also shown—they are very useful when setting up the circuit model in COMSOL.

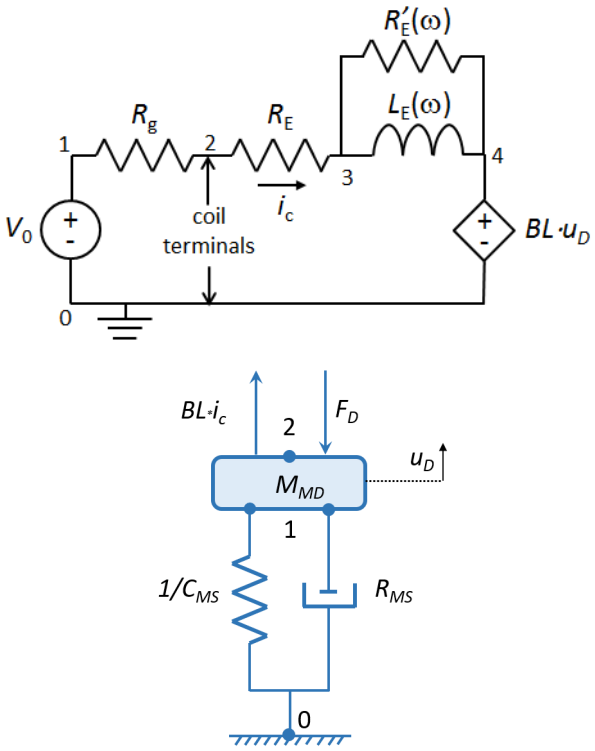


Figure 2: The lumped (circuit) representation for the electrical (top) and mechanical (bottom) components of the speaker driver.

In Figure 2 (top) the external voltage source is denoted by V_0 and the generator output resistance is R_g, in this model $R_g = 0\ \Omega$. The voice coil resistance is R_E, and the voice coil inductance is $L_E(\omega)$, which is frequency dependent. The losses in the magnetic circuit are modeled through the frequency dependent resistance $R'_E(\omega)$. The current controlled voltage source $BL \cdot u_D$, represents the back induced electromagnetic voltage generated when the voice coil (of length L) moves with velocity u_D in the magnetic field B. Here BL is the product of the magnetic field strength and the voice coil length L (see also Ref. 2 on how this can be modeled). In the electrical circuit the current is denoted i_c.

The mechanical system given in Figure 2 (bottom) has the following components:

- Mass (M_{MD}) representing the mass of voice coil and diaphragm assembly
- Spring ($1/C_{MS}$) representing the stiffness of speaker suspensions (both spider and outer suspension)
- Damping (R_{MS}) representing the possible losses in suspensions

The speaker diaphragm is subjected to the following forces acting in the axial direction:

- Lorentz force: The Lorentz force is given by $BL \cdot i_c$ for a voice coil of length L with current i_c, where B is the magnetic flux density.
- Pressure force: The acoustic pressure force is given by

$$F_D = \int (\Delta p \cdot n_z) dA \tag{1}$$

where Δp is the pressure drop across the diaphragm and n_z is the axial component of the surface normal $\mathbf{n}$ (see Figure 1). This expression gives the couplings from the acoustic finite element model to the lumped mechanical model. On the other hand, the coupling from the lumped mechanical model to the finite element model comes from specifying the acceleration on the surface of the diaphragm.

For more details about the model including small signal parameters, and finite element modeling refer Ref. 3.

Results and Discussion

The generated pressure field is shown in Figure 3 for a frequency of 1000 Hz and 5000 Hz. This plot shows the directive characteristic of the speaker cone at increasing frequencies, this nature is discussed more at the end of this section when discussing the directivity plot Figure 7.

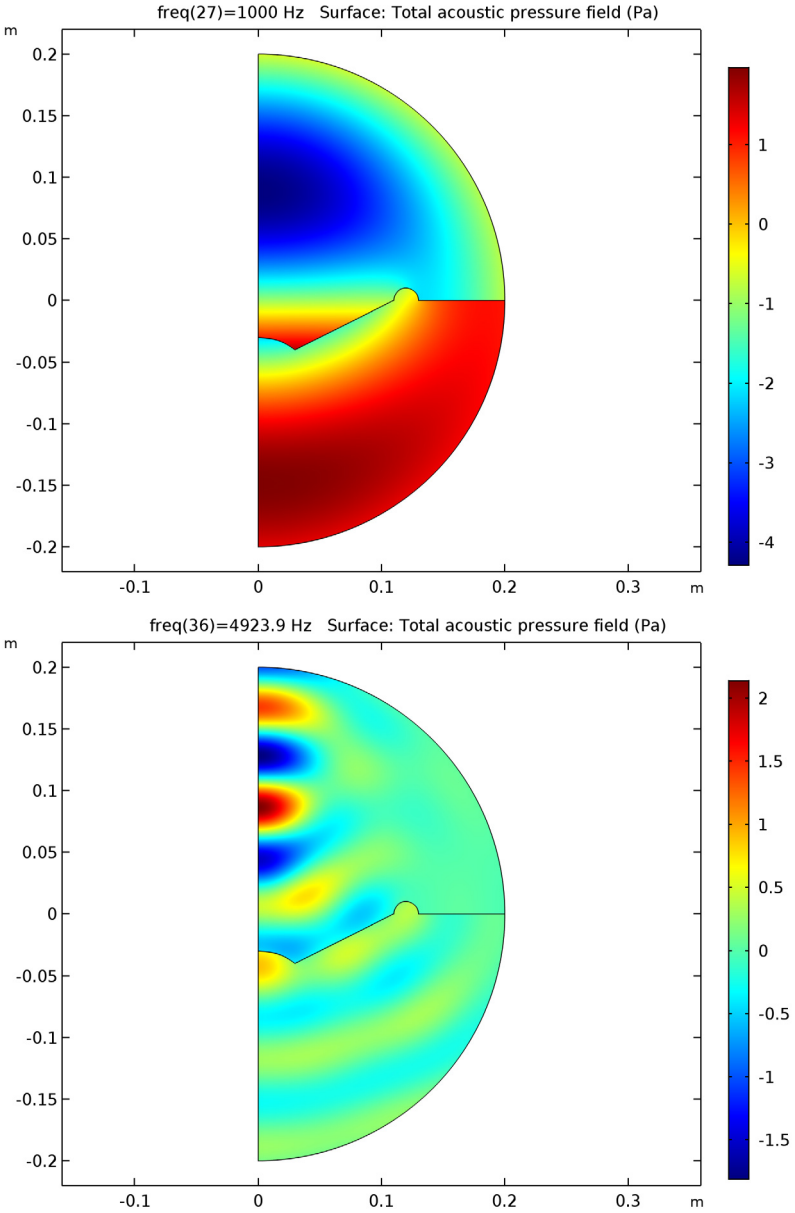


Figure 3: Acoustic pressure for a frequency of 1000 Hz (top) and 5000 Hz (bottom).

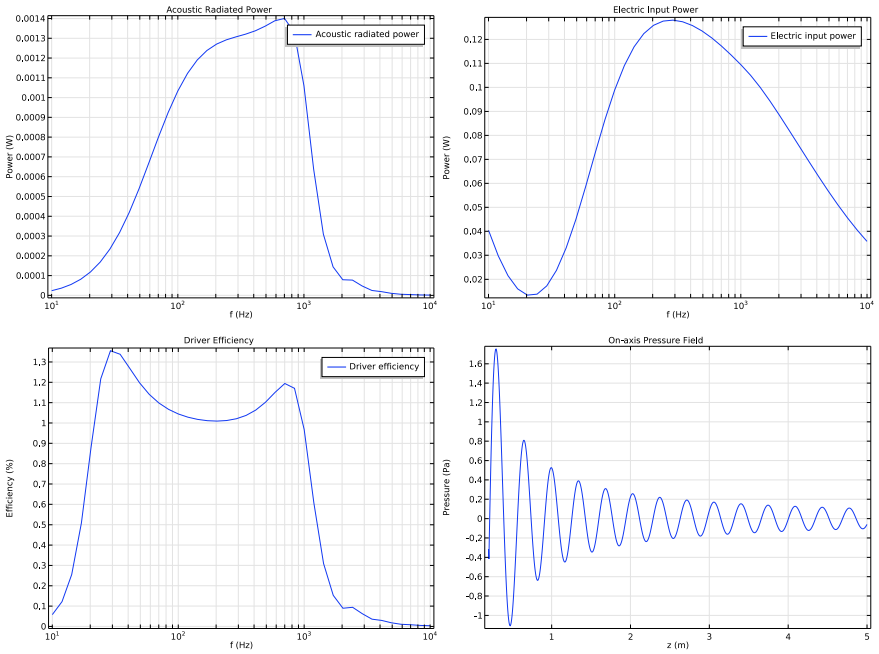


Figure 4: The frequency dependent acoustic radiated power (top left), electric input power (top right), efficiency (bottom left), and on-axis pressure field (bottom right).

The first two graphs in Figure 4 (top left and top right) represent the acoustically radiated power P_{AR} and the electric input power P_{E}.

The third (bottom left) graph of Figure 4 represents the driver efficiency given in percent (%), that is, the ratio of the acoustic radiated power and the input electric power.

The pressure field along the z-axis is shown in the last graph of Figure 4 from $z = R_{\mathrm{air}}$ to $z = 5$ m, evaluated at a frequency of 1000 Hz. In the model a *Parametrized Curve 2D* is used to evaluate the far-field pressure outside the computational mesh.

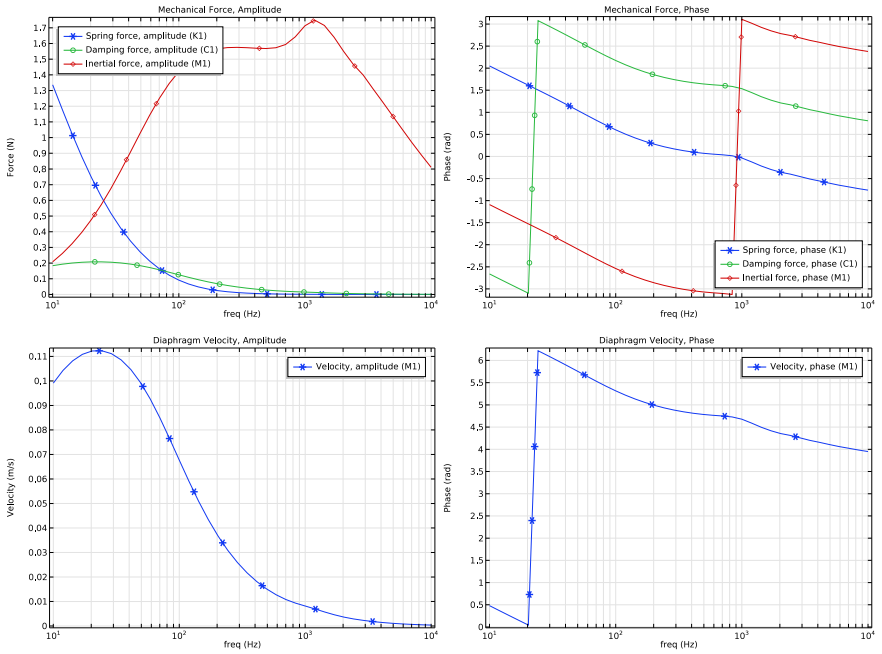


Figure 5: The frequency dependent mechanical force amplitude (top left), corresponding phase (top right), diaphragm velocity amplitude (bottom left), and corresponding phase (bottom right).

In the top row of Figure 5 the frequency variation of the mechanical forces in mass, spring, and damper components are shown. The left graph shows the force amplitude where as on the right graph, the phase is plotted. It can be seen that spring force is dominant at low frequencies whereas the inertial force starts to dominate at higher frequencies. In the bottom row of Figure 5 the frequency variation of the amplitude and phase of the speaker cone axial velocity is shown.

The pressure and the sound pressure level evaluated at 1 m, using the dedicated far-field plots, are shown in Figure 6, here evaluated at 10 kHz. In the figure the 0° mark corresponds to the axial z direction. Both figures show a very strong directive pattern as expected at this high frequency.

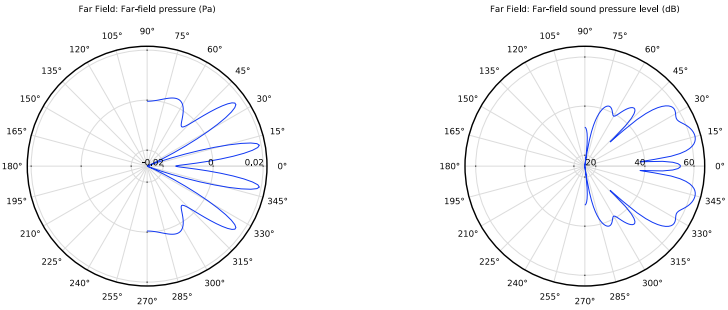


Figure 6: Far-field pressure and sound pressure level evaluated at a distance of 1 m the half sphere in front of the speaker and at 10 kHz.

In Figure 7 (top) the radiated intensity is illustrated which is evaluated at 100 Hz. The color plot represents the magnitude of the intensity vector $\mathbf{I}$, the domain vector field represents the components of the intensity vector, and finally the vectors plotted on the edges represent the surface normals.

The final plot of this model is shown in Figure 7 (bottom), it is a so-called directivity plot of the speaker unit. The plot represents a contour plot of the sound pressure level L_p evaluated along a half circle in front of the speaker as function of the angle and the frequency, that is, $L_\mathrm{p}(\theta, f)$. As this plot is a more advanced and nonstandard plot in COMSOL the axis labels are a bit off. The x-axis represents the angle and actually runs from –90° to 90°. The y-axis is a logarithmic frequency axis running from 10^1 Hz to 10^4 Hz. The plot illustrates how the spatial response goes from a nearly omni-directional constant value at the low frequencies, through a single lobe response at intermediate frequencies, and ends up as a complex directive pattern at high frequencies. This type of plot is very often used, in industry, to characterize speakers and speaker units.

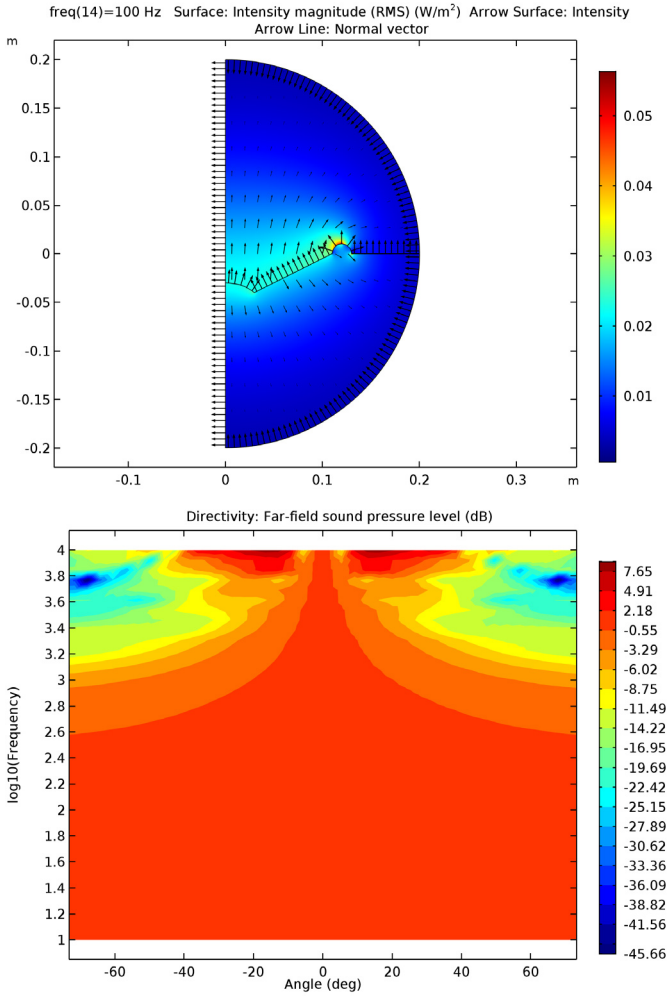


Figure 7: Top: intensity magnitude (color plot), intensity vector field (domain arrows), and surface normals (edge arrows) for 100 Hz. Bottom: directivity plot for the speaker. The x-axis is a scaled azimuthal angle that runs from -90° to 90° and the y axis is a logarithmic frequency axis that runs from 10^1 Hz to 10^4 Hz.

References

1. W. Marshall Leach, Jr., *Introduction to Electroacoustics and Audio Amplifier Design*, Kendall Hunt, 2010.

2. *Loudspeaker Driver Model Documentation*, from the COMSOL Application Library.

3. *Lumped Loudspeaker Driver Model Documentation*, from the COMSOL Application Library.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver_mechanical`

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>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.
5. Click **Add**.
6. In the **Select Physics** tree, select **Structural Mechanics>Lumped Mechanical System (lms)**.
7. Click **Add**.
8. Click **Study**.
9. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.
10. Click **Done**.

GLOBAL DEFINITIONS

Load small signal and geometric parameters from a file.

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `lumped_loudspeaker_driver_mechanical_parameters.txt`.

Import a simple 2D axisymmetric geometry of the speaker driver.

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `lumped_loudspeaker_driver_mechanical.mphbin`.

5 Click **Import**.

6 On the **Home** toolbar, click **Build All**.

Now, set up all the variables, selections, and component couplings under the **Definitions** node.

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, type `Model variables` in the **Label** text field.

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `lumped_loudspeaker_driver_mechanical_variables.txt`.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Speaker` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 6, 11, and 14 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Internal wall` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 7, 8, and 15 only.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Speaker**.

5 In the **Operator name** text field, type `intop`.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 4 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

5 In the **PML scaling factor** text field, type `0.5`.

6 In the **PML scaling curvature parameter** text field, type `5`.

You have now changed the default settings for the perfectly matched layer (PML). The new settings will improve the performance of the PML at very low frequencies.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Now, set up the physics and the boundary conditions for the model.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 In the **Settings** window for **Interior Sound Hard Boundary (Wall)**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Internal wall**.

Interior Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Normal Acceleration**.

2 In the **Settings** window for **Interior Normal Acceleration**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Speaker**.

4 Locate the **Interior Normal Acceleration** section. Specify the $\mathbf{a}_0$ vector as

0	r
lms.M1_a	z

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 Select Boundary 12 only.

3 In the **Settings** window for **Far-Field Calculation**, locate the **Far-Field Calculation** section.

4 Select the **Symmetry in the z=0 plane** check box.

5 From the **Type of integral** list, choose **Full integral**.

Proceed to set up the electric circuit system and lumped mechanical system part of the model. When building this look at Figure 2 for the references to the node numbers used in the model.

ELECTRICAL CIRCUIT (CIR)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

Voltage Source V1

1 On the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	0

4 Locate the **Device Parameters** section. In the V_{src} text field, type V0.

Resistor R1

1 On the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	1
n	2

4 Locate the **Device Parameters** section. In the R text field, type `R_g`.

Resistor R2

1 On the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	3

4 Locate the **Device Parameters** section. In the R text field, type `R_E`.

Inductor L1

1 On the **Electrical Circuit** toolbar, click **Inductor**.

2 In the **Settings** window for **Inductor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the L text field, type `L_E`.

Resistor R3

1 On the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	4

4 Locate the **Device Parameters** section. In the R text field, type `Rp_E`.

Voltage Source V2

1 On the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	4
n	0

4 Locate the **Device Parameters** section. In the V_{src} text field, type `BL*lms.M1_v`.

LUMPED MECHANICAL SYSTEM (LMS)

In the **Model Builder** window, under **Component 1 (comp1)** click **Lumped Mechanical System (lms)**.

Spring K1

1 On the **Physics** toolbar, click **Global** and choose **Spring**.

2 In the **Settings** window for **Spring**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p1	0
p2	1

4 Locate the **Component Parameters** section. In the k text field, type `1/C_MS`.

5 Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

Damper C1

1 On the **Physics** toolbar, click **Global** and choose **Damper**.

2 In the **Settings** window for **Damper**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p1	0
p2	1

4 Locate the **Component Parameters** section. In the c text field, type `R_MS`.

5 Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

Mass M1

1 On the **Physics** toolbar, click **Global** and choose **Mass**.

2 In the **Settings** window for **Mass**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p1	1
p2	2

4 Locate the **Component Parameters** section. In the m text field, type `M_MD`.

5 Locate the **Results** section. Find the **Add the following to default results** subsection. Clear the **Displacement** check box.

6 Select the **Velocity** check box.

Force Node 1

1 On the **Physics** toolbar, click **Global** and choose **Force Node**.

2 In the **Settings** window for **Force Node**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node name
p1	2

4 Locate the **Terminal Parameters** section. In the f_{p10} text field, type `BL*cir.R2_i-F_D`.

Now, build the mesh. It consists of a triangular mesh around the speaker and an extruded mesh in the PML region. In the boundary where the far-field is calculated add a single boundary layer.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `343[m/s]/fmax/8`.

5 In the **Minimum element size** text field, type `343[m/s]/fmax/100`.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.

Boundary Layers 1

1 Right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 Select Boundaries 10 and 12 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.

4 In the **Number of boundary layers** text field, type `1`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `343[m/s]/fmax/100`.

7 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 `10^{range(1,3/39,4)}`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

First, look at the default plots. Investigate the 2D **Sound Pressure Level (acpr)** plot to verify the performance of the perfectly matched layer (PML). After doing this you can disable plotting in the PML region, which is non-physical. Secondly, look at the default far-field plots and make a few changes, before setting up a range of plots to investigate the loudspeaker driver performance.

Sound Pressure Level (acpr)

Look at the sound pressure level (SPL) plots at the frequencies of 10 kHz, 1 kHz, and 10 Hz. Note that the SPL decreases nearly 100 dB over the width of the thickness of the PML. This means that the outgoing waves are extremely damped.

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **1000**.

4 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

5 From the **Parameter value (freq (Hz))** list, choose **10**.

6 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

Study 1/Solution 1 (1) (sol1)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/ Solution 1 (1) (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **1000**.

4 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

6 From the **Parameter value (freq (Hz))** list, choose **4923.9**.

7 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

These two plots should reproduce Figure 4.

Far-Field Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Far-Field Sound Pressure Level (acpr)**.

2 In the **Settings** window for **Polar Plot Group**, locate the **Data** section.

3 From the **Parameter selection (freq)** list, choose **Last**.

Far Field 1

1 In the **Model Builder** window, expand the **Far-Field Sound Pressure Level (acpr)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type `180`.

4 From the **Restriction** list, choose **Manual**.

5 In the ϕ **start** text field, type `-90`.

6 In the ϕ **range** text field, type `180`.

7 On the **Far-Field Sound Pressure Level (acpr)** toolbar, click **Plot**.

Far-Field Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Far-Field Pressure (acpr)**.

2 In the **Settings** window for **Polar Plot Group**, locate the **Data** section.

3 From the **Parameter selection (freq)** list, choose **Last**.

Far Field 1

1 In the **Model Builder** window, expand the **Far-Field Pressure (acpr)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Evaluation** section.

3 Find the **Angles** subsection. In the **Number of angles** text field, type `180`.

4 From the **Restriction** list, choose **Manual**.

5 In the ϕ **start** text field, type `-90`.

6 In the ϕ **range** text field, type `180`.

7 On the **Far-Field Pressure (acpr)** toolbar, click **Plot**.

These two polar plots should reproduce Figure 7. Note that, 0 deg. in the polar plot corresponds to the vertical z-direction.

Now modify the default plots of **Lumped Mechanical System** interface to reproduce Figure 6.

Force, Amplitude (K1)

1 In the **Model Builder** window, under **Results** click **Force, Amplitude (K1)**.

2 In the **Settings** window for **1D Plot Group**, type `Mechanical Force, Amplitude` in the **Label** text field.

3 Click to expand the **Title** section. In the **Title** text area, type `Mechanical Force, Amplitude`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Force (N)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

6 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

7 On the **Mechanical Force, Amplitude** toolbar, click **Plot**.

Force, Phase (K1)

1 In the **Model Builder** window, under **Results** click **Force, Phase (K1)**.

2 In the **Settings** window for **1D Plot Group**, type `Mechanical Force, Phase` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Mechanical Force, Phase`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Phase (rad)`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

6 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

7 On the **Mechanical Force, Phase** toolbar, click **Plot**.

Velocity, Amplitude (M1)

1 In the **Model Builder** window, under **Results** click **Velocity, Amplitude (M1)**.

2 In the **Settings** window for **1D Plot Group**, type `Diaphragm Velocity, Amplitude` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Diaphragm Velocity, Amplitude`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Velocity (m/s)`.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

6 On the **Diaphragm Velocity, Amplitude** toolbar, click **Plot**.

Velocity, Phase (M1)

1 In the **Model Builder** window, under **Results** click **Velocity, Phase (M1)**.

2 In the **Settings** window for **1D Plot Group**, type `Diaphragm Velocity, Phase` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Diaphragm Velocity, Phase`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Phase (rad)`.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

6 On the **Diaphragm Velocity, Phase** toolbar, click **Plot**.

1D Plot Group 11

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Acoustic Radiated Power` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Acoustic Radiated Power`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Power (W)`.

Global 1

1 Right-click **Acoustic Radiated Power** 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
P_AR	W	Acoustic radiated power

4 On the **Acoustic Radiated Power** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 5 (top left).

1D Plot Group 12

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Electric Input Power` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Electric Input Power`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Power (W)`.

Global 1

1 Right-click **Electric Input Power** 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
`P_E`	`W`	`Electric input power`

4 On the **Electric Input Power** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 5 (top right).

1D Plot Group 13

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Driver Efficiency` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Driver Efficiency`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Efficiency (%)`.

Global 1

1 Right-click **Driver Efficiency** 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
`eta*100`	`1`	`Driver efficiency`

4 On the **Driver Efficiency** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

This plot should reproduce Figure 5 (bottom left).

Set up a parametrized curve used to evaluate the far-field outside of the computational mesh.

Parameterized Curve 2D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 2D**.

2 In the **Settings** window for **Parameterized Curve 2D**, locate the **Expressions** section.

3 In the **R** text field, type `0`.

4 In the **Z** text field, type `s*5[m]+(1-s)*Rair`.

5 Select the **Only evaluate globally defined expressions** check box.

6 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 (1) (sol1)**.

1D Plot Group 14

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `On-axis Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Parameterized Curve 2D 1**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `On-axis Pressure Field`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `z (m)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Pressure (Pa)`.

10 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **From list**.

11 In the **Parameter values (freq (Hz))** list, select **1000**.

12 Locate the **Legend** section. Clear the **Show legends** check box.

Line Graph 1

1 Right-click **On-axis Pressure** and choose **Line Graph**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `pfar(r,z)`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `z`.

7 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

8 On the **On-axis Pressure** toolbar, click **Plot**.

This plot should reproduce Figure 5 (bottom right).

Create a 2D intensity plot that includes the magnitude of the intensity vector `acpr.I_rms` as well as an arrow surface (vector field plot) of the intensity vector, with the components (`acpr.Ir`,`acpr.Iz`).

2D Plot Group 15

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Intensity` in the **Label** text field.

Surface 1

1 Right-click **Intensity** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.I_rms`.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Intensity** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1> Pressure Acoustics, Frequency Domain>Intensity>acpr.Ir,acpr.Iz - Intensity**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Arrow Line 1

1 Right-click **Intensity** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1> Pressure Acoustics, Frequency Domain>Geometry>acpr.nr,acpr.nz - Normal vector**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

4 On the **Intensity** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Now change the evaluation frequency to 5000 Hz, 1000 Hz, and 100 Hz in order to plot and reproduce Figure 7.

Intensity

1 In the **Model Builder** window, under **Results** click **Intensity**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **4923.9**.

4 On the **Intensity** toolbar, click **Plot**.

5 From the **Parameter value (freq (Hz))** list, choose **1000**.

6 On the **Intensity** toolbar, click **Plot**.

7 From the **Parameter value (freq (Hz))** list, choose **100**.

8 On the **Intensity** toolbar, click **Plot**.

Next create the directivity plot of the speaker.

2D Plot Group 16

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Directivity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

Directivity 1

1 On the **Directivity** toolbar, click **More Plots** and choose **Directivity**.

2 In the **Settings** window for **Directivity**, locate the **Evaluation** section.

3 Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.

4 In the ϕ **start** text field, type `-90`.

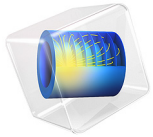
5 In the ϕ **range** text field, type `180`.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Layout** list, choose **Frequency on y-axis**.

7 On the **Directivity** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

This should reproduce the directivity plot depicted in Figure 7. You can tailor the plot to your needs using the normalization options or defining the specific levels to use in the contour plot.



Lumped Receiver Connected to Test Setup with an 0.4-cc Coupler

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

When simulations are involved in the development of mobile devices, consumer electronics, hearing aids, or headsets, it is necessary to consider how the transducers interact with the rest of the system. For some tasks, such as analyzing the vibration isolation of an elastic mounting of the transducer, it may be necessary to include a fully detailed multiphysics model of the transducer itself. In applications where only the electroacoustic response of the device is of interest, a lumped parameter model of the transducer (an electroacoustic analogy as would be implemented in Spice) can be coupled with a multiphysics model of the rest of the system.

In this example a Knowles ED-23146 balanced armature receiver (miniature loudspeaker) is connected to a test set-up consisting of a 50 mm (1 mm ID) tube into a generic 0.4-cc measurement coupler.[1] This test set-up represents the receiver in a behind-the-ear hearing aid driving an ear canal with a deeply inserted ear mold via a long narrow tube. Data collected when using the 0.4-cc coupler will give a more realistic assessment of acoustic data for deeply inserted devices compared to using the 711 coupler (see Ref. 1). The model shows how to connect a receiver modeled as a lumped Spice network to the acoustic system of the tube and a measurement coupler modeled in the finite element domain. The response at the measurement microphone in the coupler as well as the electric input impedance to the receiver are compared with measurements. The losses in the long narrow tube are in this model included using one of the equivalent fluid models of the Narrow Region Acoustics feature in the Pressure Acoustics, Frequency Domain interface.

Note: This application requires the Acoustics Module and the AC/DC Module.

Model Definition

A sketch of the measurement set-up analyzed in this model is depicted in Figure 1. It consists of the Knowles ED-23146 receiver which is modeled as an electric Spice circuit using the Electrical Circuit interface. The tube has a length of 50 mm and a diameter of 1 mm and the 0.4-cc coupler is here simply represented by a cylinder with a volume close to 0.4 cm^3 (and a small intrusion at one end representing the microphone location). Real

1. This model was created based upon data provided by Knowles, IL, USA.

measurement couplers have more complex geometries. The acoustics inside the tube and the coupler are modeled using the Pressure Acoustics, Frequency Domain interface.

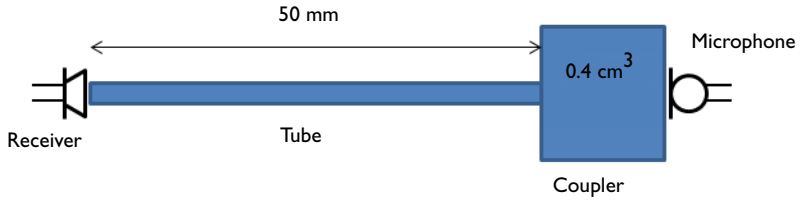


Figure 1: Schematic representation of the modeled system consisting of receiver, tube, coupler, and measurement microphone. The blue domains are modeled using finite elements.

The actual receiver is depicted in Figure 2. It is of the balanced armature type. Knowles provides lumped models for all their transducers enabling engineers to model the devices where they are used.

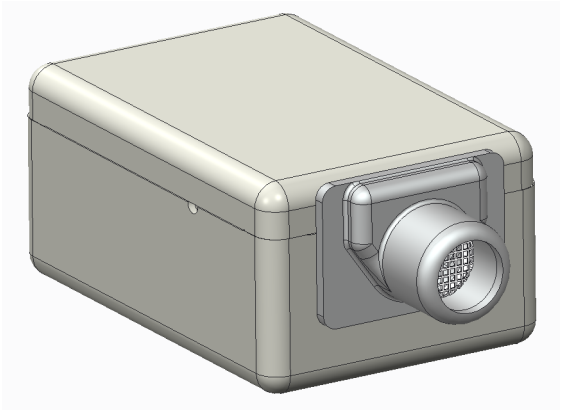


Figure 2: Rendering of the Knowles ED-23146 balanced armature receiver. The device is roughly 4 mm wide, 6 mm deep, and 3 mm high. Image credit: Knowles, IL, USA.

In the Spice network used for the Knowles ED-23146 receiver the current at the output corresponds to a volume flow (m^3/s) and voltage over the output terminals corresponds to the pressure (Pa), as measured at the outlet of the transducer. See Ref. 2 and Ref. 3 for further details.

The current I_{out} at the output of the Spice receiver (using units of m^3/s) model corresponds to the volume flow at the sprout (out let of the receiver). This means that the inward normal acceleration a_{n} at the inlet of the tube is given by

$$a_n = i\omega \frac{I_{out}}{A} \qquad A = \int_{\Omega} 1 dA \tag{1}$$

where A is the cross section area of the tube inlet. The average pressure at the tube inlet is coupled back to the Spice network and added as a voltage source. In this way there is a two way coupling between the finite element domain and the Spice network.

At the boundary where the measurement microphone is normally located in the coupler, a normal impedance condition has been added to account for the compliance of the microphone diaphragm. In this model, the compliance C_{mic} is equivalent to a volume of 6 mm^3. The acoustic mass L_{mic} and resistance R_{mic} of the diaphragm are set to typical values. The impedance is given by a simple RLC model

$$Z = \frac{1}{i\omega C_{mic}} + R_{mic} + i\omega L_{mic} . \tag{2}$$

This is implemented by applying the built-in RCL option found in the impedance boundary condition.

Results and Discussion

The response, as measured at the location of the microphone, is depicted in Figure 3. The model results are compared with measurements performed on an actual system and with results obtained without the viscous and thermal acoustics losses. The measurements are seen to agree well with the full model results. At high frequencies (above 14 kHz) the results start not to match well. Here the wavelength becomes comparable to length scales of structures in the coupler (quarter wave length is 0.6 cm) that are not included in this simple cylinder representation, for example, the protective mesh of the microphone. In addition, of course, the lumped parameter model of the receiver is inexact at these high frequencies. Note that the model curves where no acoustic losses are added still show signs of damping. This is due to the losses included in the Spice model of the transducer.

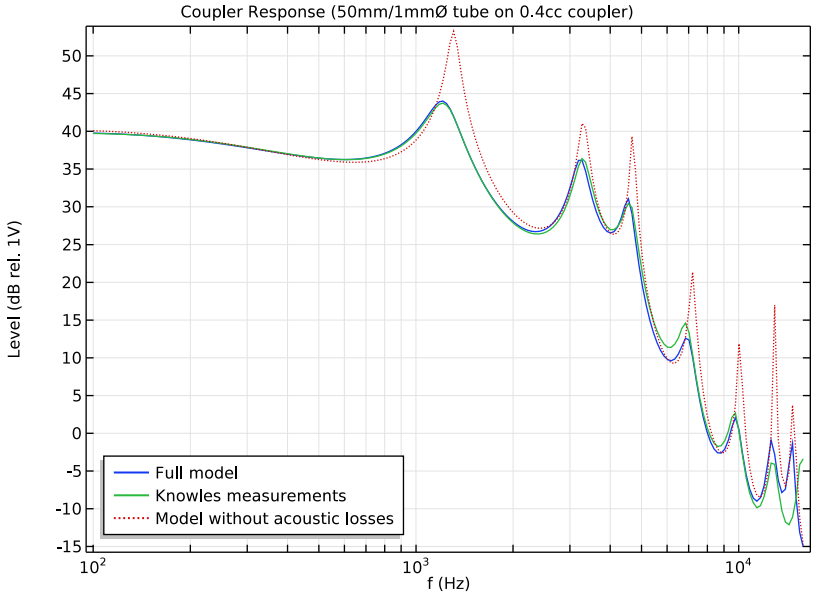


Figure 3: The microphone response comparing the model including the thermal and viscous losses via the narrow region acoustics feature (blue curve), the model without acoustics losses (red dotted curve), and measurements (green curve). Measurements data provided by Knowles, IL, USA.

The frequency dependency of the electric input impedance (real and imaginary part) of the transducer are depicted in Figure 4. The results are compared to measurements and show good agreement.

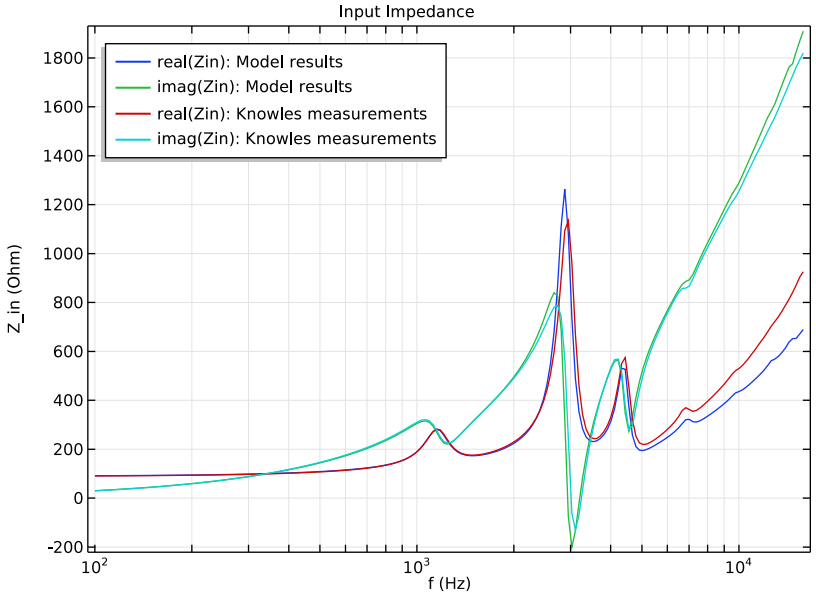


Figure 4: The electric input impedance (real and imaginary part) as a function of the frequency comparing model results (blue and green curves) and measurements (red and cyan curves). Measurements data provided by Knowles, IL, USA.

In Figure 5 and Figure 6, the pressure and sound pressure level distribution inside the tube and coupler system are depicted at three different frequencies (1200, 3200, and 4600 Hz). The evaluated frequencies correspond to the three first peaks in the response. They correspond to the quarter, half, and three quarter wave resonances of the tube-coupler system, respectively.

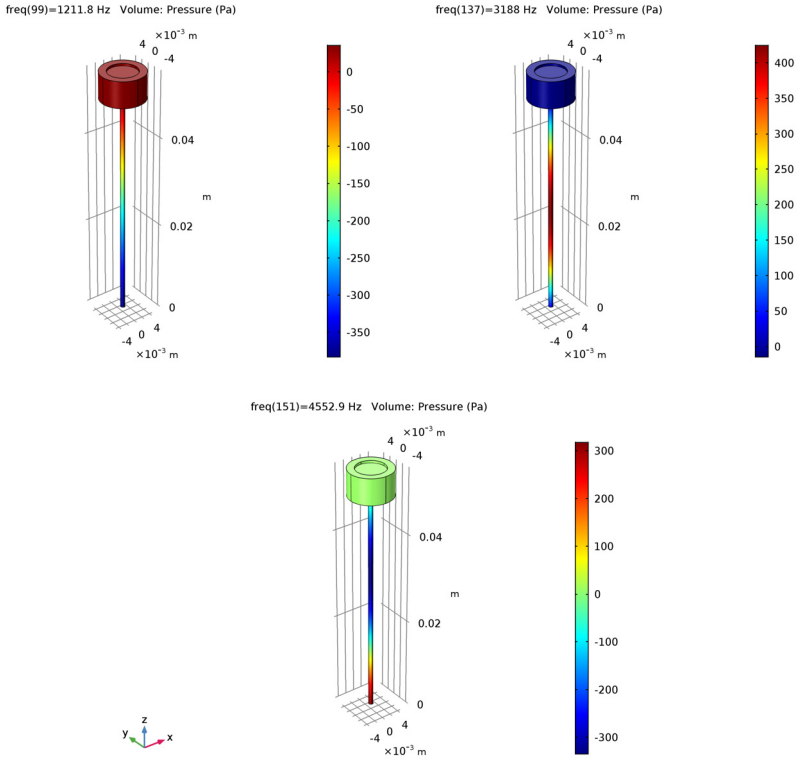


Figure 5: Pressure distribution (real part of the pressure) at f = 1200, 3200, and 4600 Hz.

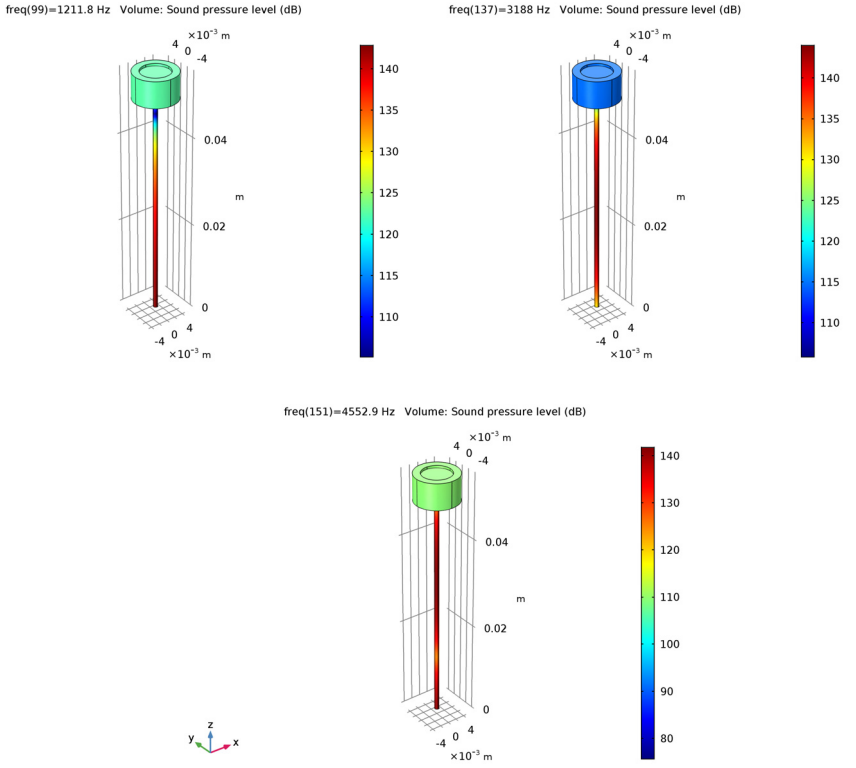


Figure 6: Sound pressure level at f = 1200, 3200, and 4600 Hz

Notes About the COMSOL Implementation

- In this model the viscous and thermal losses associated with the acoustic boundary layer in the narrow tube are modeled using the Narrow Region Acoustics feature in Pressure acoustics. For long structures of constant cross section, the losses are exact when compared to a full Thermoacoustics model. The computational cost is low compared to a full Thermoacoustics model. However, for complex geometrical structures Thermoacoustics should be used. Note also that the losses associated with the impedance jump from the narrow tube to the coupler are not included.
- The Electrical Circuit interface uses electrical units. Conversion from electrical to acoustic lumped units are performed by multiplication with the necessary units. For

example, in the voltage source representing the acoustic pressure at the transducer inlet the expression is `Pin[V/Pa]`. Resulting in correct equivalent electric units volts.

- In the lumped Spice model of the receiver, the effects of variation in the skin depth of eddy currents in the steel armature is approximated by a semi-capacitor, a special component with a complex admittance proportional to the square root of $i\omega$. In the imported Spice net-list, the value of this component, here a resistor, is temporarily set to 1, using:

  ```
  RKarm N0025 N0015 1
  ```

  Then the correct value for this component is entered manually, as a formula, to fit the COMSOL notation:

  ```
  1/(4.85e-11[1/ohm]*sqrt(i*2*pi*freq[1/Hz]))
  ```

  The component is, in the unmodified Knowles Spice net-lists, included as an advance voltage-controlled current source as:

  ```
  GKarm N0025 N0015 laplace {V(N0025,N0015)}={4.85e-11*sqrt(s)}
  ```

  This notation is not supported by the Spice import functionality of COMSOL.

References

1. *Generic 711 Coupler: An Occluded Ear-Canal Simulator* model in the Application Library of the Acoustics Module: `Acoustics_Module/Electroacoustic_Transducers/generic_711_coupler`.

2. J. Jensen, F. T. Agerkvist and J. M. Hart, "Nonlinear Time-Domain Modeling of Balanced Armature Receivers," *J. Audio Eng. Soc.*, vol. 59,pp. 91–101, 2011.

3. *Lumped Loudspeaker Driver* model in the Application Library of the Acoustics Module: `Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver`.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/lumped_receiver_04cc`

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 **AC/DC>Electrical Circuit (cir)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.
8. Click **Done**.

ROOT

First, import the model parameters, set up an interpolation function for the measurement data, import variables used in the model, and import the geometry. The instructions to the geometry can be found in the appendix at the end of this document.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `lumped_receiver_04cc_parameters.txt`.

Interpolation 1 (int1)

1. On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
2. In the **Settings** window for **Interpolation**, locate the **Definition** section.
3. From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `lumped_receiver_04cc_measured_data.txt`.

6 In the **Number of arguments** text field, type `1`.

7 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
`preal`	1
`pimag`	2
`Zreal`	3
`Zimag`	4

8 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

9 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

10 Locate the **Definition** section. Click **Import**.

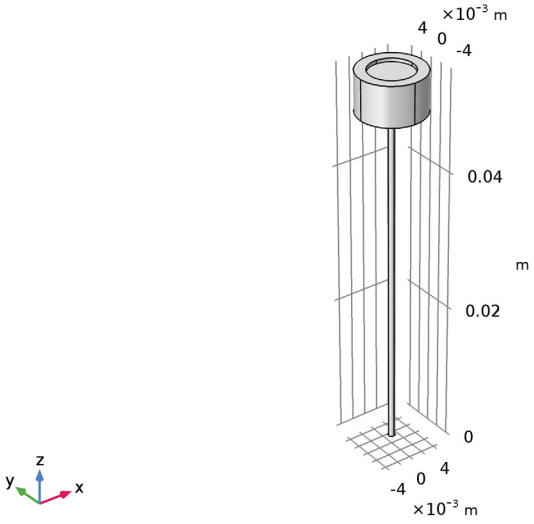
GEOMETRY 1

1 On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `lumped_receiver_04cc_geom_sequence.mph`.

3 On the **Geometry** toolbar, click **Build All**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.



DEFINITIONS

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `lumped_receiver_04cc_variables.txt`.

Once the geometry has been imported you set up coupling operators that refer to specific geometry boundaries.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_in` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

5 Select Boundary 10 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_mic` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 7 only.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component 1**.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

Now, set up the circuit model. It includes the driving voltage source, the sub-circuit for the receiver (imported), and a voltage source that couples the pressure back to the circuit model.

ELECTRICAL CIRCUIT (CIR)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrical Circuit (cir)** and choose **Import SPICE Netlist**.

2 Browse to the model's Application Libraries folder and double-click the file `lumped_receiver_04cc_ED23146.cir`.

Subcircuit Definition ED23146

1 In the **Settings** window for **Subcircuit Definition**, locate the **Node Connections** section.

2 In the table, enter the following settings:

Node names
P1
N1
P2
N2

The component RKARM needs special attention by manual editing as it is a non-trivial function of frequency. The imported Spice net-list is courtesy of Knowles, IL, USA.

Resistor RKARM

1 In the **Model Builder** window, expand the **Subcircuit Definition ED23146** node, then click **Resistor RKARM**.

2 In the **Settings** window for **Resistor**, locate the **Device Parameters** section.

3 In the R text field, type `1/(4.85e-11[1/ohm]*sqrt(i*2*pi*freq[1/Hz]))`.

Voltage Source V1

1 In the **Model Builder** window, right-click **Electrical Circuit (cir)** and choose **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	p1
n	0

4 Locate the **Device Parameters** section. In the V_{src} text field, type `V0`.

Subcircuit Instance X1

1 Right-click **Electrical Circuit (cir)** and choose **Subcircuit Instance**.

2 In the **Settings** window for **Subcircuit Instance**, locate the **Node Connections** section.

3 From the **Name of subcircuit link** list, choose **Subcircuit Definition ED23146**.

4 In the table, enter the following settings:

Local node names	Node names
P1	p1
N1	0
P2	p2
N2	0

5 Right-click **Electrical Circuit (cir)** and choose **Voltage Source**.

Set up the voltage source that couples the pressure. The variable Pin is defined under variables and calculates the average pressure at the tube inlet.

Voltage Source V2

1 In the **Settings** window for **Voltage Source**, locate the **Node Connections** section.

2 In the table, enter the following settings:

Label	Node names
p	p2
n	0

3 Locate the **Device Parameters** section. In the V_{src} text field, type `Pin[V/Pa]`.

The electric circuit has now been set up. Proceed and set up the pressure acoustics physics.

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 **Model Inputs** section.

3 In the T text field, type `T0`.

4 In the p_A text field, type `p0`.

5 In the **Model Builder** window, click **Pressure Acoustics, Frequency Domain (acpr)**.

Narrow Region Acoustics 1

1 On the **Physics** toolbar, click **Domains** and choose **Narrow Region Acoustics**.

2 Select Domain 2 only.

3 In the **Settings** window for **Narrow Region Acoustics**, locate the **Model Input** section.

4 In the T text field, type `T0`.

5 In the p_A text field, type `p0`.

6 Locate the **Narrow Region Acoustics** section. From the **Duct type** list, choose **Circular duct**.

7 In the a text field, type `a`.

Narrow Region Acoustics 2

1 On the **Physics** toolbar, click **Domains** and choose **Narrow Region Acoustics**.

2 Select Domain 1 only.

3 In the **Settings** window for **Narrow Region Acoustics**, locate the **Model Input** section.

4 In the T text field, type `T0`.

5 In the p_A text field, type `p0`.

6 Locate the **Narrow Region Acoustics** section. From the **Duct type** list, choose **Circular duct**.

7 In the a text field, type `a_cpl`.

Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Acceleration**.

The normal acceleration couples the circuit model to the finite element domain. The variable An is defined under variables and represents the average normal acceleration at the tube inlet, see Equation 1.

2 Select Boundary 10 only.

3 In the **Settings** window for **Normal Acceleration**, locate the **Normal Acceleration** section.

4 In the a_n text field, type `An`.

Impedance 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Impedance**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Impedance**, locate the **Impedance** section.

4 From the **Impedance model** list, choose **RCL**.

5 In the R_{ac} text field, type `Rmic`.

6 In the C_{ac} text field, type `Cmic`.

7 In the L_{ac} text field, type `Lmic`.

Now, mesh the geometry and then solve the model

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 3 and 11 only.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `3*a`.

5 In the **Minimum element size** text field, type `0.1[mm]`.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 11 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 `a`.

8 Click **Build Selected**.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

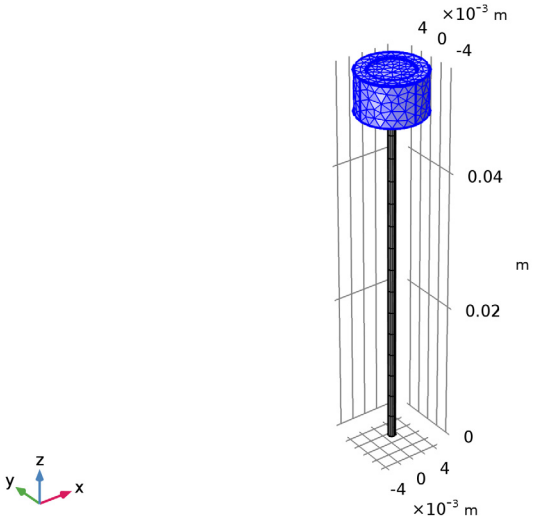
3 In the **Number of elements** text field, type `15`.

4 Click **Build Selected**.

Free Tetrahedral 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.



Solve the model. First, solve the full model including the equivalent acoustic loss model. Secondly, solve the model without the losses. Do this by deactivating the Narrow Region Acoustics domain features in the second study.

STUDY 1

1 In the **Settings** window for **Study**, type `Study 1 - Narrow Region` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 1 - NARROW REGION

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - Narrow Region** 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 `10^{range(2,2.2/199,4.2)}`.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1 - Narrow Region>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Study 1 - Narrow Region>Solver Configurations>Solution 1 (sol1)> Stationary Solver 1** and choose **Fully Coupled**.

5 On the **Study** toolbar, click **Compute**.

ADD STUDY

1 On the **Study** 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - Lossless Acoustics` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 2 - LOSSLESS ACOUSTICS

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - Lossless Acoustics** 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 `10^{range(2,2.2/199,4.2)}`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Pressure Acoustics, Frequency Domain (acpr)>Narrow Region Acoustics 1**.

6 Click **Disable**.

7 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Pressure Acoustics, Frequency Domain (acpr)>Narrow Region Acoustics 2**.

8 Click **Disable**.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2 - Lossless Acoustics> Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node.

4 Right-click **Study 2 - Lossless Acoustics>Solver Configurations>Solution 2 (sol2)> Stationary Solver 1** and choose **Fully Coupled**.

5 On the **Study** toolbar, click **Compute**.

RESULTS

1D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Microphone Response` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Coupler Response (50mm/1mmØ tube on 0.4cc coupler)`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Level (dB rel. 1V)`.

9 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 Right-click **Microphone Response** 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
`20*log10(abs(intop_mic(p)/ intop_mic(1))/V0)`		`Full model`
`20*log10(abs(preal(freq)+i* pimag(freq)))`		`Knowles measurements`

Global 2

1 In the **Model Builder** window, under **Results** right-click **Microphone Response** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - Lossless Acoustics/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`20*log10(abs(intop_mic(p)/intop_mic(1))/V0)`		`Model without acoustic losses`

5 On the **Microphone Response** toolbar, click **Plot**.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

7 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

The modeled and measured microphone response are depicted in Figure 3.

1D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Electric Input Impedance` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Input Impedance`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Z_in (Ohm)`.

9 Click to collapse the **Legend** section. Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Electric Input Impedance** 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
`real(-cir.V1_v/cir.V1_i)`	Ω	`real(Zin): Model results`
`imag(-cir.V1_v/cir.V1_i)`	Ω	`imag(Zin): Model results`
`Zreal(freq)`		`real(Zin): Knowles measurements`
`Zimag(freq)`		`imag(Zin): Knowles measurements`

4 On the **Electric Input Impedance** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

The electric input impedance graph is depicted in Figure 4.

3D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Pressure` in the **Label** text field.

Volume 1

1 Right-click **Pressure** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 In the **Expression** text field, type `p`.

4 On the **Pressure** toolbar, click **Plot**.

Change the evaluation frequency and plot the pressure at the desired frequencies. The pressure is depicted in Figure 5 at the first three resonance peaks.

3D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level` in the **Label** text field.

Volume 1

1 Right-click **Sound Pressure Level** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

3 On the **Sound Pressure Level** toolbar, click **Plot**.

Change the evaluation frequency and plot the sound pressure level at the desired frequencies. The sound pressure level is depicted in Figure 6 at the first three resonance peaks.

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

Cylinder 1 (cyl1)

On the **Geometry** toolbar, click **Cylinder**.

GEOMETRY 1

Cylinder 1 (cyl1)

1. In the **Settings** window for **Cylinder**, locate the **Selections of Resulting Entities** section.
2. Select the **Resulting objects selection** check box.
3. Locate the **Size and Shape** section. In the **Radius** text field, type `0.5[mm]`.
4. In the **Height** text field, type `49[mm]`.

Cylinder 2 (cyl2)

1. On the **Geometry** toolbar, click **Cylinder**.
2. In the **Settings** window for **Cylinder**, locate the **Selections of Resulting Entities** section.
3. Select the **Resulting objects selection** check box.
4. Locate the **Size and Shape** section. In the **Radius** text field, type `4.72[mm]`.
5. In the **Height** text field, type `5.7[mm]`.
6. Locate the **Position** section. In the **z** text field, type `49[mm]`.

Cylinder 3 (cyl3)

1. On the **Geometry** toolbar, click **Cylinder**.
2. In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `3.175[mm]`.
4. In the **Height** text field, type `0.5[mm]`.
5. Locate the **Position** section. In the **z** text field, type `54.2[mm]`.

Difference 1 (dif1)

1. On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2. Select the object **cyl2** only.
3. In the **Settings** window for **Difference**, locate the **Difference** section.
4. Find the **Objects to subtract** subsection. Select the **Active** toggle button.
5. Select the object **cyl3** only.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Explicit Selection 1 (sel1)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Normal Acceleration` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 10 only.

Explicit Selection 2 (sel2)

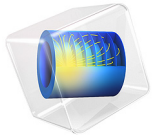
1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Impedance` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 7 only.

5 On the **Geometry** toolbar, click **Build All**.



Nonlinear Acoustics—Modeling of the ID Westervelt Equation

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Although linear acoustics does an outstanding job of explaining most acoustical phenomena, there are an increasing number of applications that require abandoning the small-signal assumption and applying the wave equation for finite-amplitude sound or nonlinear acoustics. For example, the effects of finite amplitude propagation can be seen in almost every medical use of ultrasound. The use of high intensity ultrasound to induce tissue heating for cancer therapy or bleeding control, and the use of shock waves in extracorporeal and endoscopic lithotripsy of kidney stones and gallstones provide additional examples. Even in the field of diagnostic ultrasound, nonlinear acoustics is of interest because the higher harmonics emerged during wave propagation can be exploited to produce a better image quality.

The nonlinear phenomenon of wave distortion is both a local effect and a cumulative effect due to variation of propagation speed over the waveform, which causes distortion that accumulates with propagation distance. The local effect is usually small compared to cumulative distortion and can be neglected once the propagation distance becomes much greater than a wavelength; see Ref. 1. Therefore, a transient analysis is necessary to model the cumulative distortion along with the wave propagation.

This example shows how to model nonlinear propagation of finite-amplitude acoustic waves in fluids using the Acoustics Module. The nonlinear term is manually added into the linear wave equation, which makes the problem identical to solving a 2nd-order Westervelt equation (here in the lossless limit). The Westervelt equation is an approximation of the full 2nd-order wave equation when cumulative nonlinear effects dominate local nonlinear effects; see Ref. 1. The current model simulates a finite-amplitude wave propagating in 1D in a loss-less fluid. The computed numerical solution is compared to an analytical solution.

Model Definition

The wave equation solved in the model is identical to the 2nd-order Westervelt equation in the limit of no losses (no diffusive terms). The full Westervelt equation is given by

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} - \nabla \cdot \left(\frac{1}{\rho}\left(\nabla p + \frac{\delta}{\rho c^2}\frac{\partial(\nabla p)}{\partial t}\right)\right) = \frac{\beta}{\rho^2 c^4}\frac{\partial^2 p^2}{\partial t^2} \tag{1}$$

where p is the acoustic pressure, ρ and c are density and speed of sound, β is the coefficient of nonlinearity, and δ is a measure of the acoustic diffusivity (see Ref. 2). In this model δ is set equal to zero in order to compare to an analytical solution.

The nonlinear term on the right-hand side of Equation 1 can be added to the wave equation used for transient pressure acoustics in COMSOL Multiphysics by setting it equal to a monopole domain source term Q_m.

The analytical solution used for comparison is known as the Fubini solution, which is valid for the spatial region near the source (Ref. 1). The solution is an infinite series given by

$$p(x,t) = p_0 \sum_{n=1}^{\infty} B_n(x) \sin\left(n\omega\left(t - \frac{x}{c}\right)\right) \quad \text{where} \quad B_n(x) = \frac{2}{n\sigma} J_n(n\sigma) \tag{2}$$

where J_n is the Bessel function of the first kind of order n, p_0 is the wave amplitude, and σ is the dimensionless spatial coordinate

$$\sigma = \frac{x}{x_{sh}} \quad \text{where} \quad x_{sh} = \frac{1}{\beta\varepsilon k}$$

Here β is the coefficient of nonlinearity, $k = \omega/c$ is the wave number, $\varepsilon = u_0/c$ is the peak particle velocity Mach number, and x_{sh} is the shock formation distance. The solution given by Equation 2 is only valid for $\sigma \leq 1$.

Table 1 shows the material properties and some critical parameters used in the simulation. Given those numbers, the shock formation distance for the current model is about 0.52 m. Therefore, the simulation domain in this model is x = 0 to 0.5 m, as shown in Figure 1. A sinusoidal pressure source of amplitude p_0 is applied at x = 0, and the Plane Wave Radiation boundary condition is applied at x = 0.5 m to model an outgoing wave.

TABLE 1: MATERIAL PROPERTIES AND SOME CRITICAL MODEL PARAMETERS.

NAME	VALUE	DESCRIPTION
ρ	999.6 kg/m^3	Density at 20 °C and 1 atm
c	1481.44 m/s	Speed of sound at 20 °C and 1 atm
δ	0 Np/m/MHz	Attenuation coefficient
β	10	Coefficient of nonlinearity
f	0.1 MHz	Source driving frequency
p_0	1 MPa	Source pressure amplitude
dx_0	2.4 to 0.8 mm	Mesh element size (decreasing along propagation)
dt	< 8.3·10^{-8} s	Time step

Figure 1: 1D geometry of the model, with source located at x = 0 and shock formation zone a bit further than x = L =0.5 m.

Results and Discussion

The increasing distortion of the waveform when the wave travels away from the acoustic source is illustrated in Figure 2. The nonlinear effect is apparent by comparing the solution with the linear analytical solution. The waveform distortion is caused by the dependence of propagation speed on the pressure or particle velocity. The peaks of the wave travel faster than the troughs, and the waveform gradually turns into a sawtooth-like shape. The distorted waveform obtained from the model (blue solid line) agrees well with the

nonlinear analytical solution (red solid line), as shown in Figure 2.

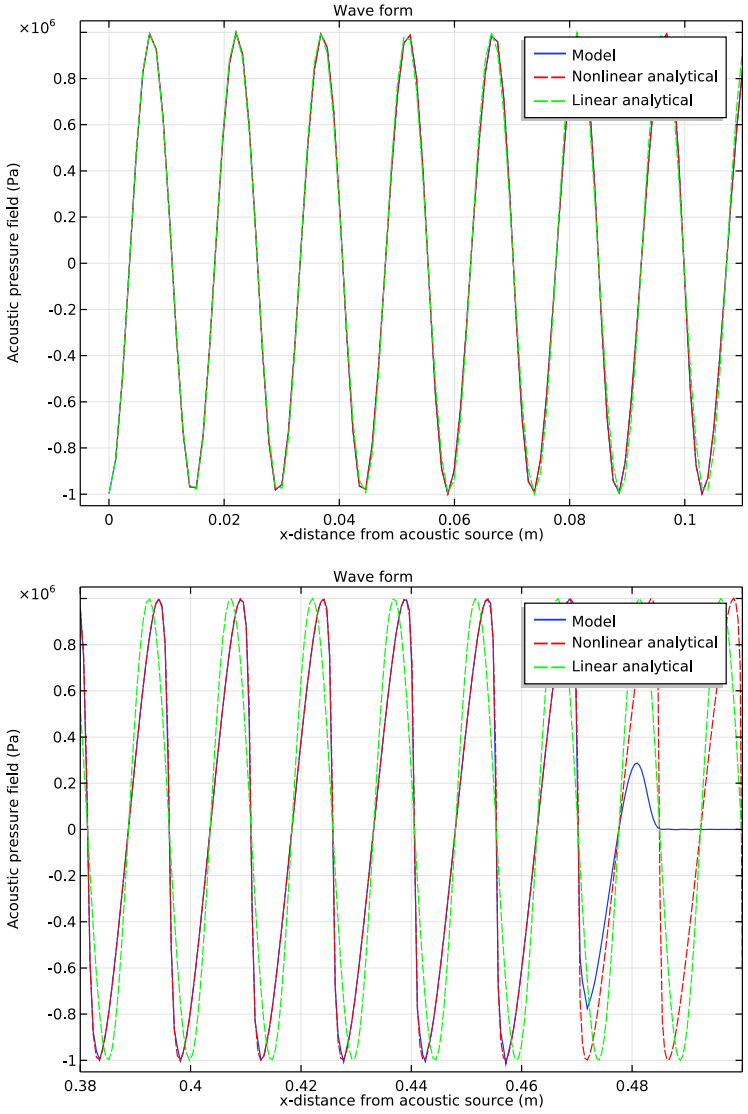


Figure 2: Comparison of model solution (blue solid line) with nonlinear analytic solution (red dashed line) and with the linear analytical solution (green dashed line). The waveforms shown are those close to the Acoustic source (top) and before shock formation distance (bottom). The numerical solution agrees well with the analytical solution of the nonlinear problem.

The distortion of the waveform results in the generation of higher harmonic components. The further the wave travels, the more energy is transferred to the higher harmonic components from the fundamental frequency of the harmonic source signal. This effect is demonstrated in the plots of Figure 3 through Figure 5. The plots compare the model solution (blue line) with the nonlinear analytical solution (red dashed line) for both waveform and frequency spectrum at three locations with different displacements from the source ($x = 0.1 \cdot L$, $0.5 \cdot L$, and $0.8 \cdot L$, respectively). The Fourier transforms used to determine the spectrum are applied in the time regime after the wave arrives at those specific locations.

At $x = 0.1 \cdot L$ in Figure 3, the 2nd harmonic components start to show up; at $x = 0.5 \cdot L$ in Figure 4, the first three harmonics are readily seen; and at $x = 0.8 \cdot L$ in Figure 5, up to ten harmonics appear in the frequency spectrum. These results clearly show how the wave deforms when traveling away from the source and how the acoustic energy is pumped into higher harmonics from the fundamental frequency.

This model intentionally does not include energy dissipation because it is intended for comparison with the analytical solution. However, energy loss can be easily added in COMSOL by choosing a lossy fluid model. For thermally conducting and viscous fluids, the acoustic absorption coefficient usually increases with frequency so the higher harmonic components damp out more rapidly than the lower ones. Therefore, when taking viscosity and thermal conduction into account, the energy dissipation generates a counter effect which tends to weaken the nonlinear effect.

Note: A description of the different lossy fluid models available for Pressure Acoustics is given in the documentation; see the *Acoustics Module User's Guide* (which you can locate by first choosing **Help>Documentation** and then searching for *fluid models*).

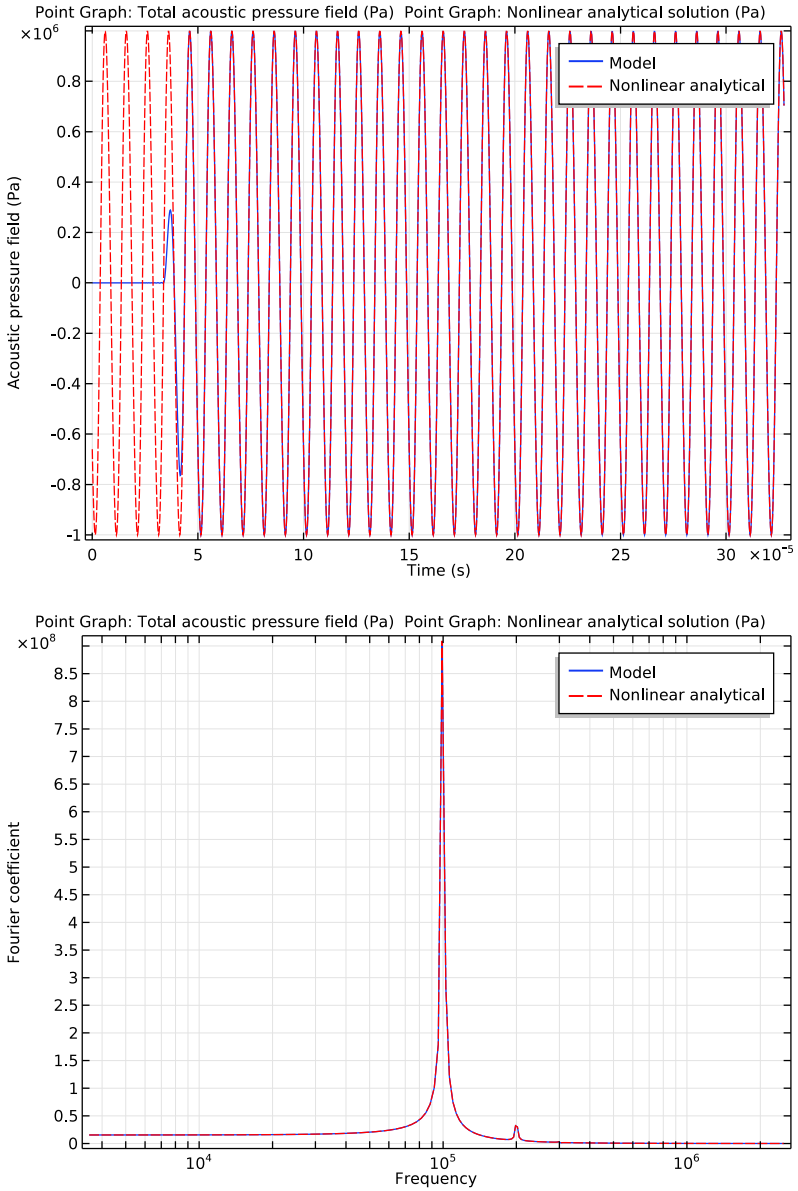


Figure 3: Comparison of model solution (blue solid line) with nonlinear analytical solution (red dashed line) at x = 0.1·L. The top plot shows the pressure profile and the bottom plot the frequency spectrum.

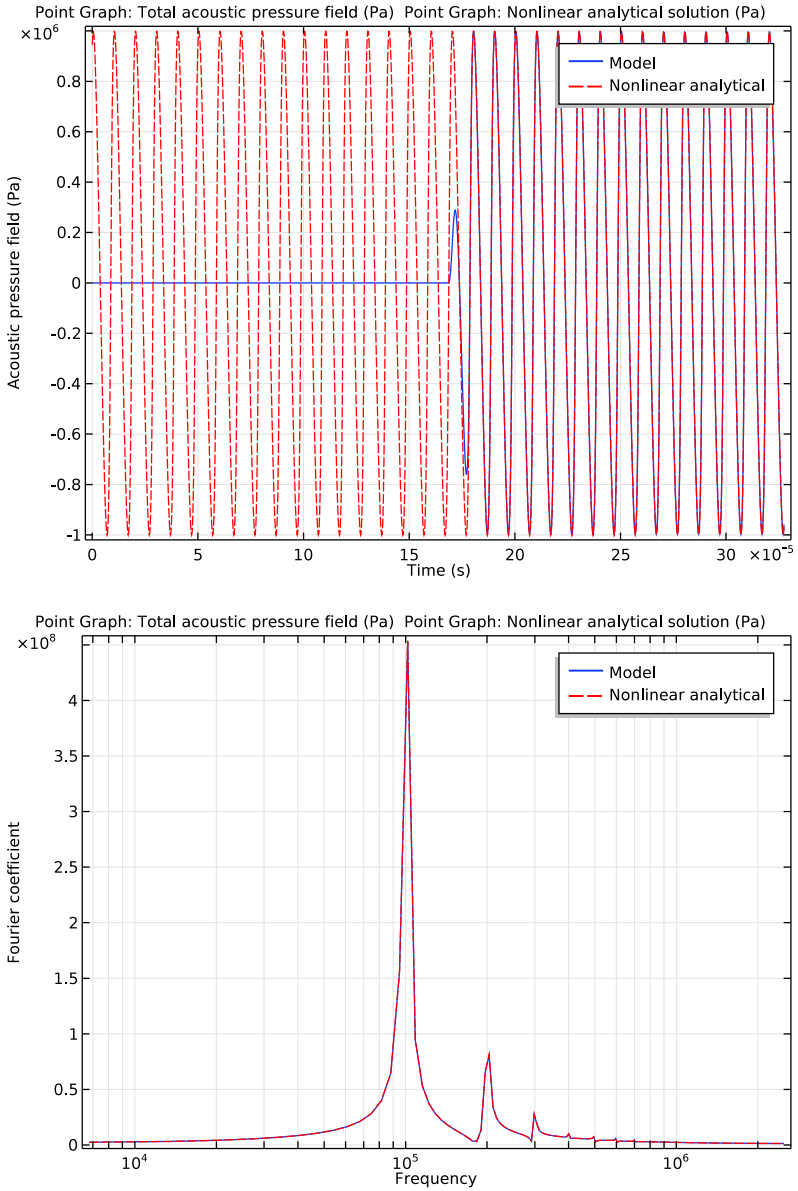


Figure 4: Comparison of model solution (blue solid line) with nonlinear analytical solution (red dashed line) at x = 0.5·L. L = 0.5 m is the domain length. The top plot shows the pressure profile and the bottom plot the frequency spectrum.

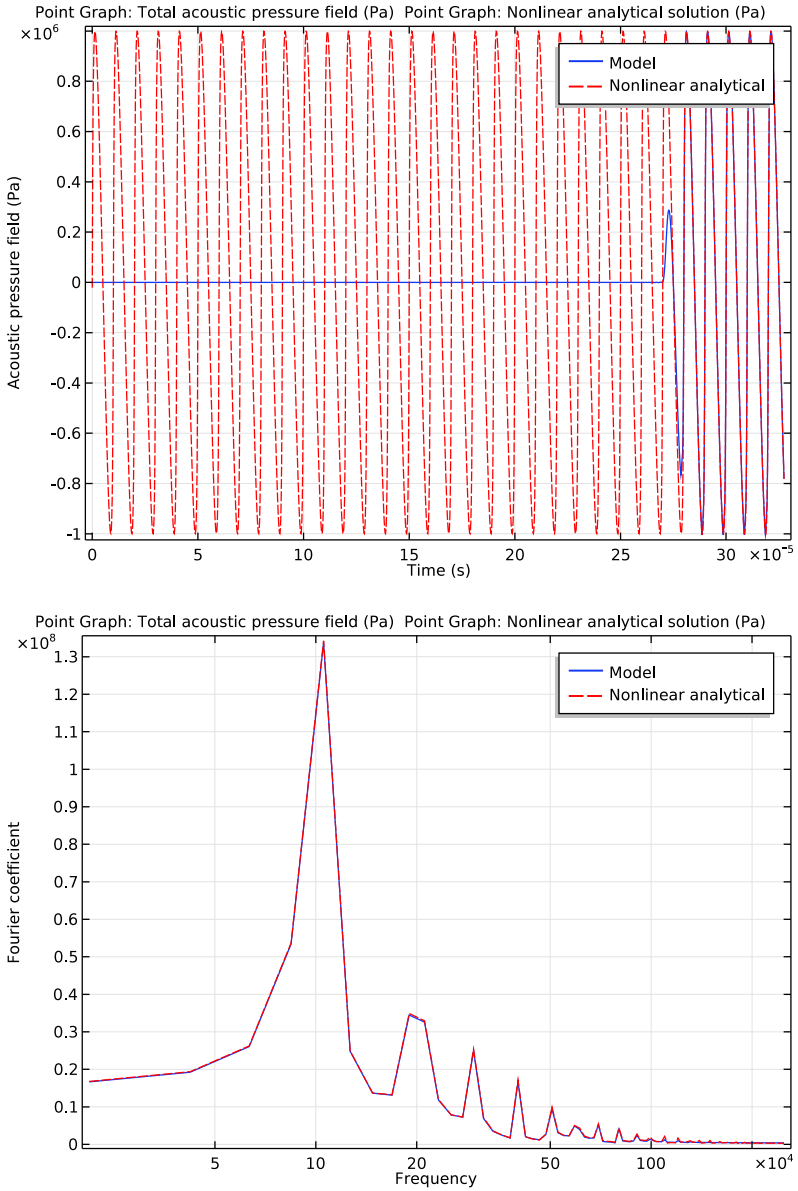


Figure 5: Comparison of model solution (blue solid line) with nonlinear analytical solution (red dashed line) at x = 0.8·L. L = 0.5 m is the domain length. The top plot shows the pressure profile and the bottom plot the frequency spectrum.

MESH

To be able to resolve higher harmonics along the wave propagation direction, use a structured mesh with decreasing element size along the x direction. Use the Edge Distribution mesh feature to specify a mesh with exponentially decreasing element size along the x-axis. Achieving an optimal mesh may require analyzing the mesh. Figure 6 shows how the mesh size changes with the distance the wave travels (blue curve). The graph also shows the mesh size necessary to resolve the first harmonic of the source (green curve), and the mesh size necessary to resolve the Nth harmonic, here $N = 4$.

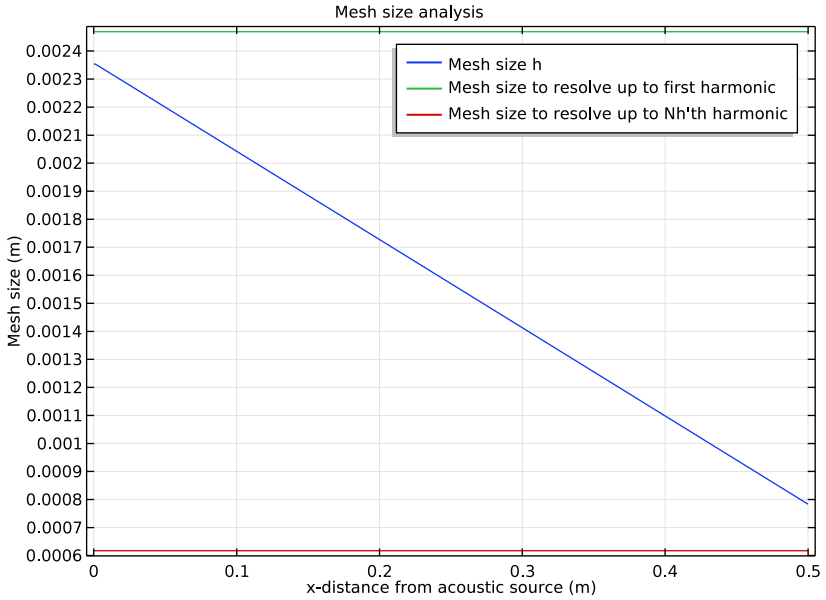


Figure 6: Mesh analysis representing the mesh size h as function of location along the x-axis (blue curve), the mesh size necessary to resolve the first harmonic of the source (green curve), and the mesh size necessary to resolve the Nth harmonic, here N = 4.

ELEMENTS

To accurately resolve the acoustic pressure, use at least 2nd-order (quadratic) elements. The current model uses quartic (4th-order) elements for the pressure, which has been proven to give more accurate results compared with 2nd-order elements while keeping the number of degrees of freedom (DOFs) the same.

TIME STEPPING

It has also been proven that using the time-stepping method BDF with the maximum BDF order set to 5 (the default) seems to give the best result by comparing the numerical solution to the analytical solution.

The relationship between mesh size and time step length is known as the CFL number: $CFL_m = c \cdot dt/dx$, where dt is the time step, dx is the mesh size, and m is the element order. In practice, a CFL_2 number of 0.2 proves to be near optimal (for quadratic elements). To speed up the solving of the model, CFL_2 is chosen even though quartic elements are used. The maximum time step used for this model is $8.3 \cdot 10\text{-}8$ s based on the smallest mesh size used for resolving waves up to the 4th harmonics. For quartic elements one must, in principle, use $CFL_4 = 0.05$ for the temporal resolution to be optimal compared to the spatial resolution.

References

1. M.F. Hamilton and D.T. Blackstock, eds., *Nonlinear Acoustics*, Academic Press, San Diego, CA, 1998.

2. D.T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.

Application Library path: `Acoustics_Module/Nonlinear_Acoustics/nonlinear_acoustics_westervelt_1d`

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 **1D**.
2. In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.
3. Click **Add**.
4. Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Load the parameters used in the model from a file. Some of the parameters are presented in Figure 1.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `nonlinear_acoustics_westervelt_1d_parameters.txt`.

Now, define the scaled Bessel functions B_n used in the nonlinear wave solution defined in Equation 2.

Analytic 1 (an1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `Bn` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `P0*(2/(n*(x/Xsh)))*besselj(n,n*x/Xsh)*sin(n*omega*(t-x/c))`.

4 In the **Arguments** text field, type `n, t, x`.

5 Locate the **Units** section. In the **Arguments** text field, type `1, s, m`.

6 In the **Function** text field, type `Pa`.

To improve convergence, define a ramp function that gently increases the amplitude of the pressure source.

Ramp 1 (rm1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Ramp**.

2 In the **Settings** window for **Ramp**, locate the **Parameters** section.

3 In the **Slope** text field, type `1/T`.

4 Select the **Cutoff** check box.

5 Click to expand the **Smoothing** section. Select the **Size of transition zone at start** check box.

6 In the associated text field, type `0.2*T`.

7 Select the **Size of transition zone at cutoff** check box.

8 In the associated text field, type `0.2*T`.

Define variables for the linear and nonlinear solutions of the problem. Use the `sum()` operator to sum up the scaled Bessel functions given in Equation 2, here 100 terms are used.

DEFINITIONS

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
`linear_solution`	`P0*sin(omega*t-k*x)`	Pa	`Linear analytical solution`
`nonlinear_solution`	`sum(Bn(n,t,x),n,1,100)`	Pa	`Nonlinear analytical solution`

GEOMETRY 1

Interval 1 (i1)

1 On the **Geometry** toolbar, click **Interval**.

2 In the **Settings** window for **Interval**, locate the **Interval** section.

3 In the **Right endpoint** text field, type `L`.

4 Click **Build All Objects**.

The geometry should look like the one presented in Figure 1.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient (actd)**.

3 In the **Settings** window for **Pressure Acoustics, Transient**, click to expand the **Discretization** section.

4 From the **Pressure** list, choose **Quartic Lagrange**.

The previous steps change the default quadratic shape functions to fourth order shape functions. The reason for this is discussed in the Notes About the COMSOL Implementation section.

Transient Pressure Acoustics Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Transient (actd)** click **Transient Pressure Acoustics Model 1**.

2 In the **Settings** window for **Transient Pressure Acoustics Model**, locate the **Transient Pressure Acoustics Model** section.

3 From the c list, choose **User defined**. In the associated text field, type `c`.

4 From the ρ list, choose **User defined**. In the associated text field, type `rho`.

Pressure 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Pressure**, locate the **Pressure** section.

4 In the p_0 text field, type `P0*sin(omega*t)*rm1(t[1/s])`.

Plane Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 Select Boundary 2 only.

Now, set up the additional right-hand side terms defined in Equation 1 to change the classical linear wave equation into the 2nd-order Westervelt equation. Add a domain monopole source that contains the nonlinear domain term.

Monopole Domain Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Monopole Domain Source**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

3 In the **Settings** window for **Monopole Domain Source**, locate the **Monopole Domain Source** section.

4 In the Q_m text field, type `(beta/(actd.rho^2*actd.c^4))*d(d(p*p,t),t)`.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edge**.

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 `dx`.

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Edge 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 From the **Distribution properties** list, choose **Predefined distribution type**.

4 In the **Number of elements** text field, type `350`.

5 In the **Element ratio** text field, type `3`.

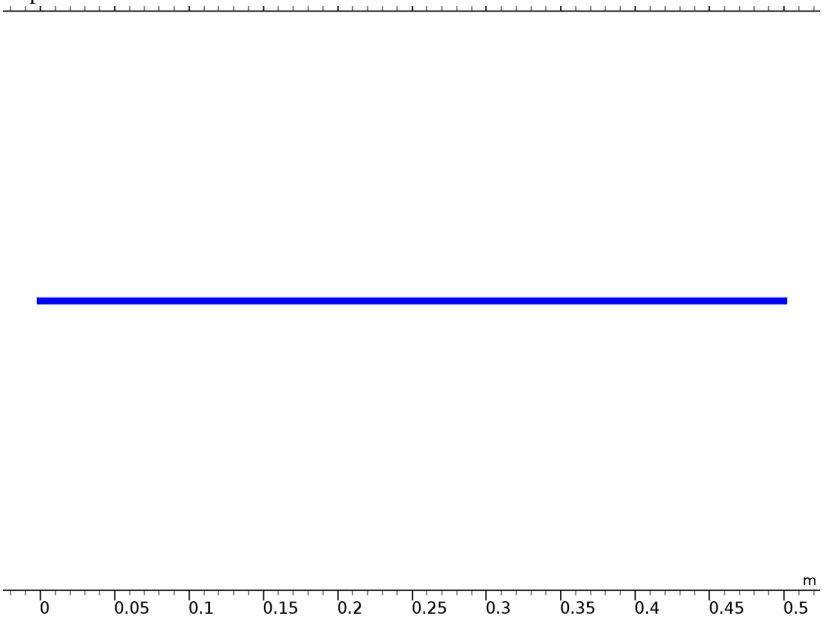
6 From the **Distribution method** list, choose **Geometric sequence**.

7 Select the **Reverse direction** check box.

8 Click **Build All**.

The mesh should look like that in the figure below. In order to better study the mesh, use the **Zoom Box** and **Zoom Extents** tools to zoom in on the different parts of the mesh. A plot representing the size distribution of the mesh elements is given in Figure 6.

Obtaining the desired mesh behavior may require a few iterations when selecting mesh parameters.



STUDY 1

Step 1: Time Dependent

1. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
2. In the **Times** text field, type `range(0,T/50,Tend)`.
3. From the **Tolerance** list, choose **User controlled**.
4. In the **Relative tolerance** text field, type `0.001`.

Finally, set up the transient solver according to the discussion given in the section Notes About the COMSOL Implementation. Use the BDF time stepping method and set the maximal time steps to be `dt`. These settings override the automatic settings that can be edited in the **Transient Solver Settings** section at the physics level. These default settings are not tuned to nonlinear problems.

Solution 1 (sol1)

1. On the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. From the **Method** list, choose **BDF**.

5 From the **Steps taken by solver** list, choose **Strict**.

6 Select the **Initial step** check box.

7 In the associated text field, type `dt/2`.

8 Select the **Maximum step** check box.

9 In the associated text field, type `dt`.

10 Click to expand the **Absolute tolerance** section. Locate the **Absolute Tolerance** section. Clear the **Update scaled absolute tolerance** check box.

11 On the **Study** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (actd)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (actd)**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Time selection** list, choose **Last**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Wave form`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `x-distance from acoustic source (m)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Acoustic pressure field (Pa)`.

Line Graph 2

1 Right-click **Results>Acoustic Pressure (actd)** and choose **Line Graph**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `nonlinear_solution`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

8 From the **Color** list, choose **Red**.

Line Graph 3

1 Right-click **Acoustic Pressure (actd)** and choose **Line Graph**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `linear_solution`.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

8 From the **Color** list, choose **Green**.

Line Graph

1 In the **Model Builder** window, under **Results>Acoustic Pressure (actd)** click **Line Graph**.

2 In the **Settings** window for **Line Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Model

Line Graph 2

1 In the **Model Builder** window, under **Results>Acoustic Pressure (actd)** click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Nonlinear analytical

Line Graph 3

1 In the **Model Builder** window, under **Results>Acoustic Pressure (actd)** click **Line Graph 3**.

2 In the **Settings** window for **Line Graph**, locate the **Legends** section.

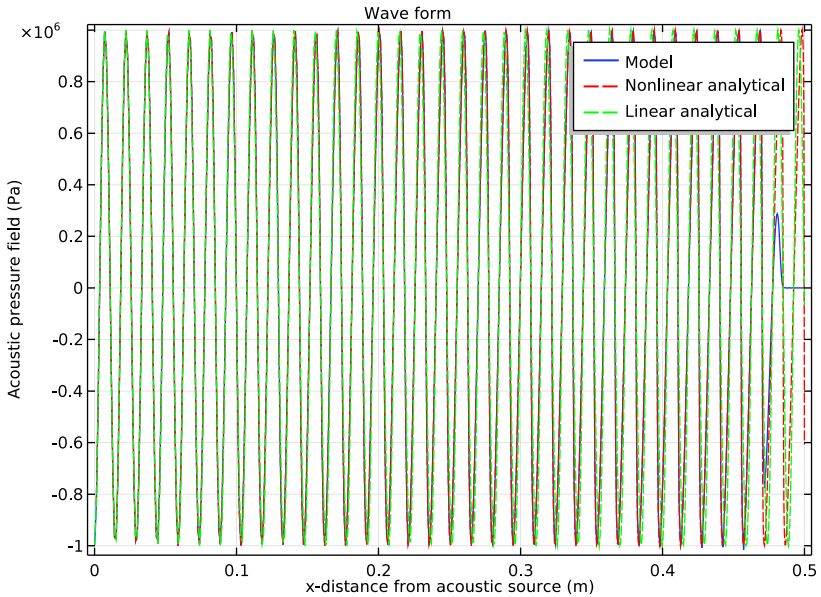
3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Linear analytical

6 On the **Acoustic Pressure (actd)** toolbar, click **Plot**.



You can examine the plot in greater detail by zooming around parts of the plot using the **Zoom Box** tool.

First, zoom in on the part of the plot near to the source at `x = 0`. The plot should look like the top plot in Figure 2. Then, use the **Zoom Extents** tool and then again the **Zoom Box** tool to study the part of the plot near the shock formation at `x = L`. The plot should look like the bottom plot in Figure 2.

Point Graph 1

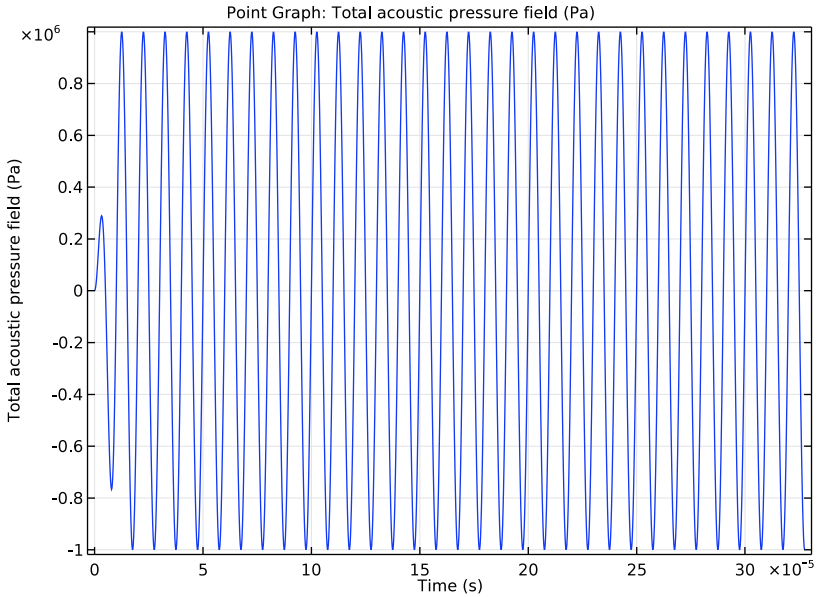
1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Point Graph**.

3 Select Boundary 1 only.

4 On the **1D Plot Group 2** toolbar, click **Plot**.

The figure should look like the plot below.



Cut Point 1D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Cut Point 1D**.

2 In the **Settings** window for **Cut Point 1D**, locate the **Point Data** section.

3 In the **X** text field, type `0.1*L`.

Cut Point 1D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Cut Point 1D**.

2 In the **Settings** window for **Cut Point 1D**, locate the **Point Data** section.

3 In the **X** text field, type `0.5*L`.

Cut Point 1D 3

1 On the **Results** toolbar, click **More Data Sets** and choose **Cut Point 1D**.

2 In the **Settings** window for **Cut Point 1D**, locate the **Point Data** section.

3 In the **X** text field, type `0.8*L`.

1D Plot Group 3

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 1D 1**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Time (s)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Acoustic pressure field (Pa)`.

Point Graph 1

1 Right-click **1D Plot Group 3** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Model`

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 3** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `nonlinear_solution`.

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Red**.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Nonlinear analytical`

9 On the **1D Plot Group 3** toolbar, click **Plot**.

The plot should look like that in Figure 3 (top).

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 1D 1**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, choose from `4.5e-5` to `3.274e-4`, only.

Point Graph 1

1 Right-click **1D Plot Group 4** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Frequency spectrum**.

4 Locate the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`Model`

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `nonlinear_solution`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Frequency spectrum**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Nonlinear analytical`

10 On the **1D Plot Group 4** toolbar, click **Plot**.

11 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

The plot should look like that in Figure 3 (bottom).

1D Plot Group 5

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 3** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 1D 2**.

4 On the **1D Plot Group 5** toolbar, click **Plot**.

The plot should look like that in Figure 4 (top).

1D Plot Group 6

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 1D 2**.

4 In the **Times (s)** list, choose from `1.8e-4` to `3.274e-4`, only.

5 On the **1D Plot Group 6** toolbar, click **Plot**.

The plot should look like that in Figure 4 (bottom).

1D Plot Group 7

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 3** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 1D 3**.

4 On the **1D Plot Group 7** toolbar, click **Plot**.

The plot should look like that in Figure 5 (top).

1D Plot Group 8

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 1D 3**.

4 In the **Times (s)** list, choose from `2.8e-4` to `3.274e-4`, only.

5 On the **1D Plot Group 8** toolbar, click **Plot**.

The plot should look like that in Figure 5 (bottom).

1D Plot Group 9

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Time selection** list, choose **First**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Mesh size analysis`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `x-distance from acoustic source (m)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Mesh size (m)`.

Line Graph 1

1 Right-click **1D Plot Group 9** and choose **Line Graph**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `h`.

5 Locate the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Mesh size h

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 9** and choose **Line Graph**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `dx0`.

5 Locate the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Mesh size to resolve up to first harmonic

Line Graph 3

1 Right-click **1D Plot Group 9** and choose **Line Graph**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `dx`.

5 Locate the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

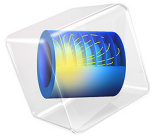
7 In the table, enter the following settings:

Legends
`Mesh size to resolve up to Nh'th harmonic`

8 On the **1D Plot Group 9** toolbar, click **Plot**.

The plot should look like that in Figure 6.

Optionally, give the plot group nodes more descriptive names by editing the **Label**.



Open Pipe

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

In this tutorial model a vibrating piston is mounted inside one end of a cylindrical pipe, while the other end of the pipe opens into an infinite domain. The problem is solved using two different approaches to illustrate their strengths and weaknesses. In the first case, an impedance boundary condition modeling radiation losses into an infinite domain is specified on the pipe end while the acoustics in the domain is not solved explicitly. This is consequently a fast approach. In the second case, a perfectly matched layer condition is applied on the domain. This condition offers a consistent and physically sound method for modeling infinite domains using finite-sized regions. Thus, the domain acoustics is explicitly solved for, but only in a region close to the pipe outlet.

The model considers two different types of pipes: a flanged and an unflanged. In both cases, the impedance-based model is compared to the model using the perfectly matched layer condition.

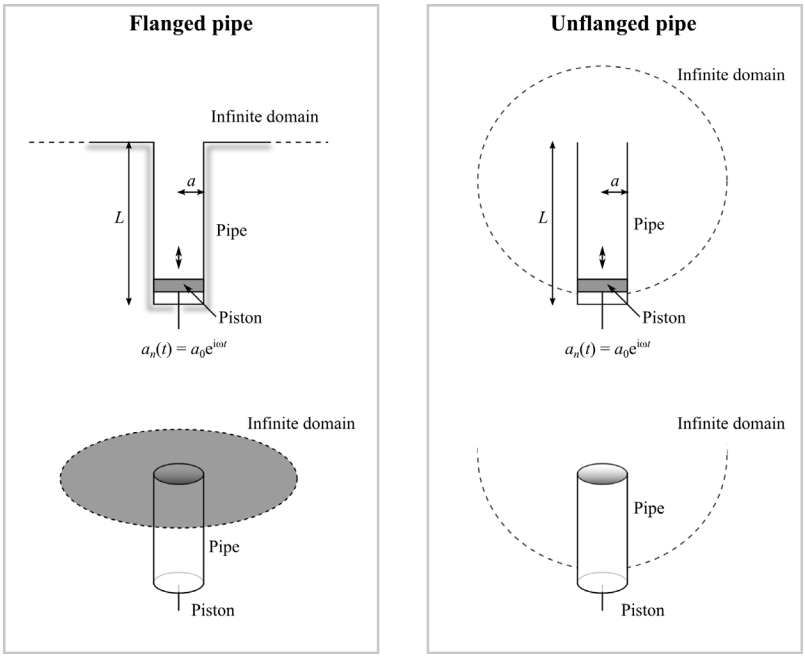


Figure 1: Sketch of the two pipe geometries considered in this model: the flanged pipe (left) and the unflanged pipe (right).

We consider the acoustics of an open pipe with a harmonically oscillating piston mounted in one end. Figure 1 shows the geometry for the two cases considered: the flanged pipe (left) and an unflanged pipe (right). In both cases, the pipe opens into an unbounded domain, and the piston is mounted at the bottom end of the pipe.

The pipe has length L = 1.5 m and radius a = 0.25 m. The piston provides a harmonically oscillating acceleration $a_n(t)=a_0 e^{i\omega t}$ in the axial direction at the end of the pipe, where $\omega = 2\pi f$ [rad/s] is the angular frequency corresponding to the frequency f [Hz]. The model sweeps in the frequency f, from 10 Hz to 1000 Hz. The acoustic medium is air with the density 1.25 kg m^{-3} and the speed of sound c = 343 m/s.

The axial symmetry of both the physics *and* the geometry allows us to exploit the 2D axisymmetry interface.

MODELING THE UNBOUNDED DOMAIN

For either geometry, we model the infinite domain in two separate ways. In the first case, we apply an impedance boundary condition on the pipe outlet, and otherwise ignore the external domain. The impedance is an analytical, geometry-specific model, so the choice of impedance model specifies the geometry of the domain (that is, whether the pipe is flanged or not). This is a computationally efficient method, because we do not need to explicitly model the external domain. However, the analytical expression for the impedance assumes that the incident waves is strictly plane on the outlet. This is always true for low frequencies, but for higher frequencies the geometry can also support non-plane waves. Moreover, the farfield of the sound radiation will in this case start to exhibit directivity (a farfield beam pattern with lobes), which will introduce additional deviations for the simple impedance approach. The cut-off frequency is given by Ref. 1

$$f_c = \frac{\alpha \frac{c}{a}}{2\pi}$$

where $\alpha = 3.832$ is the first root of the Bessel function of the first kind of order 1 and the rest of the parameters are given above. The exact value is $f_c = 836.76$ Hz. Above this frequency we cannot expect the impedance-based model to provide the exact result.

In the second case, we explicitly model the surrounding domain. We use a perfectly matched layer (PML) condition to model the unboundedness of the domain and therefore only model a finite region. The PML condition is an artificial absorbing layer which can be used to truncate the domain while still simulating open boundaries. Compared to the impedance-based model, this approach is more computationally heavy because it also solves for the acoustics in the domain, albeit a truncated one. However, because the domain is modeled explicitly, this approach automatically supports frequencies above the cut-off frequency f_c where non-plane waves can exists.

GOVERNING EQUATION

Being a time-harmonic problem, this model uses the frequency-domain Helmholtz equation for the sound pressure amplitude p

$$\nabla \cdot \left[-\frac{1}{\rho} \nabla p \right] - \frac{\omega^2}{\rho c^2} p = 0 \tag{1}$$

where ρ is the density of the air. The full pressure is given by $p(t) = pe^{i\omega t}$. Comparing to the equation display in the COMSOL window, Equation 1 does not contain the dipole source $\mathbf{q}$, because no such dipole sources are present in the domain.

BOUNDARY CONDITIONS

As already touched upon, we use two different types of boundary conditions to model the unbounded domain in this problem: an impedance condition imposed on the pipe outlet, or an explicit model of the surrounding domain with a perfectly matched layer to simulate the open boundaries. The former case requires the addition of a separate equation on the boundary as detailed below, while the latter does not.

An impedance boundary condition stipulates a relationship between the pressure p and the boundary-normal velocity v at the pipe outlet. It is defined as $Z_i = p/v$ and is typically frequency-dependent. The impedance boundary conditions are invoked by imposing the following equation on the open end of the pipe

$$-\mathbf{n} \cdot \left(\frac{1}{\rho}\nabla p\right) = \frac{i\omega p}{Z_i},$$

where $\mathbf{n}$ is the outward pointing normal vector to the boundary. A complex value of the impedance indicates losses and associated phase-shift between the pressure and the boundary-normal velocity.

In addition to handling the unbounded domain, we also impose conditions to model the oscillating piston as well as the hard side walls of the pipe and the flange (the latter where applicable). The oscillating piston is modeled by imposing a harmonically oscillating acceleration at the pipe inlet

$$\mathbf{n} \cdot \left(\frac{1}{\rho}\nabla p\right) = a_0,$$

where a_0 is the amplitude of the harmonically oscillating acceleration.

For the flanged pipe, the hard walls of the pipe and flange are given by applying the following equation on the appropriate boundaries

$$\mathbf{n} \cdot \left(\frac{1}{\rho}\nabla p\right) = 0. \tag{2}$$

The unflanged pipe furthermore has places with air on both sides of the pipe wall. On these wall segments, the condition above applies to the fluid on both sides,

$$\mathbf{n} \cdot \left(\frac{1}{\rho}\nabla p\right)_1 = 0$$

$$\mathbf{n} \cdot \left(\frac{1}{\rho}\nabla p\right)_2 = 0$$

with the fluid on the two sides of the pipe wall denoted by subscripts '1' and '2', respectively. Wall segments with air on only one side are modeled using Equation 2.

Results and Discussion

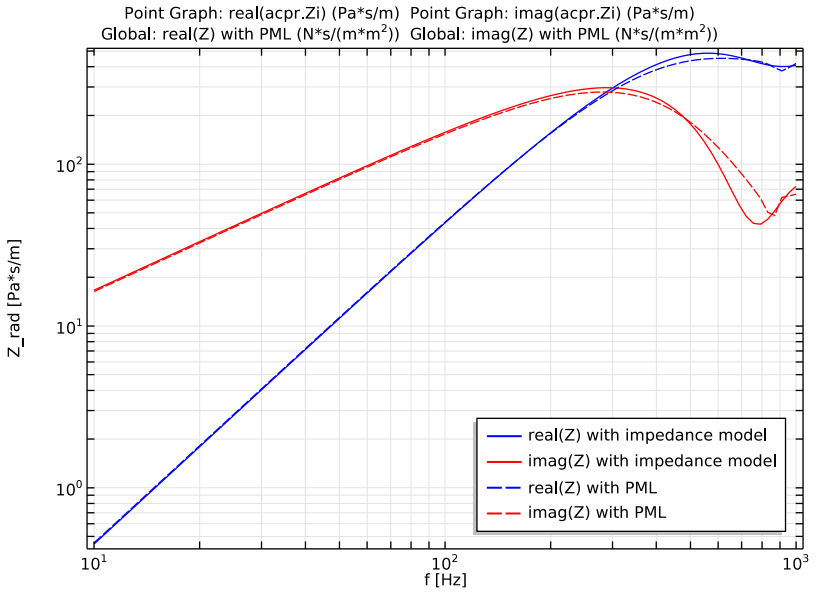


Figure 2: Impedance Z_i as a function of driving frequency f for the flanged pipe. For the PML model, the implied impedance is derived according to Equation 3.

FLANGED PIPE

We first consider the results from the flanged pipe. We are ultimately interested in comparing the pipe outlet pressure between the two modeling approaches (impedance or perfectly matched layer (PML)), but an important factor in this regard is to investigate the radiated losses at the pipe outlet. For the impedance-based model, these are exactly what the impedance Z_i expresses. To compare this to the model with PML, we compute an implied impedance on the pipe outlet by taking the ratio of the boundary-average pressure $\langle p \rangle$ to the boundary-average of the boundary-normal velocity $\langle v \rangle$. Thus, since the pipe outlet boundary is circular with radius a, the implied impedance for the PML model is

$$Z_i = \frac{\langle p \rangle}{\langle v \rangle} = \frac{\frac{1}{\pi a^2} 2\pi \int_0^a rp \ \mathrm{d}r}{\frac{1}{\pi a^2} 2\pi \int_0^a rv \ \mathrm{d}r} = \frac{\int_0^a rp \ \mathrm{d}r}{\int_0^a rv \ \mathrm{d}r}. \quad (3)$$

Figure 2 shows the result of this comparison. As expected, there is good comparison between the two modeling approaches until the cut-off frequency of f_c = 836.76 Hz. Above this, the non-plane waves supported by the geometry are observed to become increasingly influential as the impedance-based model (solid lines) begin to deviate.

Several impedance models have been derived as analytical approximations valid only for limited regimes of the relative wave number $k \cdot a$. To illustrate this, Figure 3 repeats the plot in Figure 2 but shown as a function of $k \cdot a$ rather than f.

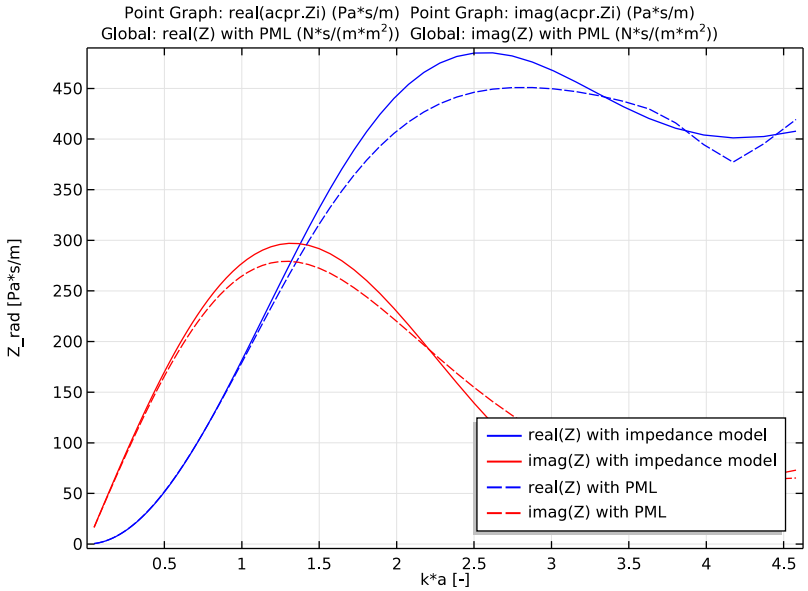


Figure 3: Impedance Z_i as a function of relative wave number $k \cdot a$ for the flanged pipe. For the PML model, the implied impedance is derived according to Equation 3.

Finally, we compare in Figure 4 the pressure at the centerline of the pipe outlet from the two models. We find good agreement across all frequencies even though the impedances exhibit some variations at higher frequencies (see Figure 2).

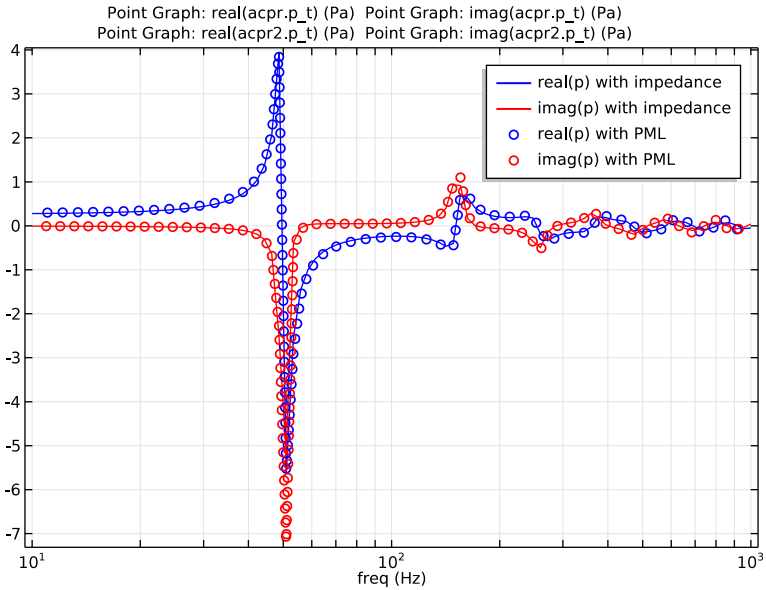


Figure 4: Pipe outlet pressure p for the flanged pipe for both impedance and PML models. In spite of noticeable variations for the impedances at higher frequencies (Figure 2), the predicted pressure from the two models is in general in good agreement across all frequencies.

UNFLANGED PIPE

We repeat the analysis for an unflanged pipe. In this case, the impedance model is an approximate analytical solution valid only for small values of $k \cdot a$. We present in Figure 5-Figure 7 the results of this analysis; these are analogous to the pipe results above for the flanged pipe and therefore warrant only short mention.

Figure 5 illustrates that the analytical and implied impedances are comparable across the entire frequency range. However, for $k \cdot a$ greater than about 2, the analytical impedance deviates quite significantly, see Figure 6. This is not quite as expected since the impedance model should apply for $k \cdot a < 3.83$, however, this could be due to the model itself being an approximation, not an exact analytical result even for low $k \cdot a$. Nonetheless, Figure 7 illustrates that the predicted centerline outlet pressure from either the impedance-based model or the PML model are in fair agreement across the entire range of probed frequencies.

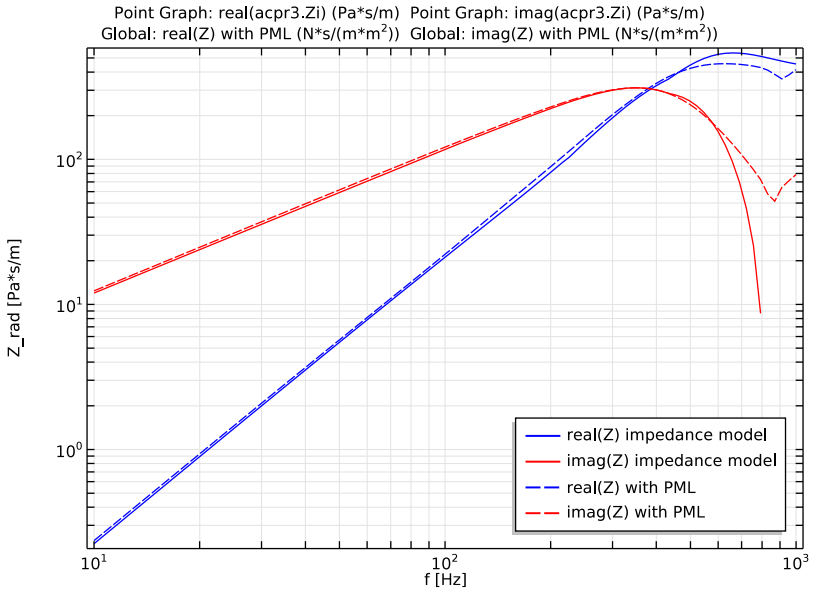


Figure 5: Impedance Z_i as a function of driving frequency f for the unflanged pipe. For the PML model, the implied impedance is derived according to Equation 3.

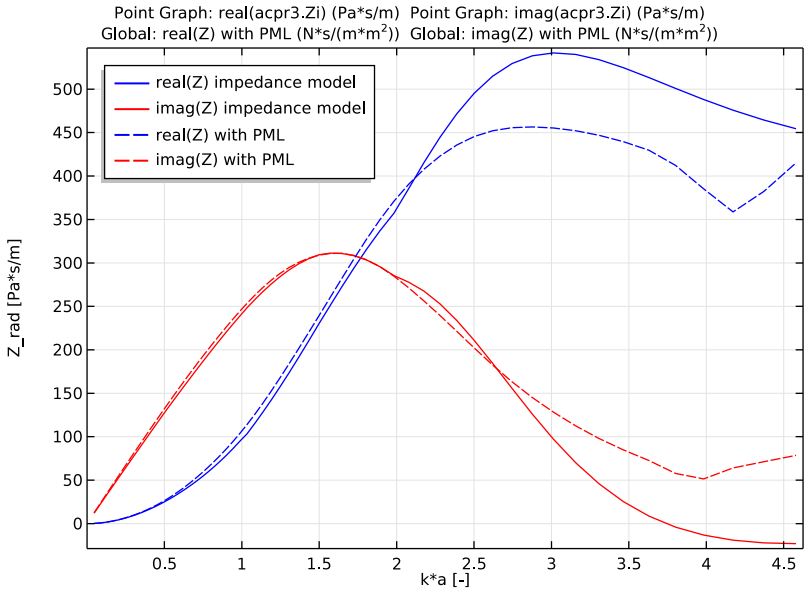


Figure 6: Impedance Z_i as a function of driving relative wave number $k \cdot a$ for the flanged pipe. For the PML model, the implied impedance is derived according to Equation 3.

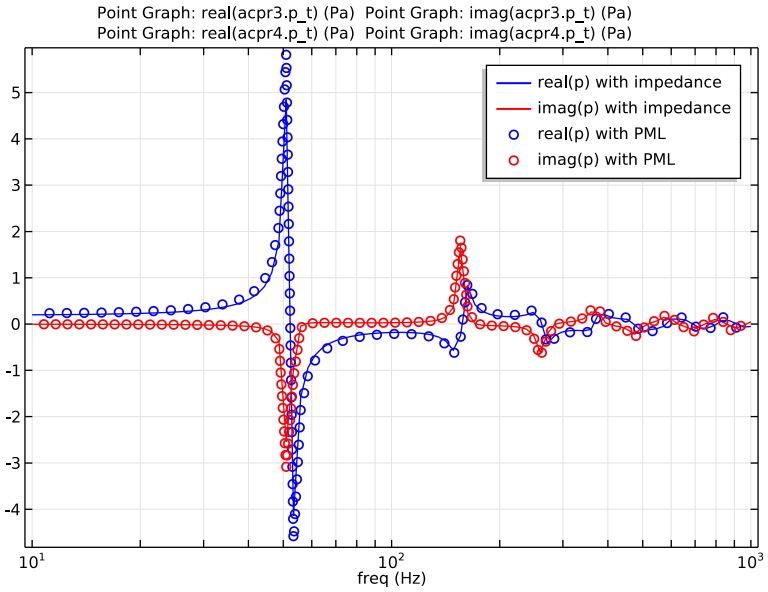


Figure 7: Pressure p at the centerline of the pipe outlet for the unflanged pipe.

Notes About the COMSOL Implementation

In anticipation of using a frequency sweep covering several decades, we define the frequency limits `fmin` and `fmax` as exponents without units and set the frequency range to $10^{\texttt{fmin}} - 10^{\texttt{fmax}}$.

Reference

1. D. T. Blackstock, *Fundamentals of physical acoustics*, John Wiley & Sons, 2000.

Application Library path: `Acoustics_Module/Verification_Examples/open_pipe`

Modeling Instructions

You start by modeling a flanged pipe. You will be comparing the two different approaches for solving the problem, so start by immediately adding the physics option twice.

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>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Add**.
5. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `open_pipe_parameters.txt`.

GEOMETRY 1

First you model the pipe.

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `a`.
4. In the **Height** text field, type `L`.
5. Locate the **Position** section. In the **z** text field, type `-L`.

Now model the surroundings. Only use a quarter circle to model the flange on the pipe.

Circle 1 (c1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Circle**.
2. In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `6*a`.

4 In the **Sector angle** text field, type `90`.

Add an external layer to the circle. You will later apply the Perfectly Matched Layer (PML) condition in this layer.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`2*a`

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

In pressure acoustics you only need to specify the density and speed of sound. When these parameters are known, this is most easily done by adding a blank material and providing the parameter values. Otherwise, add a built-in material. In this case you add the parameter values for air.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`rho_air`	kg/m³	Basic
Speed of sound	c	`c_air`	m/s	Basic

COMPONENT 1 (COMP1)

1 In the **Model Builder** window, click **Component 1 (comp1)**.

2 In the **Settings** window for **Component**, type `Component 1 - flanged pipe radiation` in the **Label** text field.

Set up the model of the flanged pipe using an impedance boundary condition.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 - flanged pipe radiation (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, type `Pressure Acoustics, Frequency Domain - impedance model` in the **Label** text field.

3 Select Domain 1 only.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN - IMPEDANCE MODEL (ACPR)

Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Acceleration**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Normal Acceleration**, locate the **Normal Acceleration** section.

4 In the a_n text field, type `a0`.

Impedance 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Impedance**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Impedance**, locate the **Impedance** section.

4 From the **Impedance model** list, choose **Waveguide end impedance**.

5 In the a text field, type `a`.

Now set up the model of the flanged pipe using a perfectly matched layer condition.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 2 (ACPR2)

1 In the **Model Builder** window, under **Component 1 - flanged pipe radiation (comp1)** click **Pressure Acoustics, Frequency Domain 2 (acpr2)**.

2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, type `Pressure Acoustics, Frequency Domain 2 - perfectly matched layers` in the **Label** text field.

Set up the perfectly matched layer. Also set up an integration operator on the pipe outlet, this will be used for post-processing.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 3 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

5 From the **Physics** list, choose **Pressure Acoustics, Frequency Domain 2 - perfectly matched layers (acpr2)**.

6 In the **PML scaling factor** text field, type `0.5`.

7 In the **PML scaling curvature parameter** text field, type `5`.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 2 - PERFECTLY MATCHED LAYERS (ACPR2)

Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Acceleration**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Normal Acceleration**, locate the **Normal Acceleration** section.

4 In the a_n text field, type `a0`.

Create the mesh.

MESH 1

1 In the **Model Builder** window, under **Component 1 - flanged pipe radiation (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

3 From the **Sequence type** list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 - flanged pipe radiation (comp1)> Mesh 1** 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 `hmax`.

5 In the **Minimum element size** text field, type `6.0E-4`.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 - flanged pipe radiation (comp1)> Mesh 1** click **Free Triangular 1**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1 and 2 only.

Size 1

1 Right-click **Component 1 - flanged pipe radiation (comp1)>Mesh 1>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 **Boundary**.

4 Select Boundaries 4 and 7 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 `hmax/3`.

Size 2

1 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 **Point**.

4 Select Point 6 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 `hmax/10`.

MESH 1

Free Triangular 1

In the **Model Builder** window, collapse the **Component 1 - flanged pipe radiation (comp1)> Mesh 1>Free Triangular 1** node.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundary 8 only.

4 In the **Settings** window for **Distribution**, locate the **Distribution** section.

5 In the **Number of elements** text field, type 14.

MESH 1

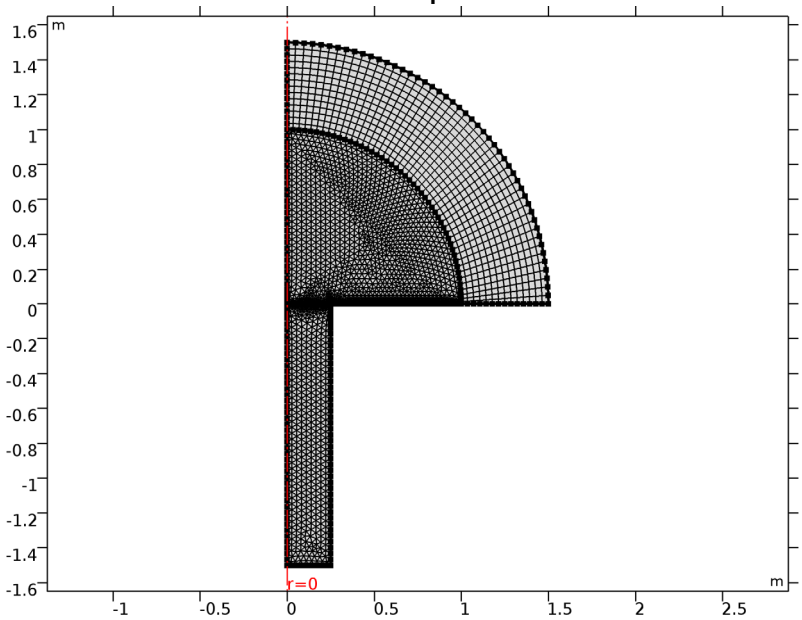
Mapped 1

1 In the **Model Builder** window, collapse the **Component 1 - flanged pipe radiation (comp1)> Mesh 1>Mapped 1** node.

2 In the **Model Builder** window, click **Mesh 1**.

3 In the **Settings** window for **Mesh**, click **Build All**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.



Mesh used to simulate the flanged pipe. The layer external to the quarter hemisphere is used for the perfectly meshed layer.

Next add two separate studies, one for each model (impedance-based model and model with perfectly matched layers).

ADD STUDY

1 On 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 **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

5 Click **Add Study** in the window toolbar.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

3 Click **Add Study** in the window toolbar.

4 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

Set up the study for the impedance-based model. This includes specifying solution frequencies and making sure that only the impedance-based model is solved. As default, the software will include both models, so you disable the model with perfectly matched layers.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study 1 - flanged pipe with impedance BC` in the **Label** text field.

STUDY 1 - FLANGED PIPE WITH IMPEDANCE BC

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - flanged pipe with impedance BC** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

5 In the **Start** text field, type `fmin`.

6 In the **Stop** text field, type `fmax`.

7 In the **Number of values** text field, type `100`.

8 From the **Function to apply to all values** list, choose **exp10(x)** – **Exponential function (base 10)**.

9 Click **Replace**.

Here you disable the model with perfectly matched layers.

10 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

11 In the table, clear the **Solve for** check box for **Pressure Acoustics, Frequency Domain 2 - perfectly matched layers (acpr2)**.

Set up the study for the model with perfectly matched layers similar to the other study above.

STUDY 2

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - flanged pipe with PML` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 2 - FLANGED PIPE WITH PML

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2 - flanged pipe with PML** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

5 In the **Start** text field, type `fmin`.

6 In the **Stop** text field, type `fmax`.

7 In the **Number of values** text field, type `100`.

8 From the **Function to apply to all values** list, choose **exp10(x)** – **Exponential function (base 10)**.

9 Click **Replace**.

Disable the impedance-based model.

10 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

11 In the table, clear the **Solve for** check box for **Pressure Acoustics, Frequency Domain - impedance model (acpr)**.

STUDY 1 - FLANGED PIPE WITH IMPEDANCE BC

On the **Home** toolbar, click **Compute**.

STUDY 2 - FLANGED PIPE WITH PML

Click **Compute**.

Compare the pipe outlet impedance between the impedance-based model and the model with perfectly matched layers.

RESULTS

Create the plot in Figure 2.

1D Plot Group 5

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Flanged pipe: Z_rad vs f` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **None**.
4. Locate the **Plot Settings** section. Select the **x-axis label** check box.
5. In the associated text field, type `f [Hz]`.
6. Select the **y-axis label** check box.
7. In the associated text field, type `Z_rad [Pa*s/m]`.

Point Graph 1

1. Right-click **Flanged pipe: Z_rad vs f** and choose **Point Graph**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. From the **Data set** list, choose **Study 1 - flanged pipe with impedance BC/Solution 1 (sol1)**.
4. Select Point 2 only.
5. Locate the **y-Axis Data** section. In the **Expression** text field, type `real(acpr.Zi)`.
6. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
7. Click to expand the **Legends** section. Select the **Show legends** check box.
8. From the **Legends** list, choose **Manual**.
9. In the table, enter the following settings:

Legends
`real(Z) with impedance model`

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Flanged pipe: Z_rad vs f** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1 - flanged pipe with impedance BC/Solution 1 (sol1)**.

4 Select Point 2 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `imag(acpr.Zi)`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`imag(Z) with impedance model`

10 On the **Flanged pipe: Z_rad vs f** toolbar, click **Plot**.

Global 1

1 Right-click **Flanged pipe: Z_rad vs f** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - flanged pipe with PML/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`intop1(p2)`	`N`	

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Blue**.

7 Click to expand the **Legends** section. On the **Flanged pipe: Z_rad vs f** toolbar, click **Plot**.

Global 2

1 Right-click **Flanged pipe: Z_rad vs f** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - flanged pipe with PML/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`imag(intop1(p2)/ intop1(acpr2.vz))`	`N*s/(m*m^2)`	`imag(Z) with PML`

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Red**.

7 On the **Flanged pipe: Z_rad vs f** toolbar, click **Plot**.

8 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

9 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

Flanged pipe: Z_rad vs f

1 In the **Model Builder** window, under **Results** click **Flanged pipe: Z_rad vs f**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

3 From the **Position** list, choose **Lower right**.

Global 1

1 In the **Model Builder** window, under **Results>Flanged pipe: Z_rad vs f** click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`intop1(p2)/intop1(acpr2.vz)`	`N*s/(m*m^2)`	`real(Z) with PML`

4 On the **Flanged pipe: Z_rad vs f** toolbar, click **Plot**.

You can create an alternative version of the plot by duplicating the plot group and changing the x-axis from `f` to k*a in all plots. See result in Figure 3.

COMPONENT 1 - FLANGED PIPE RADIATION (COMP1)

In the **Model Builder** window, collapse the **Component 1 - flanged pipe radiation (comp1)** node.

Repeat the analysis for an unflanged pipe. Start by adding a new component.

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>2D Axisymmetric**.

COMPONENT 2 (COMP2)

1 In the **Model Builder** window, click **Component 2 (comp2)**.

2 In the **Settings** window for **Component**, type `Component 2 - unflanged pipe radiation` in the **Label** text field.

Create an unflanged pipe geometry.

GEOMETRY 2

First create the pipe.

1 In the **Model Builder** window, under **Component 2 - unflanged pipe radiation (comp2)** click **Geometry 2**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `a`.

4 In the **Height** text field, type `L`.

5 Locate the **Position** section. In the **z** text field, type `-L`.

Then create the surroundings.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

Add an external layer which you will use when applying the perfectly matched layer condition.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `6*a`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`2*a`

7 Click **Build All Objects**.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Model Builder** window, right-click **Geometry 2** and choose **Delete Entities**.

Delete overlaps between the pipe and the surroundings.

Delete Entities 1 (del1)

1 On the object **uni1**, select Boundaries 11 and 12 only.

2 On the **Geometry** toolbar, click **Build All**.

3 Click the **Zoom Extents** button on the **Graphics** toolbar.

Now add physics and boundary conditions.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 3 (ACPR3)

1 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, type `Pressure Acoustics, Frequency Domain 3 - impedance BC` in the **Label** text field.

2 Locate the **Domain Selection** section. From the **Selection** list, choose **Manual**.

3 Select Domain 1 only.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 3 - IMPEDANCE BC (ACPR3)

Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Acceleration**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Normal Acceleration**, locate the **Normal Acceleration** section.

4 In the a_n text field, type `a0`.

Impedance 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Impedance**.

Remember to add the correct boundary impedance model.

2 Select Boundary 5 only.

3 In the **Settings** window for **Impedance**, locate the **Impedance** section.

4 From the **Impedance model** list, choose **Waveguide end impedance**.

5 From the list, choose **Unflanged pipe, circular**.

6 In the a text field, type `a`.

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 2 - unflanged pipe radiation (comp2)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`rho_air`	kg/m³	Basic
Speed of sound	c	`c_air`	m/s	Basic

ADD PHYSICS

1 On the **Physics** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

4 Click **Add to Component** in the window toolbar.

5 On the **Physics** toolbar, click **Add Physics** to close the **Add Physics** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 4 (ACPR4)

In the **Settings** window for **Pressure Acoustics, Frequency Domain**, type `Pressure Acoustics, Frequency Domain 4 - perfectly matched layers` in the **Label** text field.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN 4 - PERFECTLY MATCHED LAYERS (ACPR4)

Normal Acceleration 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Acceleration**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Normal Acceleration**, locate the **Normal Acceleration** section.

4 In the a_n text field, type `a0`.

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 Select Boundaries 8 and 9 only.

Now define the perfectly matched layer condition and an integration operator acting on the pipe outlet.

DEFINITIONS

Perfectly Matched Layer 2 (pml2)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 3 and 4 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Physics** list, choose **Pressure Acoustics, Frequency Domain 4 - perfectly matched layers (acpr4)**.

5 In the **PML scaling factor** text field, type `0.5`.

6 In the **PML scaling curvature parameter** text field, type `5`.

7 From the **Coordinate stretching type** list, choose **Rational**.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 5 only.

Create the mesh.

MESH 2

Edge 1

1 In the **Model Builder** window, under **Component 2 - unflanged pipe radiation (comp2)** right-click **Mesh 2** and choose **More Operations>Edge**.

2 Select Boundaries 6, 8, and 10 only.

3 In the **Settings** window for **Edge**, click to expand the **Control entities** section.

Size

1 In the **Model Builder** window, under **Component 2 - unflanged pipe radiation (comp2)> Mesh 2** 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 `hmax`.

5 In the **Minimum element size** text field, type `6.0E-4`.

Size 1

1 In the **Model Builder** window, right-click **Mesh 2** and choose **Free Triangular**.

2 Right-click **Free Triangular 1** and choose **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 5 and 8 only.

6 Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `hmax/5`.

Size 2

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Point**.

5 Select Point 9 only.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `hmax/15`.

Free Triangular 1

1 In the **Model Builder** window, under **Component 2 - unflanged pipe radiation (comp2)> Mesh 2** click **Free Triangular 1**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1 and 2 only.

5 In the **Model Builder** window, collapse the **Free Triangular 1** node.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 2** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

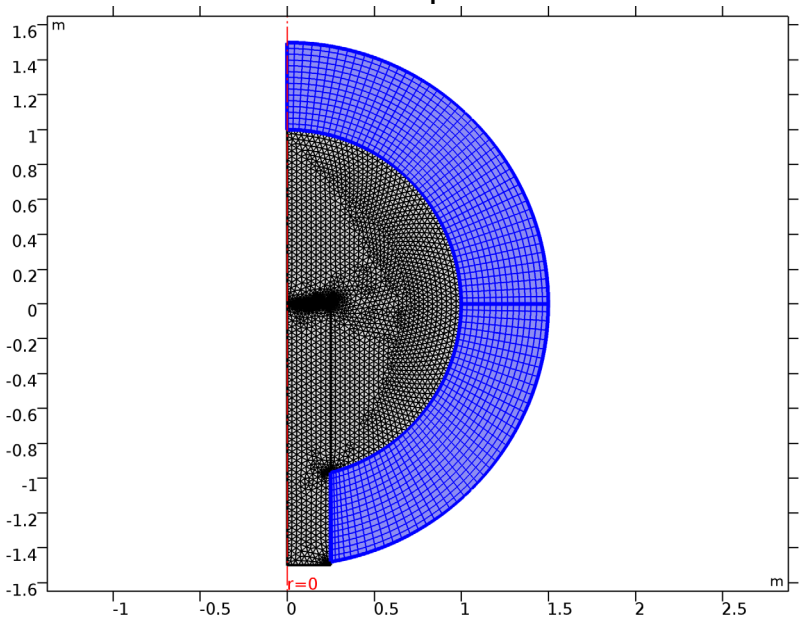
3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 3 and 4 only.

5 Click to expand the **Control entities** section. Click to expand the **Advanced settings** section. Locate the **Advanced Settings** section. Select the **Adjust evenly distributed edge mesh** check box.

6 Click **Build All**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.



Mesh used to simulate the unflanged pipe. The highlighted regions are used to apply the perfectly matched layer condition.

Next, add two studies: one for the impedance-based model and another for the model with perfectly matched layers. For each study, keep only one physics model (impedance-based or with perfectly matched layers) and immediately disable all other physics models. This is similar to what you did for the flanged pipe model in **Component 1**.

ADD STUDY

1 On 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 **Preset Studies**.

4 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

5 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Pressure Acoustics, Frequency Domain - impedance model (acpr)	
Pressure Acoustics, Frequency Domain 2 - perfectly matched layers (acpr2)	
Pressure Acoustics, Frequency Domain 3 - impedance BC (acpr3)	√
Pressure Acoustics, Frequency Domain 4 - perfectly matched layers (acpr4)	

6 Click **Add Study** in the window toolbar.

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

3 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Pressure Acoustics, Frequency Domain - impedance model (acpr)	
Pressure Acoustics, Frequency Domain 2 - perfectly matched layers (acpr2)	
Pressure Acoustics, Frequency Domain 3 - impedance BC (acpr3)	
Pressure Acoustics, Frequency Domain 4 - perfectly matched layers (acpr4)	√

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

Step 1: Frequency Domain

Set up the studies.

STUDY 3

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

4 In the **Label** text field, type `Study 3 - unflanged pipe with impedance BC`.

STUDY 3 - UNFLANGED PIPE WITH IMPEDANCE BC

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3 - unflanged pipe with impedance BC** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

5 In the **Start** text field, type `fmin`.

6 In the **Stop** text field, type `fmax`.

7 In the **Number of values** text field, type `100`.

8 From the **Function to apply to all values** list, choose **exp10(x)** – **Exponential function (base 10)**.

9 Click **Replace**.

10 On the **Home** toolbar, click **Compute**.

STUDY 4

1 In the **Model Builder** window, click **Study 4**.

2 In the **Settings** window for **Study**, type `Study 4 - unflanged pipe with PML` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 4 - UNFLANGED PIPE WITH PML

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 4 - unflanged pipe with PML** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

5 In the **Start** text field, type `fmin`.

6 In the **Stop** text field, type `fmax`.

7 In the **Number of values** text field, type `100`.

8 From the **Function to apply to all values** list, choose **exp10(x)** – **Exponential function (base 10)**.

9 Click **Replace**.

10 On the **Home** toolbar, click **Compute**.

RESULTS

Now compare the pipe outlet impedance from the two models in a plot. The result is shown in Figure 5.

1D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Unflanged pipe: Z_rad vs f` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `f [Hz]`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Z_rad [Pa*s/m]`.

8 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Point Graph 1

1 Right-click **Unflanged pipe: Z_rad vs f** and choose **Point Graph**.

2 Select Point 3 only.

3 In the **Settings** window for **Point Graph**, locate the **Data** section.

4 From the **Data set** list, choose **Study 3 - unflanged pipe with impedance BC/ Solution 3 (4) (sol3)**.

5 Select Point 3 only.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(acpr3.Zi)`.

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

8 Locate the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
real(Z) impedance model

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Unflanged pipe: Z_rad vs f** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3 - unflanged pipe with impedance BC/ Solution 3 (4) (sol3)**.

4 Select Point 3 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `imag(acpr3.Zi)`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
imag(Z) impedance model

10 On the **Unflanged pipe: Z_rad vs f** toolbar, click **Plot**.

Global 1

1 Right-click **Unflanged pipe: Z_rad vs f** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 4 - unflanged pipe with PML/Solution 4 (6) (sol4)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
intop2(p4)/intop2(acpr4.vz)	N*s/(m*m^2)	real(Z) with PML

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Blue**.

7 On the **Unflanged pipe: Z_rad vs f** toolbar, click **Plot**.

Global 2

1 Right-click **Unflanged pipe: Z_rad vs f** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 4 - unflanged pipe with PML/Solution 4 (6) (sol4)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`imag(intop2(p4)/ intop2(acpr4.vz))`	`N*s/(m*m^2)`	`imag(Z) with PML`

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Red**.

7 On the **Unflanged pipe: Z_rad vs f** toolbar, click **Plot**.

8 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

9 Click the **y-Axis Log Scale** button on the **Graphics** toolbar.

You can duplicate the plot group and change the x-axis from `f` to `k*a` to replot the result as shown in Figure 6.

Compare the centerline outlet pressure using both the impedance-based model and the model with perfectly matched layers. Do this for both the flanged and unflanged pipes. See the results in Figure 4 and Figure 7.

1D Plot Group 9

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Flanged pipe: centerline outlet pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

Point Graph 1

1 Right-click **Flanged pipe: centerline outlet pressure** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1 - flanged pipe with impedance BC/Solution 1 (sol1)**.

4 Select Point 2 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(acpr.p_t)`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`real(p) with impedance`

10 On the **Flanged pipe: centerline outlet pressure** toolbar, click **Plot**.

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Flanged pipe: centerline outlet pressure** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1 - flanged pipe with impedance BC/Solution 1 (sol1)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `imag(acpr.p_t)`.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

6 Locate the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`imag(p) with impedance`

9 On the **Flanged pipe: centerline outlet pressure** toolbar, click **Plot**.

10 Locate the **Selection** section. Select the **Active** toggle button.

11 Select Point 2 only.

12 On the **Flanged pipe: centerline outlet pressure** toolbar, click **Plot**.

Point Graph 3

1 Right-click **Flanged pipe: centerline outlet pressure** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - flanged pipe with PML/Solution 2 (sol2)**.

4 Select Point 2 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(acpr2.p_t)`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

7 Find the **Line style** subsection. From the **Line** list, choose **None**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

9 In the **Number** text field, type 100.

10 Locate the **Legends** section. Select the **Show legends** check box.

11 From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
real(p) with PML

13 On the **Flanged pipe: centerline outlet pressure** toolbar, click **Plot**.

Point Graph 4

1 Right-click **Flanged pipe: centerline outlet pressure** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - flanged pipe with PML/Solution 2 (sol2)**.

4 Select Point 2 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `imag(acpr2.p_t)`.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 From the **Color** list, choose **Red**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

9 In the **Number** text field, type 100.

10 Locate the **Legends** section. Select the **Show legends** check box.

11 From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
imag(p) with PML

13 On the **Flanged pipe: centerline outlet pressure** toolbar, click **Plot**.

14 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Now the unflanged pipe.

1D Plot Group 10

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Unflanged pipe: centerline outlet pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

Point Graph 1

1 Right-click **Unflanged pipe: centerline outlet pressure** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3 - unflanged pipe with impedance BC/ Solution 3 (4) (sol3)**.

4 Select Point 3 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(acpr3.p_t)`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`real(p) with impedance`

10 On the **Unflanged pipe: centerline outlet pressure** toolbar, click **Plot**.

Point Graph 2

1 In the **Model Builder** window, under **Results** right-click **Unflanged pipe: centerline outlet pressure** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3 - unflanged pipe with impedance BC/ Solution 3 (4) (sol3)**.

4 Select Point 3 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `imag(acpr3.p_t)`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`imag(p) with impedance`

10 On the **Unflanged pipe: centerline outlet pressure** toolbar, click **Plot**.

Point Graph 3

1 Right-click **Unflanged pipe: centerline outlet pressure** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 4 - unflanged pipe with PML/Solution 4 (6) (sol4)**.

4 Select Point 3 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `real(acpr4.p_t)`.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 From the **Color** list, choose **Blue**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

9 Locate the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
`real(p) with PML`

12 On the **Unflanged pipe: centerline outlet pressure** toolbar, click **Plot**.

13 Locate the **Coloring and Style** section. In the **Number** text field, type `100`.

14 On the **Unflanged pipe: centerline outlet pressure** toolbar, click **Plot**.

Point Graph 4

1 Right-click **Unflanged pipe: centerline outlet pressure** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 4 - unflanged pipe with PML/Solution 4 (6) (sol4)**.

4 Select Point 3 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `imag(acpr4.p_t)`.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 From the **Color** list, choose **Red**.

8 Locate the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
imag(p) with PML

11 On the **Unflanged pipe: centerline outlet pressure** toolbar, click **Plot**.

12 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

13 In the **Number** text field, type 100.

14 On the **Unflanged pipe: centerline outlet pressure** toolbar, click **Plot**.

15 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Disable the unflanged interfaces in the first two studies, so that you can run them later.

STUDY 1 - FLANGED PIPE WITH IMPEDANCE BC

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1 - flanged pipe with impedance BC** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for
Pressure Acoustics, Frequency Domain - impedance model (acpr)	√
Pressure Acoustics, Frequency Domain 2 - perfectly matched layers (acpr2)	
Pressure Acoustics, Frequency Domain 3 - impedance BC (acpr3)	
Pressure Acoustics, Frequency Domain 4 - perfectly matched layers (acpr4)	

STUDY 2 - FLANGED PIPE WITH PML

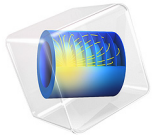
1 In the **Model Builder** window, click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for
Pressure Acoustics, Frequency Domain - impedance model (acpr)	
Pressure Acoustics, Frequency Domain 2 - perfectly matched layers (acpr2)	√

Physics interface	Solve for
Pressure Acoustics, Frequency Domain 3 - impedance BC (acpr3)	
Pressure Acoustics, Frequency Domain 4 - perfectly matched layers (acpr4)	



Organ Pipe Design

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

In this model an organ flue pipe is designed and analyzed using a pipe acoustics model. Because it is a 1D model, it is fast to solve but still retains most of the relevant physical parameters when designing an organ pipe. The model includes the elastic properties of the organ pipe walls, the end impedance properties at the open pipe end, and the possibility to add a small background airflow inside the pipe. In this model the pipe is driven at 440 Hz which is the A4 note (or a').

A sketch of an organ pipe is shown in Figure 1. An airflow is pushed in at the bottom of the organ pipe and out via the mouth. At the mouth, an air jet strikes the sharp upper lip and this sets the air into vibration. The vibrations resonate with the organ pipe body to create the note of the pipe. In an open pipe, like the one sketched here, the fundamental tone corresponds to a half wave resonance in the pipe. The harmonics are then multiples of this frequency.

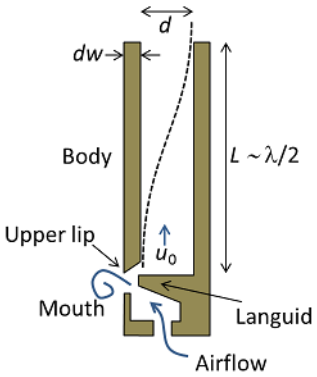


Figure 1: Sketch of an organ pipe including the mouth and the pipe body.

Note: This application requires the Acoustics Module and the Pipe Flow Module.

The timbre of the organ pipe depends on the combination of the fundamental tone and all the harmonics. This depends on the shape of the pipe (the length and diameter) as well as on the elastic properties of the pipe walls and their thickness. Moreover, a small residual airflow in the pipe u_0, may alter the natural frequencies slightly (this effect is not modeled here). Changing any of these parameters influences not only the natural frequencies of the organ pipe, but also the damping and Q value of the corresponding frequency response resonance peaks. This in turn yields a different timbre.

Model Definition

The organ pipe geometry is defined in terms of its length L, inner pipe diameter d, wall thickness dw (see Figure 1), and cross section shape (here circular). Only the length is used when drawing the pipe geometry as a straight line segment. The inner radius, wall thickness, and pipe shape are parameters entering the governing equations. The elastic properties of the pipe wall are Young's modulus E_{w} and Poisson's ratio ν_{w}. The model parameters are given in the table below.

TABLE 0-1: MODEL PARAMETERS

NAME	EXPRESSION	DESCRIPTION
f	440 Hz	Frequency of an A4 note
L_{guess}	$c_0/(2\,f)$	Half wavelength at f
L	0.3805 m	Pipe length giving a resonance at 440 Hz
d	3 cm	Inner pipe diameter
dw	2 mm	Wall thickness
E_{w}	10^9 Pa	Young's modulus of the wall material
ν_{w}	0.4	Poisson's ratio of the wall material
c_0	343 m/s	Speed of sound
ka	0.12	Wavenumber times pipe radius
h_{min}	c_0 / 3000 Hz / 20	Mesh size at 3000 Hz.

The open end of the pipe is modeled by adding an end impedance property. This is an engineering relation for the case of a pipe of circular cross section ending sitting in free space (an unflanged pipe).

Results and Discussion

The frequency response of the pipe is obtained by plotting the sound pressure level L_{p} at the open pipe end,

$$L_{\mathrm{p}} = 10\log\left[\left(\frac{p_{\mathrm{rms}}}{p_{\mathrm{ref}}}\right)^2\right] \qquad p_{\mathrm{rms}}^2 = \frac{1}{2}pp^*$$

where p_{ref} is the reference pressure for air 20 µPa and * is the complex conjugate.

The frequency response around the first resonance frequency if plotted in Figure 2 for several values of the pipe diameter. Changing the pipe diameter clearly shifts the resonance

frequency but also changes the damping and Q value, that is, the width of the peak. Hence this is an important factor when designing organ pipes.

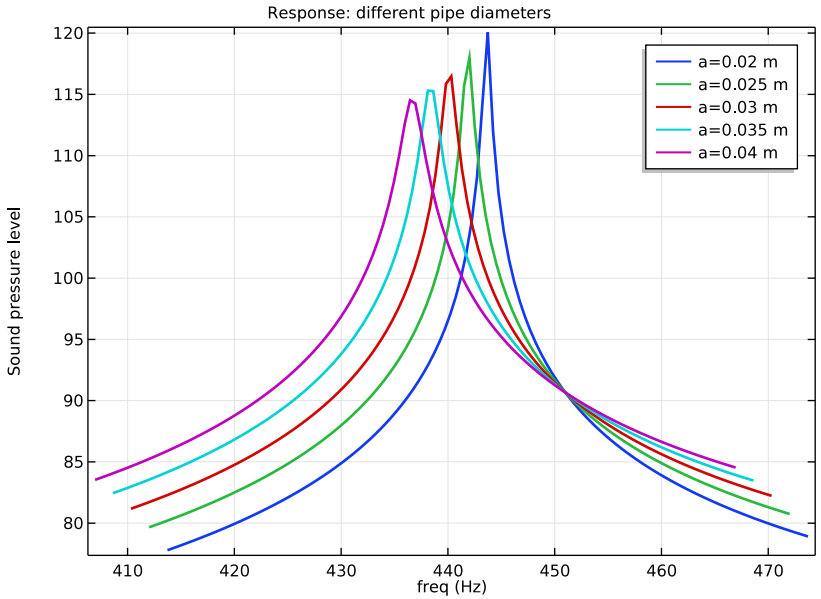


Figure 2: Resonance peak of the fundamental frequency at 440 Hz for different inner pipe diameters.

The response for different values of the pipe wall width is plotted in Figure 3. Here it is also seen that changing the pipe wall width (in general any of the pipe wall properties) changes the resonance slightly. This is because the elastic properties of the pipe wall has influence on the effective compressibility of the system in a given cross section. This in turn changes the effective speed of sound in the pipe and thus the resonance. The value of Young's modulus in this example model was chosen to demonstrate this effect. A typical circular organ pipe is however made of a tin-lead alloy, which has a much higher value of Young's modulus.

In the final plot in Figure 4 the parameter values giving a fundamental resonance at 440 Hz are selected (see the parameters list) and the response is plotted for frequencies from 100 Hz to 3000 Hz. The plot shows the fundamental resonance at 440 Hz as well as the following five resonance frequencies of the organ pipe. The shape of this curve is related to the timbre of the pipe.

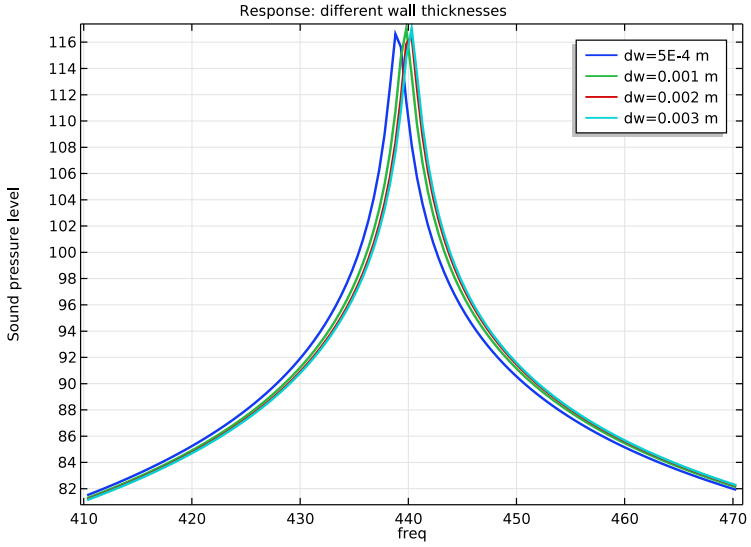


Figure 3: Resonance peak of the fundamental frequency at 440 Hz for different pipe wall thickness.

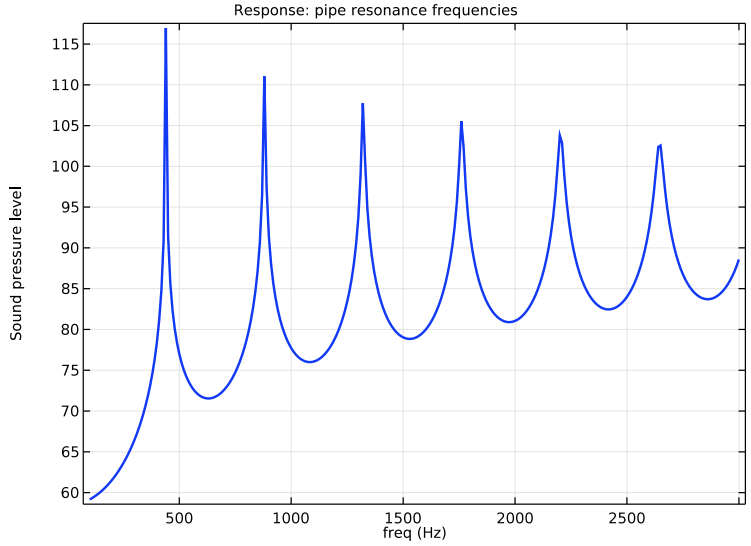


Figure 4: Resonance peaks of the six first natural frequencies of the pipe from 100 to 3000 Hz.

Note: This application requires the Acoustics Module and the Pipe Flow Module.

Application Library path: `Acoustics_Module/Tutorials/organ_pipe_design`

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 **Acoustics>Acoustic-Structure Interaction>Pipe Acoustics, Frequency Domain (pafd)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `organ_pipe_design_parameters.txt`.

GEOMETRY 1

Polygon 1 (pol1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the **x** text field, type `0 0`.

4 In the **y** text field, type `0 0`.

5 In the **z** text field, type `0 L`.

6 Click **Build All Objects**.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PIPE ACOUSTICS, FREQUENCY DOMAIN (PAFD)

Set the cross section shape of the organ pipe and the elastic properties of the pipe walls.

Fluid Properties 1

1 In the **Settings** window for **Fluid Properties**, locate the **Model Input** section.

2 In the T_0 text field, type `T0`.

3 In the p_0 text field, type `p0`.

Pipe Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pipe Acoustics, Frequency Domain (pafd)** click **Pipe Properties 1**.

2 In the **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.

3 From the list, choose **Circular**.

4 In the d_i text field, type `d`.

5 Locate the **Pipe Model** section. From the **Pipe model** list, choose **Anchored at one end**.

6 From the E list, choose **User defined**.

7 In the associated text field, type `Ew`.

8 From the ν list, choose **User defined**.

9 In the associated text field, type `nuw`.

10 From the Δw list, choose **User defined**.

11 In the associated text field, type `dw`.

End Impedance 1

At the open end, the organ pipe is sitting in free air. Use the **Unflanged pipe, circular** end impedance to get the correct acoustic behavior here. Note that a low ka limit version also exists but this one is only valid for $k \cdot a << 1$. In this model $k \cdot a \geq 0.12$ as seen in the parameters list.

1. On the **Physics** toolbar, click **Points** and choose **End Impedance**.
2. Select Point 2 only.
3. In the **Settings** window for **End Impedance**, locate the **End Impedance** section.
4. From the **Impedance model** list, choose **Unflanged pipe, circular**.

Pressure 1

1. On the **Physics** toolbar, click **Points** and choose **Pressure**.
2. Select Point 1 only.
3. In the **Settings** window for **Pressure**, locate the **Pressure** section.
4. In the p_{in} text field, type `1`.

MESH 1

Edge 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.
2. Click in the **Graphics** window and then press Ctrl+A to select all edges.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `h_min`.
5. In the **Minimum element size** text field, type `h_min/2`.
6. Click **Build All**.

STUDY 1

Step 1: Frequency Domain

1. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(f_est-30,0.5,f_est+30)`.

Here the variable `f_est` corresponds to the estimated resonance frequency of the pipe taking the end correction into account. For narrow pipes the calculated resonance `f_est` is close to the ideal A4 (or a') note of f = 440 Hz.

Parametric Sweep

The sweep over the pipe diameter is done using a Parametric Sweep.

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
d (Inner pipe diameter)	`2[cm] 2.5[cm] 3[cm] 3.5[cm] 4[cm]`	

5 On the **Study** toolbar, click **Compute**.

ADD STUDY

1 On the **Study** 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Study** 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 In the **Frequencies** text field, type `range(f_est-30,0.5,f_est+30)`.

3 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dw (Pipe wall thickness)	`0.5[mm] 1[mm] 2[mm] 3[mm]`	

6 In the **Model Builder** window, click **Study 2**.

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

9 On the **Study** toolbar, click **Compute**.

ADD STUDY

1 On the **Study** 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(100,10,3000)`.

3 In the **Model Builder** window, click **Study 3**.

4 In the **Settings** window for **Study**, locate the **Study Settings** section.

5 Clear the **Generate default plots** check box.

6 On the **Study** toolbar, click **Compute**.

RESULTS

Pressure (pafd)

The first two figures show the pressure distribution and the velocity field in the pipe section as line plots in 3D. You can select different parameter values and frequencies and evaluate the plots to study the pressure distribution in the pipe.

Next, create three plots that show the frequency response of the organ pipe. This is here the sound pressure level evaluated at the open end of the of the pipe for three different

cases. The first shows the fundamental resonance and how it depends on the inner tube diameter. The next shows the dependency on the tube wall thickness. The final plot shows the solution for a larger frequency range including the six first resonance frequencies.

1D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

Point Graph 1

1 Right-click **1D Plot Group 3** and choose **Point Graph**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Pipe Acoustics, Frequency Domain>Intensity and sound pressure level>pafd.Lp - Sound pressure level**.

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type 2.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
a=0.02 m
a=0.025 m
a=0.03 m
a=0.035 m
a=0.04 m

1D Plot Group 3

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, type `Response: different pipe diameters` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Response: different pipe diameters`.

5 Locate the **Plot Settings** section. Select the **y-axis label** check box.

6 In the associated text field, type `Sound pressure level`.

7 On the **Response: different pipe diameters** toolbar, click **Plot**.

The figure should look like the one in Figure 2

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 8 (sol8)**.

Point Graph 1

1 Right-click **1D Plot Group 4** and choose **Point Graph**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `pafd.Lp`.

5 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **freq**.

6 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type 2.

7 Click to expand the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
dw=5E-4 m
dw=0.001 m
dw=0.002 m
dw=0.003 m

1D Plot Group 4

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Response: different wall thicknesses` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Response: different wall thicknesses`.

5 Locate the **Plot Settings** section. Select the **y-axis label** check box.

6 In the associated text field, type `Sound pressure level`.

7 On the **Response: different wall thicknesses** toolbar, click **Plot**.

The figure should look like the one in Figure 3

1D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3/Solution 9 (sol9)**.

Point Graph 1

1 Right-click **1D Plot Group 5** and choose **Point Graph**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `pafd.Lp`.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

1D Plot Group 5

1 In the **Model Builder** window, under **Results** click **1D Plot Group 5**.

2 In the **Settings** window for **1D Plot Group**, type `Response: pipe resonance frequencies` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

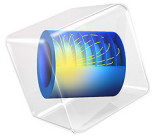
4 In the **Title** text area, type `Response: pipe resonance frequencies`.

5 Locate the **Plot Settings** section. Select the **y-axis label** check box.

6 In the associated text field, type `Sound pressure level`.

7 On the **Response: pipe resonance frequencies** toolbar, click **Plot**.

The figure should look like the one in Figure 4.



Muffler with Perforates

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The original version of this model was developed by Dr. Sabry Allam and Prof. Mats Åbom at the Marcus Wallenberg Laboratory for Sound and Vibration Research, Royal Institute of Technology, Stockholm, Sweden. Dr. Allam and Prof. Åbom also provided the experimental data used in the model.

There are two basic types of mufflers:

- *Reflective* (or *reactive*) *mufflers*—those that reflect acoustic waves by abrupt area expansions or changes of impedance.
- *Dissipative mufflers*—mufflers based on dissipation of acoustic energy into heat through viscous and thermal losses in fibrous materials or flow-related (resistive) losses in perforated pipes.

Reflective mufflers are best suited for the low frequency range where only plane waves can propagate in the system, while dissipative mufflers with fibers are efficient in the mid-to-high frequency range. Dissipative mufflers based on flow losses, on the other hand, work also at low frequencies. A typical automotive exhaust system is a hybrid construction consisting of a combination of reflective and dissipative muffler elements. The reflective parts are normally tuned to remove dominating low-frequency engine harmonics while the dissipative parts are designed to take care of higher-frequency noise.

In the industry, exhaust systems are typically analyzed with nonlinear 1D gas-dynamics codes. Such codes, however, do not capture 3D acoustic effects such as higher-order duct modes, and the modeling of fibrous materials is not satisfactory. In practice, there is therefore a need to use linear acoustic models of exhaust and intake systems to enable detailed modeling and optimization of the acoustic response, at the cost of neglecting nonlinear effects.

Model Definition

The muffler you analyze here is an example of a complex hybrid muffler in which the dissipative element is created completely by flow through perforated pipes and plates. When designing a model for a muffler without fibrous materials you need to consider the following aspects:

- *Geometry*—The design for this model is based on a modular muffler developed for research purposes. It closely resembles commercially available automotive mufflers, and was used as a test case for muffler modeling in an EC-project (ARTEMIS).

- *Mean flow distribution*—The Mach number in an exhaust system is normally less than 0.3. This means that in mufflers with flow expansions, the average Mach number is quite small (less than 0.1). For such cases you can neglect the convective flow effects, and the only important effect of the mean flow is its influence on the impedance of perforated pipes/plates. This model treats the case where there is no mean flow in the muffler.
- *Temperature distribution*—In a running engine, the air temperature inside the muffler is typically in the range 300–400 °C. There is also a temperature gradient through the muffler. However, the acoustic effect of this gradient is small and the average temperature is normally used to calculate the speed of sound. In this case, the experiments were performed at room temperature (20 °C). The model therefore assumes the temperature in the muffler to be constant and uses the default values for air density and speed of sound at 1 atm and 20 °C.

A schematic cross section of the muffler geometry is shown in Figure 1.

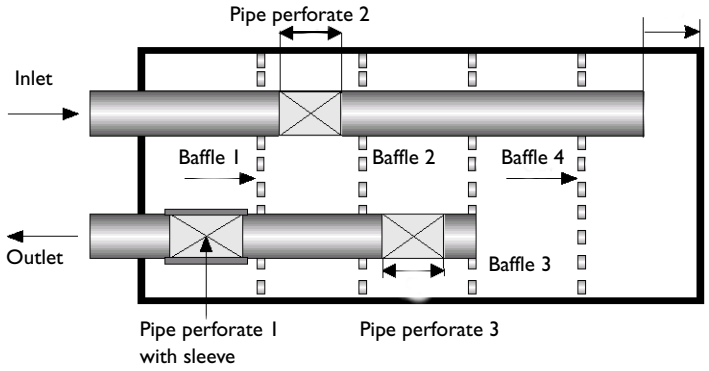


Figure 1: Muffler geometry cross section.

The detailed design and dimensions of the outlet pipe and the four baffles (as seen from the right in Figure 1) are given in Figure 2 through Figure 5.

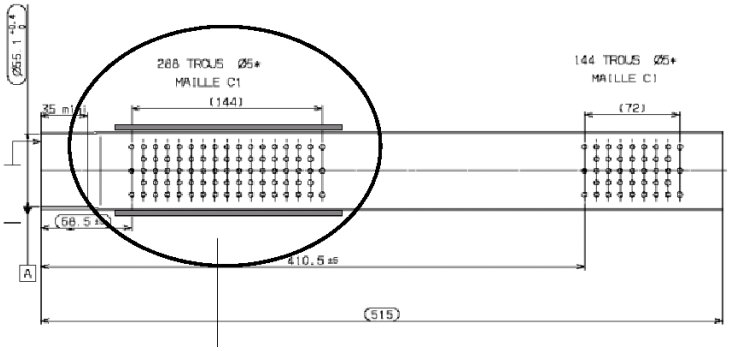


Figure 2: Outlet pipe. A stainless steel sleeve is located above the left perforated section with 288 holes. The other two pipe perforates contain 144 holes each.

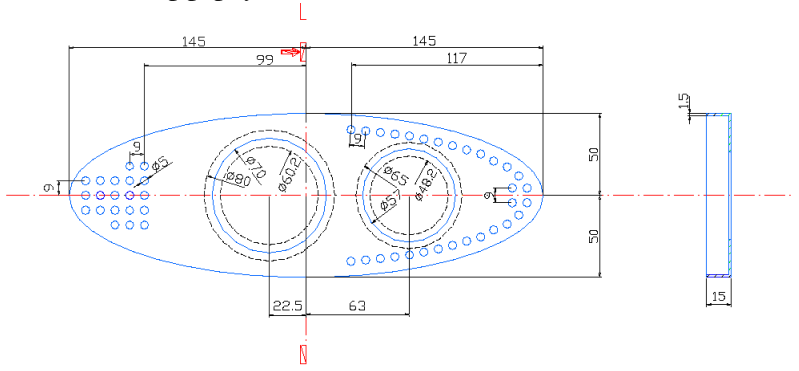


Figure 3: Baffle number 1, outlet side to the left and inlet side to the right.

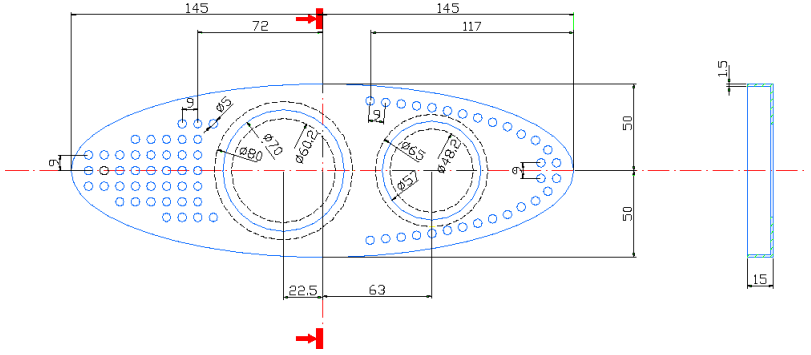


Figure 4: Baffles number 2 and 3.

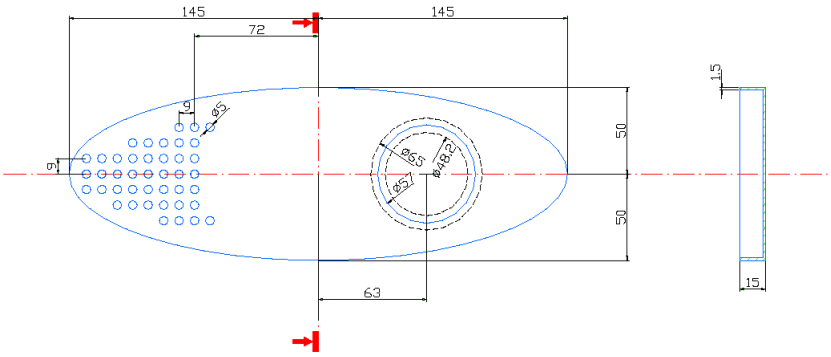


Figure 5: Baffle number 4.

The model geometry is supplied as a CAD file. To reduce computation time only the upper half of the muffler is included; a symmetry condition is added at $z = 0$. (A careful inspection of the drawings in Figure 3–Figure 5 shows that the reflection symmetry is not perfect for the perforates on the outlet side. However, the asymmetry is so minor that it is safe to neglect its effects.)

In the CAD geometry, the perforated regions are outlined by edges drawn on the corresponding boundaries. This is illustrated for baffle number 1 in Figure 6 where the perforated regions have been shaded for emphasis.

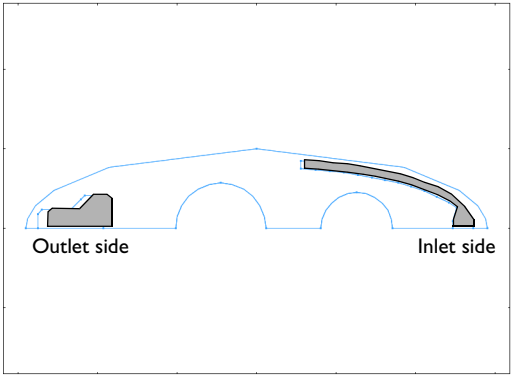


Figure 6: Perforated regions of baffle number 1.

You model the acoustic effects of the perforates by applying the Acoustics Module's Interior Perforated Plate boundary condition on these regions. A thorough description of

the perforated plate impedance models is available in Theory for the Interior Impedance Models section of The Pressure Acoustics, Frequency Domain Interface. The expression for the transfer impedance Z_i of the plate depends on the chosen model type, but is in general a function of the plate geometry, the porosity pattern, the material parameters of the fluid, and the wave number k. The last is given by $2\pi f/c$ where f denotes the frequency. You run the simulation for a range of frequencies between 20 Hz and 600 Hz.

The Interior Perforated Plate boundary condition also allows user defined contributions to the impedance. For example, you will specify the additional resistance caused by a mean flow in the muffler to include the effects of the metallic sleeve above pipe perforate number 1. For details about the effect of flow see Ref. 1, Ref. 2, and Ref. 3.

The relevant input parameters for the model are listed in Table 1. The porosity values were obtained by dividing the total area of the holes in each perforate with the area of the corresponding region in the CAD geometry. The resistance of the metallic sleeve was experimentally measured.

TABLE 1: MODEL INPUT PARAMETERS

PROPERTY	VALUE	DESCRIPTION
t_p	1.5 mm	Plate thickness
d_h	5 mm	Hole diameter in perforates
σ_p	0.22	Porosity, pipe perforates
σ_{bi}	0.46	Porosity, baffle perforates on inlet side
σ_{bo}	0.30	Porosity, baffle perforates on outlet side
θ_{sleeve}	1	Specific resistance, metallic sleeve

The parameters related to the material are taken from the surrounding fluid material data.

Results and Discussion

The transmission loss in the muffler is defined as

$$TL = 10\log_{10}\left(\frac{P_{in}}{P_{out}}\right) \tag{1}$$

where P_{in} and P_{out} denote the total acoustic power at the inlet and the outlet, respectively. The powers are defined by the surface integral of the (plane wave) intensity at the inlet and the outlet, respectively. The incident power and the power leaving the muffler are given by

$$P_{\text{in}} = 2\int_{S_5} \frac{p_0^2}{2\rho c} \qquad P_{\text{out}} = 2\int_{S_1} \frac{|p|^2}{2\rho c} \tag{2}$$

Figure 7 displays the Acoustics Module modeling results for the transmission loss as a function of sound frequency together with experimentally measured values.

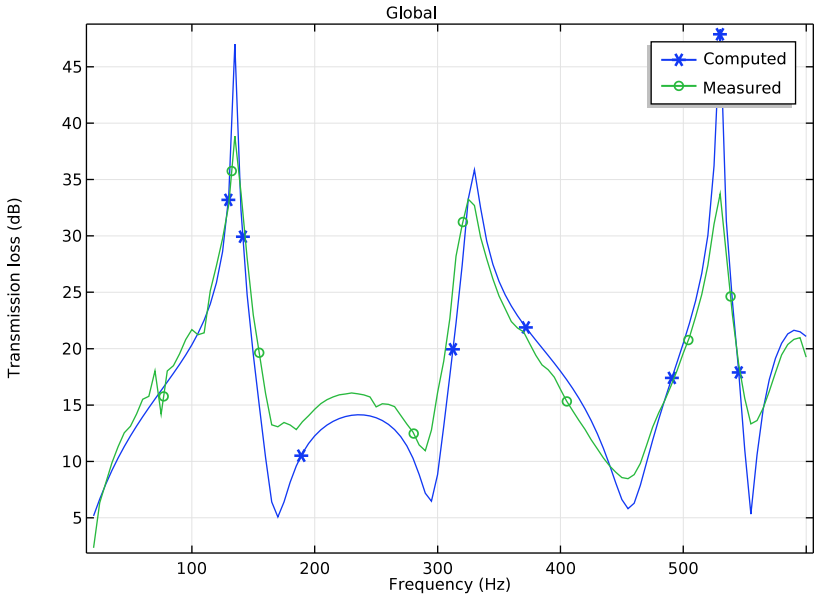


Figure 7: Transmission loss versus frequency: model simulation results and experimentally measured values.

As the figure shows, the agreement is excellent except in the range 170–300 Hz. The deviation here is presumably related to some coupled shell vibration that modifies the interior acoustic field. Shell vibrations can be included as illustrated in the Application Gallery model "Absorptive Muffler with Shells" (www.comsol.com/model/absorptive-muffler-with-shells-14717).

You can get a better sense of the results by studying the sound pressure level field inside the muffler for selected frequencies. The plots in Figure 8 display this field for the frequencies 530 Hz and 555 Hz, respectively. As Figure 7 shows, the former frequency corresponds to a local maximum for the transmission loss whereas the latter gives a local minimum. In Figure 8 you can see these how these properties are related to the sound

pressure level distributions near the muffler inlet and outlet.

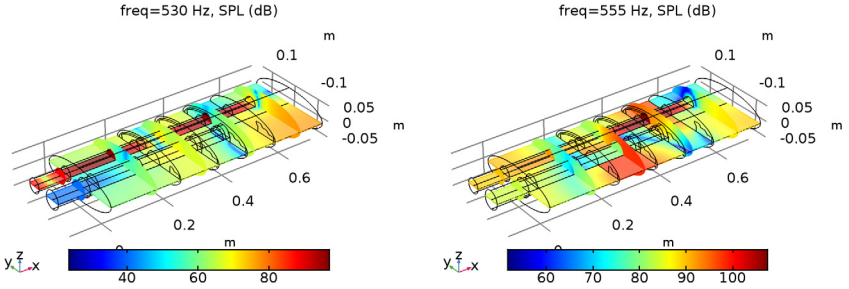


Figure 8: Sound pressure level distributions at 530 Hz (left) and 555 Hz (right).

The propagation of the acoustic energy through the muffler system is illustrated in Figure 9 as a streamline plot of the acoustic intensity vector.

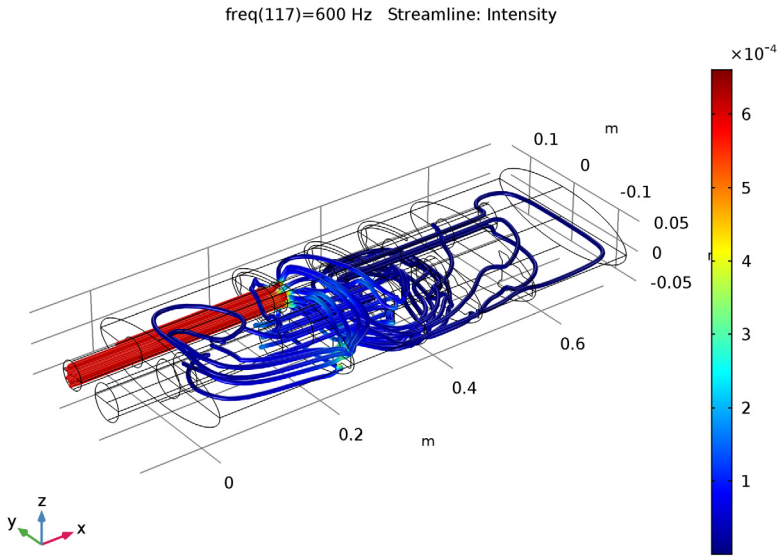


Figure 9: Streamline plot of the acoustic intensity field. The color scale represents the magnitude of the acoustic intensity vector.

References

1. E.J. Rice, "A Theoretical Study of the Acoustic Impedance of Orifices in the Presence of a Steady Grazing Flow," *NASA report TM X-71903*, 1976.

2. T. Elnady, *Modelling and Characterization of Perforates in Lined Ducts and Mufflers*, doctoral dissertation, Dept. Aeronautical and Vehicle Eng., Royal Institute of Technology, Stockholm, 2004.

3. R. Kirby, "Transmission Loss Predictions for Dissipative Silencers of Arbitrary Cross Section in the Presence of Mean Flow," *J. Aoust. Soc. Am.*, vol. 114, pp. 200–209, 2003.

Application Library path: `Acoustics_Module/Automotive/perforated_muffler`

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6. Click **Done**.

GEOMETRY 1

A horizontal symmetry plane through the muffler means that it is sufficient to model only half of the geometry. Here, the entire geometry is imported as a sequence from the geometry file. The instructions to the geometry can be found in the appendix at the end of this document.

1. On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `perforated_muffler_geom_sequence.mph`.

3 On the **Geometry** toolbar, click **Build All**.

You should now see the upper half of a muffler in the drawing area.

GLOBAL DEFINITIONS

Enter the parameters needed for the model or load them from the file perforated_muffler_parameters.txt.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
p0	1[Pa]	1 Pa	Inlet pressure
t_w	1.5[mm]	0.0015 m	Wall thickness
d_h	5[mm]	0.005 m	Hole diameter
sigma_p	0.22	0.22	Porosity, pipe perforates
sigma_bi	0.46	0.46	Porosity, baffles on inlet side
sigma_bo	0.3	0.3	Porosity, baffles on outlet side

DEFINITIONS

Make the following function, integral operators, and variable definitions so that you can compute the transmission loss and compare it with experimental data.

Interpolation 1 (int1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `perforated_muffler_exp_data.txt`.

6 Click **Import**.

7 In the **Function name** text field, type `TL_exp`.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_in` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Inlet**.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_out` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Outlet**.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

Defining your variables globally means that you can visualize them with a global plot. First, define the surface integrals given in Equation 2.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`P_in`	`intop_in(2*p0^2/(2*acpr.rho*acpr.c))`	W	`Total acoustic power at the inlet`
`P_out`	`intop_out(2*p*conj(p)/(2*acpr.rho*acpr.c))`	W	`Total acoustic power, outlet`
`TL`	`10*log10(P_in/P_out)`		`Transmission loss`

As you defined them earlier, the integral operators `intop_in()` and `intop_out()` means integration over the inlet and outlet respectively. The transmission loss is defined as a logarithmic measure to allow direct comparison with the experimental results.

MATERIALS

The walls of the muffler are considered perfectly rigid and will not use any material properties. The only material you need to select is air.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

With all selections, variables, and material settings completed, it is time to set up the physics.

All boundaries that are exterior to the geometry are by default assigned the Sound Hard boundary condition. Start by applying the Interior Sound Hard condition to the interior rigid walls.

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 In the **Settings** window for **Interior Sound Hard Boundary (Wall)**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Interior Hard Walls**.

You need four Interior Perforated Plate conditions, one for each different set of parameters. The Thin plate (default) model type is appropriate here since the value kt_p is much less than 1.

Interior Perforated Plate 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Perforated Plate**.

2 In the **Settings** window for **Interior Perforated Plate**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pipe Perforates**.

4 Locate the **Interior Perforated Plate** section. In the σ text field, type `sigma_p`.

5 In the t_p text field, type `t_w`.

6 In the d_h text field, type `d_h`.

7 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Air (mat1)**.

Interior Perforated Plate 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Perforated Plate**.

2 In the **Settings** window for **Interior Perforated Plate**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Pipe Perforate with Sleeve**.

4 Locate the **Interior Perforated Plate** section. In the σ text field, type `sigma_p`.

5 In the t_p text field, type `t_w`.

6 In the d_h text field, type `d_h`.

7 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Air (mat1)**.

8 Locate the **Interior Perforated Plate** section. Select the **User-defined contribution** check box.

9 In the $\theta^{(user)}$ text field, type `1`.

The user-defined resistance represents the impedance contribution from the metallic sleeve outside the pipe.

Interior Perforated Plate 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Perforated Plate**.

2 In the **Settings** window for **Interior Perforated Plate**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlet Baffle Perforates**.

4 Locate the **Interior Perforated Plate** section. In the σ text field, type `sigma_bi`.

5 In the t_p text field, type `t_w`.

6 In the d_h text field, type `d_h`.

7 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Air (mat1)**.

Interior Perforated Plate 4

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Perforated Plate**.

2 In the **Settings** window for **Interior Perforated Plate**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet Baffle Perforates**.

4 Locate the **Interior Perforated Plate** section. In the σ text field, type `sigma_bo`.

5 In the t_p text field, type `t_w`.

6 In the d_h text field, type `d_h`.

7 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Air (mat1)**.

Use the transparent Plane Wave Radiation condition at the inlet and outlet, with an added incident pressure field at the inlet.

Plane Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 In the **Settings** window for **Plane Wave Radiation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlet**.

Incident Pressure Field 1

1 On the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 In the **Settings** window for **Incident Pressure Field**, locate the **Incident Pressure Field** section.

3 In the p_0 text field, type `p0`.

4 From the *c* list, choose **From material**.

5 From the **Material** list, choose **Air (mat1)**.

Plane Wave Radiation 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Plane Wave Radiation**.

2 In the **Settings** window for **Plane Wave Radiation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet**.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

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 `343[m/s]/600[Hz]/8`.

You will solve this model for a range of frequencies from 20 Hz to 600 Hz. The global maximum element size is set equal to the minimal wavelength divided by 8, that is, $\lambda/8 = c/f_{\max}/8$, where c is the speed of sound.

5 Click **Build All**.

STUDY 1

Select to solve for the frequency range from 20 Hz to 600 Hz, in steps of 5 Hz.

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then 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 `range(20,5,600)`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

Create a Mirror 3D data set in order to visualize the solution on the full geometry.

Mirror 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 From the **Plane** list, choose **XY-planes**.

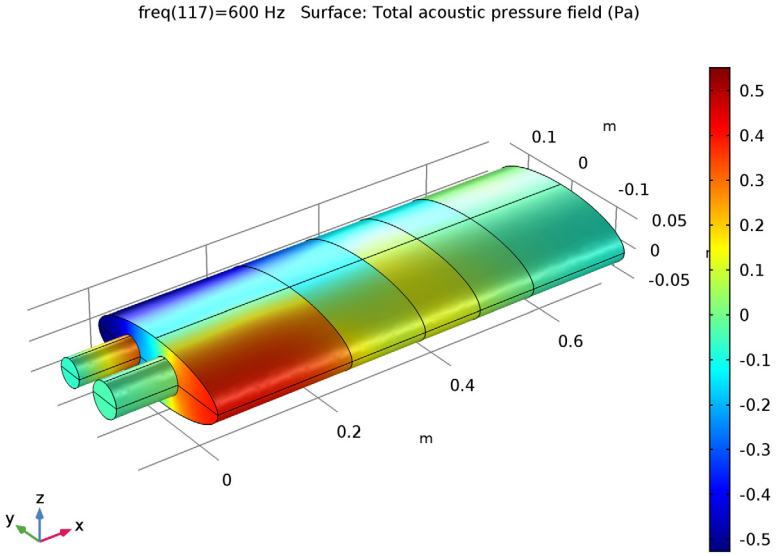
Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Mirror 3D 1**.

4 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.



The plot shows the pressure distribution on the surface of the muffler. Follow the instructions to see the sound pressure level on a slice inside it.

Surface 1

1 In the **Model Builder** window, expand the **Results>Sound Pressure Level (acpr)** node.

2 Right-click **Surface 1** and choose **Disable**.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Mirror 3D 1**.

Slice 1

1 Right-click **Results>Sound Pressure Level (acpr)** and choose **Slice**.

2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

4 In the **Planes** text field, type 1.

5 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

Slice 2

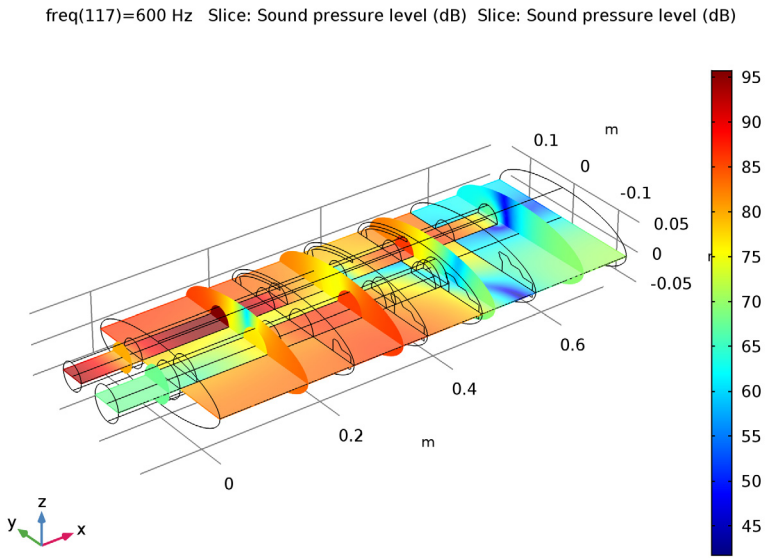
1 Right-click **Sound Pressure Level (acpr)** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.Lp`.

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

5 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.



At 600 Hz, most of the sound leaves the inlet pipe through the perforate. Take a look at some other frequencies to reproduce Figure 8.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **530**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `freq=530 Hz, SPL (dB)`.

6 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

7 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

8 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **555**.

9 Locate the **Title** section. From the **Title type** list, choose **Manual**.

10 In the **Title** text area, type `freq=555 Hz, SPL (dB)`.

11 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

Acoustic Pressure, Isosurfaces (acpr)

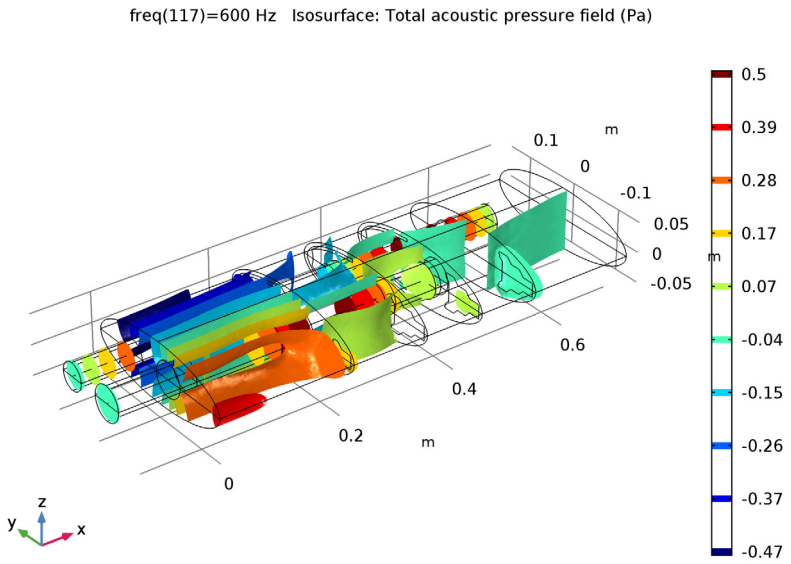
1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Isosurfaces (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Mirror 3D 1**.

4 On the **Acoustic Pressure, Isosurfaces (acpr)** toolbar, click **Plot**.

The last default plot, depicting pressure isosurfaces should look like the figure below.



To study the transmission loss as a function of the frequency and reproduce Figure 7, add a 1D plot.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Transmission Loss` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **x-axis label** check box.

4 In the associated text field, type `Frequency (Hz)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Transmission loss (dB)`.

Global 1

1 Right-click **Transmission Loss** 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
TL		Transmission loss
TL_exp(freq)		

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
Computed
Measured

7 On the **Transmission Loss** toolbar, click **Plot**.

Finally, create a streamline plot of the acoustic intensity vector (SI unit: $\mathrm{W/m^2}$ in order to visualize the direction of propagation of the acoustic energy in the muffler system.

3D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Intensity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Mirror 3D 1**.

Streamline 1

1 Right-click **Intensity** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Intensity>acpr.Ix,acpr.Iy,acpr.Iz - Intensity**.

3 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

Color Expression 1

1 Right-click **Results>Intensity>Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1> Pressure Acoustics, Frequency Domain>Intensity>acpr.I_mag - Intensity magnitude**.

3 On the **Intensity** toolbar, click **Plot**.

The result should look like the plot in Figure 9.

Appendix: Geometry Sequence Instructions

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

GEOMETRY 1

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1> Work Plane 1 (wp1)** click **Plane Geometry**.

Ellipse 1 (e1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Ellipse**.

2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `0.145`.

4 In the **b-semiaxis** text field, type `0.05`.

5 In the **Sector angle** text field, type `180`.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
0.225
0.225+0.128
0.225+0.128+0.098
0.225+0.128+0.098+0.098
0.225+0.128+0.098+0.098+0.171

Work Plane 2 (wp2)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 In the **x-coordinate** text field, type `-0.1`.

5 Locate the **Unite Objects** section. Clear the **Unite objects** check box.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1> Work Plane 2 (wp2)** click **Plane Geometry**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `28.5[mm]`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **xw** text field, type `-22.5[mm]`.

Circle 2 (c2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `22.5[mm]`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **xw** text field, type `63[mm]`.

Work Plane 2 (wp2)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 2 (wp2)**.

Extrude 2 (ext2)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **General** section.

3 In the list, select **wp2.c2**.

4 Click **Remove from Selection**.

5 Select the object **wp2.c1** only.

6 Locate the **Distances** section. In the table, enter the following settings:

Distances (m)
0.1
0.128
0.272
0.325
0.453
0.47
0.542
0.551
0.564

Extrude 3 (ext3)

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
0.1
0.325
0.347
0.437
0.453
0.551
0.649
0.73

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Work Plane 3 (wp3)

1 On the **Geometry** toolbar, click **Work Plane**.

2 Click the **Select Boundaries** button on the **Graphics** toolbar.

3 Click the **Click and Hide** button on the **Graphics** toolbar.

4 On the object **ext1**, select Boundaries 18, 13, 8, 3, 5, 10, 15, and 20 only.

5 Click the **Click and Hide** button on the **Graphics** toolbar.

6 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

7 From the **Plane type** list, choose **Face parallel**.

8 On the object **ext1**, select Boundary 6 only.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1> Work Plane 3 (wp3)** click **Plane Geometry**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `20.5[mm]`.

4 In the **Height** text field, type `11.5[mm]`.

5 Locate the **Position** section. In the **xw** text field, type `-0.1375`.

6 In the **yw** text field, type `-25[mm]`.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `20.5[mm]`.

4 In the **Height** text field, type `20.5[mm]`.

5 Locate the **Position** section. In the **xw** text field, type `-0.117`.

6 In the **yw** text field, type `-25[mm]`.

Union 1 (uni1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

Chamfer 1 (cha1)

1 On the **Work Plane** toolbar, click **Chamfer**.

2 On the object **uni1**, select Points 2 and 7 only.

3 In the **Settings** window for **Chamfer**, locate the **Distance** section.

4 In the **Distance from vertex** text field, type `2.5[mm]`.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `-117[mm]`.

5 In row **1**, set **yw** to `-13.5[mm]`.

6 In row **2**, set **xw** to `-117[mm]`.

7 In row **2**, set **yw** to `-4.5[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **xw** to `-108[mm]`.

Difference 1 (dif1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **cha1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **b1** only.

Bézier Polygon 2 (b2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `136.5[mm]`.

5 In row **1**, set **yw** to `-25[mm]`.

6 In row **2**, set **xw** to `136.5[mm]`.

7 In row **2**, set **yw** to `-20.5[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **xw** to `131[mm]`.

10 In row **2**, set **yw** to `-12.2[mm]`.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **xw** to `121[mm]`.

13 In row **2**, set **yw** to `-4[mm]`.

14 Find the **Added segments** subsection. Click **Add Linear**.

15 Find the **Control points** subsection. In row **2**, set **xw** to `113.5[mm]`.

16 In row **2**, set **yw** to `-0.1[mm]`.

17 Find the **Added segments** subsection. Click **Add Linear**.

18 Find the **Control points** subsection. In row **2**, set **xw** to `105.5[mm]`.

19 In row **2**, set **yw** to `3.3[mm]`.

20 Find the **Added segments** subsection. Click **Add Linear**.

21 Find the **Control points** subsection. In row **2**, set **xw** to `97.3[mm]`.

22 In row **2**, set **yw** to `6[mm]`.

23 Find the **Added segments** subsection. Click **Add Linear**.

24 Find the **Control points** subsection. In row **2**, set **xw** to `90[mm]`.

25 In row **2**, set **yw** to `8[mm]`.

26 Find the **Added segments** subsection. Click **Add Linear**.

27 Find the **Control points** subsection. In row **2**, set **xw** to `80.8[mm]`.

28 In row **2**, set **yw** to `10.3[mm]`.

29 Find the **Added segments** subsection. Click **Add Linear**.

30 Find the **Control points** subsection. In row **2**, set **xw** to `72.4[mm]`.

31 In row **2**, set **yw** to `12[mm]`.

32 Find the **Added segments** subsection. Click **Add Linear**.

33 Click **Add Linear**.

34 Find the **Control points** subsection. In row **2**, set **xw** to `60[mm]`.

35 In row **2**, set **yw** to `14[mm]`.

36 Find the **Added segments** subsection. Click **Add Linear**.

37 Find the **Control points** subsection. In row **2**, set **xw** to `55[mm]`.

38 In row **2**, set **yw** to `15[mm]`.

39 Find the **Added segments** subsection. Click **Add Linear**.

40 Find the **Control points** subsection. In row **2**, set **xw** to `45.9[mm]`.

41 In row **2**, set **yw** to `15.7[mm]`.

42 Find the **Added segments** subsection. Click **Add Linear**.

43 Find the **Control points** subsection. In row **2**, set **xw** to `37[mm]`.

44 In row **2**, set **yw** to `16.8[mm]`.

45 Find the **Added segments** subsection. Click **Add Linear**.

46 Find the **Control points** subsection. In row **2**, set **xw** to `28[mm]`.

47 In row **2**, set **yw** to `17.5[mm]`.

48 Find the **Added segments** subsection. Click **Add Linear**.

49 Find the **Control points** subsection. In row **2**, set **yw** to `12.5[mm]`.

50 Find the **Added segments** subsection. Click **Add Linear**.

51 Find the **Control points** subsection. In row **2**, set **xw** to `37[mm]`.

52 In row **2**, set **yw** to `11.8[mm]`.

53 Find the **Added segments** subsection. Click **Add Linear**.

54 Find the **Control points** subsection. In row **2**, set **xw** to `45.9[mm]`.

55 In row **2**, set **yw** to `10.7[mm]`.

56 Find the **Added segments** subsection. Click **Add Linear**.

57 Find the **Control points** subsection. In row **2**, set **xw** to `55[mm]`.

58 In row **2**, set **yw** to `10[mm]`.

59 Find the **Added segments** subsection. Click **Add Linear**.

60 Find the **Control points** subsection. In row **2**, set **xw** to `63.8[mm]`.

61 In row **2**, set **yw** to `8.5[mm]`.

62 Find the **Added segments** subsection. Click **Add Linear**.

63 Find the **Control points** subsection. In row **2**, set **xw** to `72.4[mm]`.

64 In row **2**, set **yw** to `7[mm]`.

65 Find the **Added segments** subsection. Click **Add Linear**.

66 Find the **Control points** subsection. In row **2**, set **xw** to `80.8[mm]`.

67 In row **2**, set **yw** to `5.3[mm]`.

68 Find the **Added segments** subsection. Click **Add Linear**.

69 Find the **Control points** subsection. In row **2**, set **xw** to `89[mm]`.

70 In row **2**, set **yw** to `3.5[mm]`.

71 Find the **Added segments** subsection. Click **Add Linear**.

72 Find the **Control points** subsection. In row **2**, set **xw** to `97.3[mm]`.

73 In row **2**, set **yw** to `1[mm]`.

74 Find the **Added segments** subsection. Click **Add Linear**.

75 Find the **Control points** subsection. In row **2**, set **xw** to `105.5[mm]`.

76 In row **2**, set **yw** to `-1.7[mm]`.

77 Find the **Added segments** subsection. Click **Add Linear**.

78 Find the **Control points** subsection. In row **2**, set **xw** to `113.5[mm]`.

79 In row **2**, set **yw** to `-5.1[mm]`.

80 Find the **Added segments** subsection. Click **Add Linear**.

81 Find the **Control points** subsection. In row **2**, set **xw** to `121[mm]`.

82 In row **2**, set **yw** to `-9[mm]`.

83 Find the **Added segments** subsection. Click **Add Linear**.

84 Find the **Control points** subsection. In row **2**, set **xw** to `126[mm]`.

85 In row **2**, set **yw** to `-12.2[mm]`.

86 Find the **Added segments** subsection. Click **Add Linear**.

87 Find the **Control points** subsection. In row **2**, set **xw** to `123.3[mm]`.

88 In row **2**, set **yw** to `-20.5[mm]`.

89 Find the **Added segments** subsection. Click **Add Linear**.

90 Find the **Control points** subsection. In row **2**, set **yw** to `-25[mm]`.

91 Find the **Added segments** subsection. Click **Add Linear**.

92 Find the **Control points** subsection. In row **2**, set **xw** to `136.5[mm]`.

Work Plane 3 (wp3)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 3 (wp3)**.

Work Plane 4 (wp4)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 From the **Plane type** list, choose **Face parallel**.

5 On the object **ext1**, select Boundary 11 only.

6 In the tree, select **ext1**.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1> Work Plane 4 (wp4)** click **Plane Geometry**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `20.5[mm]`.

4 In the **Height** text field, type `11.5[mm]`.

5 Locate the **Position** section. In the **xw** text field, type `-0.1375`.

6 In the **yw** text field, type `-25[mm]`.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `20.5[mm]`.

4 In the **Height** text field, type `20.5[mm]`.

5 Locate the **Position** section. In the **xw** text field, type `-0.117`.

6 In the **yw** text field, type `-25[mm]`.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `27[mm]`.

4 In the **Height** text field, type `20.5[mm]`.

5 Locate the **Position** section. In the **xw** text field, type `-96.5[mm]`.

6 In the **yw** text field, type `-25[mm]`.

Rectangle 4 (r4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `11.5`.

4 In the **Height** text field, type `9`.

5 Locate the **Position** section. In the **xw** text field, type `-81`.

6 In the **yw** text field, type `-4.5[mm]`.

7 In the **xw** text field, type `-81[mm]`.

8 Locate the **Size and Shape** section. In the **Height** text field, type `9[mm]`.

9 In the **Width** text field, type `11.5[mm]`.

Rectangle 5 (r5)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `6.5[mm]`.

4 In the **Height** text field, type `13.5[mm]`.

5 Locate the **Position** section. In the **xw** text field, type `-69.5[mm]`.

6 In the **yw** text field, type `-9[mm]`.

Bézier Polygon 1 (b1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `-117[mm]`.

5 In row **1**, set **yw** to `-13.5[mm]`.

6 In row **2**, set **xw** to `-117[mm]`.

7 In row **2**, set **yw** to `-4.5[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **xw** to `-108[mm]`.

Bézier Polygon 2 (b2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `136.5[mm]`.

5 In row **1**, set **yw** to `-25[mm]`.

6 In row **2**, set **xw** to `136.5[mm]`.

7 In row **2**, set **yw** to `-20.5[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **xw** to `131[mm]`.

10 In row **2**, set **yw** to `-12.2[mm]`.

11 Find the **Added segments** subsection. Click **Add Linear**.

12 Find the **Control points** subsection. In row **2**, set **xw** to `121[mm]`.

13 In row **2**, set **yw** to `-4[mm]`.

14 Find the **Added segments** subsection. Click **Add Linear**.

15 Find the **Control points** subsection. In row **2**, set **xw** to `113.5[mm]`.

16 In row **2**, set **yw** to `-0.1[mm]`.

17 Find the **Added segments** subsection. Click **Add Linear**.

18 Find the **Control points** subsection. In row **2**, set **xw** to `105.5[mm]`.

19 In row **2**, set **yw** to `3.3[mm]`.

20 Find the **Added segments** subsection. Click **Add Linear**.

21 Find the **Control points** subsection. In row **2**, set **xw** to `97.3[mm]`.

22 In row **2**, set **yw** to `6[mm]`.

23 Find the **Added segments** subsection. Click **Add Linear**.

24 Find the **Control points** subsection. In row **2**, set **xw** to `90[mm]`.

25 In row **2**, set **yw** to `8[mm]`.

26 Find the **Added segments** subsection. Click **Add Linear**.

27 Find the **Control points** subsection. In row **2**, set **xw** to `80.8[mm]`.

28 In row **2**, set **yw** to `10.3[mm]`.

29 Find the **Added segments** subsection. Click **Add Linear**.

30 Find the **Control points** subsection. In row **2**, set **xw** to `72.4[mm]`.

31 In row **2**, set **yw** to `12[mm]`.

32 Find the **Added segments** subsection. Click **Add Linear**.

33 Click **Add Linear**.

34 Find the **Control points** subsection. In row **2**, set **xw** to `60[mm]`.

35 In row **2**, set **yw** to `14[mm]`.

36 Find the **Added segments** subsection. Click **Add Linear**.

37 Find the **Control points** subsection. In row **2**, set **xw** to `55[mm]`.

38 In row **2**, set **yw** to `15[mm]`.

39 Find the **Added segments** subsection. Click **Add Linear**.

40 Find the **Control points** subsection. In row **2**, set **xw** to `45.9[mm]`.

41 In row **2**, set **yw** to `15.7[mm]`.

42 Find the **Added segments** subsection. Click **Add Linear**.

43 Find the **Control points** subsection. In row **2**, set **xw** to `37[mm]`.

44 In row **2**, set **yw** to `16.8[mm]`.

45 Find the **Added segments** subsection. Click **Add Linear**.

46 Find the **Control points** subsection. In row **2**, set **xw** to `28[mm]`.

47 In row **2**, set **yw** to `17.5[mm]`.

48 Find the **Added segments** subsection. Click **Add Linear**.

49 Find the **Control points** subsection. In row **2**, set **yw** to `12.5[mm]`.

50 Find the **Added segments** subsection. Click **Add Linear**.

51 Find the **Control points** subsection. In row **2**, set **xw** to `37[mm]`.

52 In row **2**, set **yw** to `11.8[mm]`.

53 Find the **Added segments** subsection. Click **Add Linear**.

54 Find the **Control points** subsection. In row **2**, set **xw** to `45.9[mm]`.

55 In row **2**, set **yw** to `10.7[mm]`.

56 Find the **Added segments** subsection. Click **Add Linear**.

57 Find the **Control points** subsection. In row **2**, set **xw** to `55[mm]`.

58 In row **2**, set **yw** to `10[mm]`.

59 Find the **Added segments** subsection. Click **Add Linear**.

60 Find the **Control points** subsection. In row **2**, set **xw** to `63.8[mm]`.

61 In row **2**, set **yw** to `8.5[mm]`.

62 Find the **Added segments** subsection. Click **Add Linear**.

63 Find the **Control points** subsection. In row **2**, set **xw** to `72.4[mm]`.

64 In row **2**, set **yw** to `7[mm]`.

65 Find the **Added segments** subsection. Click **Add Linear**.

66 Find the **Control points** subsection. In row **2**, set **xw** to `80.8[mm]`.

67 In row **2**, set **yw** to `5.3[mm]`.

68 Find the **Added segments** subsection. Click **Add Linear**.

69 Find the **Control points** subsection. In row **2**, set **xw** to `89[mm]`.

70 In row **2**, set **yw** to `3.5[mm]`.

71 Find the **Added segments** subsection. Click **Add Linear**.

72 Find the **Control points** subsection. In row **2**, set **xw** to `97.3[mm]`.

73 In row **2**, set **yw** to `1[mm]`.

74 Find the **Added segments** subsection. Click **Add Linear**.

75 Find the **Control points** subsection. In row **2**, set **xw** to `105.5[mm]`.

76 In row **2**, set **yw** to `-1.7[mm]`.

77 Find the **Added segments** subsection. Click **Add Linear**.

78 Find the **Control points** subsection. In row **2**, set **xw** to `113.5[mm]`.

79 In row **2**, set **yw** to `-5.1[mm]`.

80 Find the **Added segments** subsection. Click **Add Linear**.

81 Find the **Control points** subsection. In row **2**, set **xw** to `121[mm]`.

82 In row **2**, set **yw** to `-9[mm]`.

83 Find the **Added segments** subsection. Click **Add Linear**.

84 Find the **Control points** subsection. In row **2**, set **xw** to `126[mm]`.

85 In row **2**, set **yw** to `-12.2[mm]`.

86 Find the **Added segments** subsection. Click **Add Linear**.

87 Find the **Control points** subsection. In row **2**, set **xw** to `123.3[mm]`.

88 In row **2**, set **yw** to `-20.5[mm]`.

89 Find the **Added segments** subsection. Click **Add Linear**.

90 Find the **Control points** subsection. In row **2**, set **yw** to `-25[mm]`.

91 Find the **Added segments** subsection. Click **Add Linear**.

92 Find the **Control points** subsection. In row **2**, set **xw** to `136.5[mm]`.

Bézier Polygon 3 (b3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `-90[mm]`.

5 In row **1**, set **yw** to `-4.5[mm]`.

6 In row **2**, set **xw** to `-81[mm]`.

7 In row **2**, set **yw** to `-4.5[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **yw** to `4.5[mm]`.

Bézier Polygon 4 (b4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `-63[mm]`.

5 In row **1**, set **yw** to `4.5[mm]`.

6 In row **2**, set **xw** to `-60.5[mm]`.

7 In row **2**, set **yw** to `2[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **xw** to `-63[mm]`.

10 In row **2**, set **yw** to `-0.5[mm]`.

Bézier Polygon 5 (b5)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `-69.5[mm]`.

5 In row **1**, set **yw** to `-9[mm]`.

6 In row **2**, set **xw** to `-63[mm]`.

7 In row **2**, set **yw** to `-9[mm]`.

8 Find the **Added segments** subsection. Click **Add Linear**.

9 Find the **Control points** subsection. In row **2**, set **yw** to `-3.5[mm]`.

Bézier Polygon 6 (b6)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Bézier Polygon**.

2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click **Add Linear**.

4 Find the **Control points** subsection. In row **1**, set **xw** to `-137.5[mm]`.

5 In row **1**, set **yw** to `-13.5[mm]`.

6 In row **2**, set **xw** to `-135[mm]`.

7 In row **2**, set **yw** to `-13.5[mm]`.

Union 1 (uni1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **r3**, **b4**, **r1**, **r2**, **r4**, **b3**, and **r5** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

Difference 1 (dif1)

1 On the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the objects **b1**, **b6**, and **b5** only.

Work Plane 4 (wp4)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 4 (wp4)**.

Work Plane 5 (wp5)

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 4 (wp4)** and choose **Duplicate**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 Find the **Planar face** subsection. Select the **Active** toggle button.

4 In the tree, select **ext1**.

5 On the object **ext1**, select Boundary 16 only.

6 Click **Build Selected**.

Work Plane 6 (wp6)

1 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 5 (wp5)** and choose **Duplicate**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the tree, select **ext1**.

4 On the object **ext1**, select Boundary 21 only.

5 Click **Build Selected**.

6 In the **Model Builder** window, expand the **Work Plane 6 (wp6)** node.

Bézier Polygon 2 (b2)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1> Work Plane 6 (wp6)>Plane Geometry** node, then click **Bézier Polygon 2 (b2)**.

2 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 6 (wp6)>Plane Geometry>Bézier Polygon 2 (b2)** and choose **Disable**.

Work Plane 6 (wp6)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 6 (wp6)**.

2 In the **Settings** window for **Work Plane**, click **Build Selected**.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **ext3** and **ext2** only.

3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** check box.

Union 2 (uni2)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **ext1**, **wp5**, **wp6**, **wp3**, and **wp4** only.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **uni2** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Select the **Keep input objects** check box.

5 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

6 Select the object **uni1** only.

7 Click **Build Selected**.

8 On the **Geometry** toolbar, click **Delete**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Delete Entities 1 (del1)**.

2 Select the object **uni2** only.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Explicit Selection 1 (sel1)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

   This section asks you to define a number of boundary selections to use when setting up the physics. To make it easier to select the correct boundaries, you will open the Selection List, where all boundaries are available by numbers. When making your selections, you can Ctrl-click in this list to select several boundaries at once, and use the plus button on the Settings tab to add them to your selection. Note that when working on your own models, it is usually more convenient to make selections by clicking in the geometry.

2 In the **Settings** window for **Explicit Selection**, type `Inlet` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 6 only.

Explicit Selection 2 (sel2)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Outlet` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundary 1 only.

Explicit Selection 3 (sel3)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Pipe Perforates` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 50, 52, 74, and 76 only.

Explicit Selection 4 (sel4)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Pipe Perforate with Sleeve` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 26 and 28 only.

Explicit Selection 5 (sel5)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Inlet Baffle Perforates` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 43, 67, and 91 only.

Explicit Selection 6 (sel6)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Outlet Baffle Perforates` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 36, 60, 84, and 101 only.

Explicit Selection 7 (sel7)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Pipe Ends` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **fin**, select Boundaries 97 and 108 only.

Explicit Selection 8 (sel8)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Geometry` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Object**.

4 Select the object **fin** only.

Adjacent Selection 1 (adjsel1)

1 On the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.

3 Click **Add**.

4 In the **Add** dialog box, select **Geometry** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

7 Clear the **Exterior boundaries** check box.

8 Select the **Interior boundaries** check box.

9 In the **Label** text field, type `Walls`.

Difference Selection 1 (difsel1)

1 On the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, locate the **Geometric Entity Level** section.

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, select **Walls** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click **Add**.

9 In the **Add** dialog box, in the **Selections to subtract** list, choose **Pipe Perforates**, **Pipe Perforate with Sleeve**, **Inlet Baffle Perforates**, **Outlet Baffle Perforates**, and **Pipe Ends**.

10 Click **OK**.

11 In the **Settings** window for **Difference Selection**, type `Interior Hard Walls` in the **Label** text field.

Box Selection 1 (boxsel1)

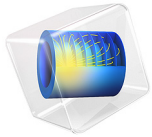
1 On the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Symmetry` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **z maximum** text field, type `0.01`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.



Spherical Piezoacoustic Transducer

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The piezoelectric effect describes an induced strain caused by an electric field in certain ferroelectric materials that are poled along a specific direction. Piezoelectric materials are composed of ferroelectric domains which are initially randomly oriented. The poling process creates a preferred orientation of these domains. In the presence of an electric field, these domains rotate from the poled direction thereby producing strain in the material.

Piezoelectric materials are widely used as actuator cores in acoustic transducers. In such devices, the piezoelectric material is excited with an electrical input, typically at high frequencies (kHz to MHz range). The harmonic electrical excitation produces structural vibrations in the material which in turn set up acoustic waves in the surrounding fluid media. This principle is used in several applications such as hydrophones, ultrasound imaging and non-destructive testing.

In this tutorial, you learn how to model the acoustic waves generated in air by a hollow spherical piezoelectric material which is poled along the radial direction of the sphere. Since the direction of poling imparts anisotropy to the material response, it is critical to incorporate it correctly in the simulation.

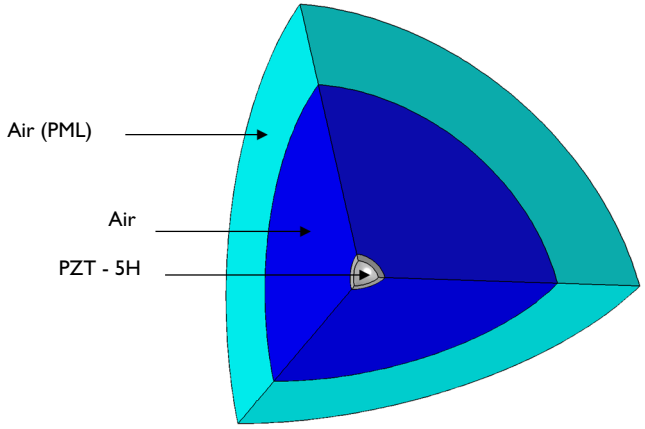


Figure 1: Pictorial representation of a 1/8 th symmetric section of a hollow, spherical, piezoelectric domain made of PZT-5H surrounded by a region of air that is infinitely extended. An additional geometric layer is used to set up a perfectly matched layer (PML). PMLs are used to efficiently absorb outgoing waves.

In this model, the geometry and mesh are parametrized with respect to the excitation frequency and speed of sound in the fluid medium. The model assumes the fluid to be air and the speed of sound to be 343 m/s. The inner radius and thickness of the hollow sphere are also parametrized; the model considers their values to be 2.5 mm and 1 mm, respectively. The inner air domain region is parametrized to be twice the acoustic wavelength, while the thickness of the PML is set equal to the wavelength. These settings always allow you to capture two stationary waves in the fluid domain irrespective of the dimension of the transducer, excitation frequency, and speed of sound in the fluid.

PHYSICS AND BOUNDARY CONDITIONS

This tutorial shows how to set up an acoustic-structure interaction model. The structure is a piezoelectric material (PZT-5H) and the surrounding fluid medium is air. The model uses the built-in Acoustic-Piezoelectric Interaction, Frequency Domain multiphysics interface. Here, you solve the Solid Mechanics and Electrostatics equations in the piezoelectric material, which are coupled via the constitutive equations for the piezo material. This coupling is taken care of by the Piezoelectric Effect coupling feature that is located under the Multiphysics node.

The Pressure Acoustics equation is solved in the fluid domain only. Once you demarcate the solid and fluid regions, the boundary condition at the common boundary between the fluid and the piezoelectric solid material is taken care of automatically by the Acoustic-Structure Boundary cooling feature located under the Multiphysics node. At this common boundary, the normal component of the acceleration of the piezo material acts as a sound source for the fluid, while the fluid pressure acts as a boundary load on the piezo material.

The inner surface of the hollow spherical piezo region is assumed to be at electrical ground, while a 100 V (zero-to-peak) potential is applied to its outer surface. The excitation frequency is 25 kHz. A spherical PML is used as the outer layer of the air domain to model absorption of outgoing waves as they propagate infinitely far away from the sound source. For more details on PMLs and the Acoustic-Piezoelectric Interaction, Frequency Domain interface, refer to the *Acoustics Module User's Guide*.

Due to the symmetry of both the geometry and physics set-up, only a 1/8 th section of the geometry is used for modeling purposes. In the Pressure Acoustics use the Symmetry condition and in the Electrostatics interfaces the default and Zero Charge boundary conditions work as the appropriate symmetry boundary conditions. For a Solid Mechanics interface, a Symmetry boundary condition is also used, which sets the normal component

of the structural displacement to zero on each of the boundaries that align with the three different planes of symmetry.

IMPLEMENTING THE POLING DIRECTION

In COMSOL Multiphysics, the poling direction of a piezoelectric material is decided based on the choice of coordinate system, in which the material properties are evaluated. By default, the material coordinate system of the piezoelectric material is assumed to be aligned with the global coordinate system, which is Cartesian. Thus, by default, most piezo materials are assumed to be z-polarized. This also means that if you assign the piezo material properties to a user-defined orthogonal coordinate system with unit vectors **x1**, **x2**, and **x3**, then the piezo can be considered to be poled along the **x3** direction. You can use this idea to create a user-defined spherical coordinate system, where **x3** is aligned along the radial direction of the piezo sphere.

Note: COMSOL Multiphysics provides a built-in option for setting up a spherical coordinate system. However, this built-in option assumes that the radial direction is the **x1** direction of such a local coordinate system. Hence, for this model you need to create a custom spherical coordinate system.

A rectangular coordinate system is identified by three mutually perpendicular unit vectors denoted by $\mathbf{e}_x$, $\mathbf{e}_y$, and $\mathbf{e}_z$. A spherical coordinate system is similarly identified by the unit vectors $\mathbf{e}_r$, $\mathbf{e}_\theta$, and $\mathbf{e}_\varphi$, which denote the radial, polar, and azimuthal directions, respectively. The next step is to find a relationship between these two sets of unit vectors using the spatial coordinates (r, θ, φ) of the spherical coordinate system. Equation 1 shows this relationship

$$\begin{aligned} \mathbf{e}_r &= \cos\varphi\sin\theta\mathbf{e}_x + \sin\varphi\sin\theta\mathbf{e}_y + \cos\theta\mathbf{e}_z \\ \mathbf{e}_\theta &= \cos\varphi\cos\theta\mathbf{e}_x + \sin\varphi\cos\theta\mathbf{e}_y - \sin\theta\mathbf{e}_z \\ \mathbf{e}_\varphi &= -\sin\varphi\mathbf{e}_x + \cos\varphi\mathbf{e}_y + 0\mathbf{e}_z \end{aligned} \tag{1}$$

Using this equation, you can create a user-defined spherical coordinate system, whose unit vectors, **x1**, **x2**, and **x3**, can be related to $\mathbf{e}_x$, $\mathbf{e}_y$, and $\mathbf{e}_z$ as shown in Equation 2.

$$\begin{aligned} \mathbf{x1} &= \cos\varphi\cos\theta\mathbf{e}_x + \sin\varphi\cos\theta\mathbf{e}_y - \sin\theta\mathbf{e}_z \\ \mathbf{x2} &= -\sin\varphi\mathbf{e}_x + \cos\varphi\mathbf{e}_y + 0\mathbf{e}_z \\ \mathbf{x3} &= \cos\varphi\sin\theta\mathbf{e}_x + \sin\varphi\sin\theta\mathbf{e}_y + \cos\theta\mathbf{e}_z \end{aligned} \tag{2}$$

You can also use the relationship between the spatial coordinates of the spherical coordinate system, (r, θ, φ), and the spatial coordinates of the rectangular coordinate system, (x, y, z):

$$\begin{aligned} \varphi &= \operatorname{atan}\frac{y}{x} \\ \theta &= \operatorname{acos}\frac{z}{\sqrt{x^2+y^2+z^2}} \end{aligned} \tag{3}$$

Combining Equation 2 and Equation 3 gives a user-defined spherical coordinate system, whose unit vectors **x1**, **x2**, and **x3** can be represented in terms of the spatial coordinates (x, y, z). Note that this user-defined coordinate system is a cyclic permutation of the default spherical coordinate system with the property that the radial direction is aligned with the **x3** direction. This coordinate system helps you to implement a radial poling direction for the piezo material.

In COMSOL Multiphysics, solid mechanics problems are solved using the Lagrangian formulation, and hence one needs to make a distinction between the Spatial Coordinate System denoted by the coordinate variables (x, y and z) and the Material Coordinate System denoted by the coordinate variables (X, Y and Z). For this reason, when setting up the user-defined spherical coordinate system, use upper case variables (X, Y and Z). For more information on this topic, refer to the solid mechanics theory section in the *Acoustics Module User's Guide*.

Results and Discussion

Figure 2 shows the structural displacement of the piezoelectric material. The displacement profile corresponds to that of the "breathing mode" of the hollow spherical structure. Figure 3 shows the voltage distribution in the piezoelectric material. Figure 4 shows the acoustic pressure variation in the core air domain as well as the adjoining PML region. The color bands denote pressure waves that are propagating radially away from the transducer.

Note that as a result of having a PML on the outer layer, the pressure monotonically drops to zero within this layer.

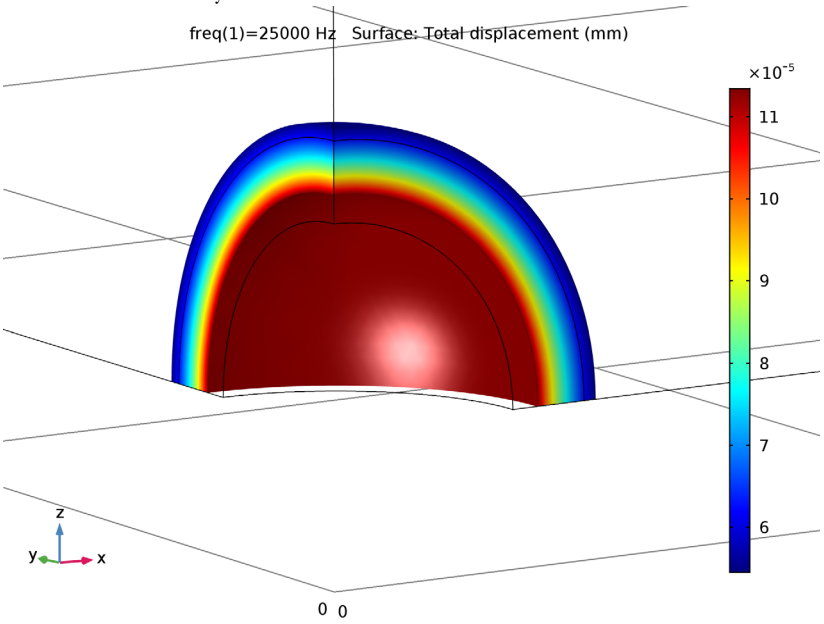


Figure 2: Surface plot of scaled deformation of the piezoelectric material when excited with 100 V at 25 kHz.

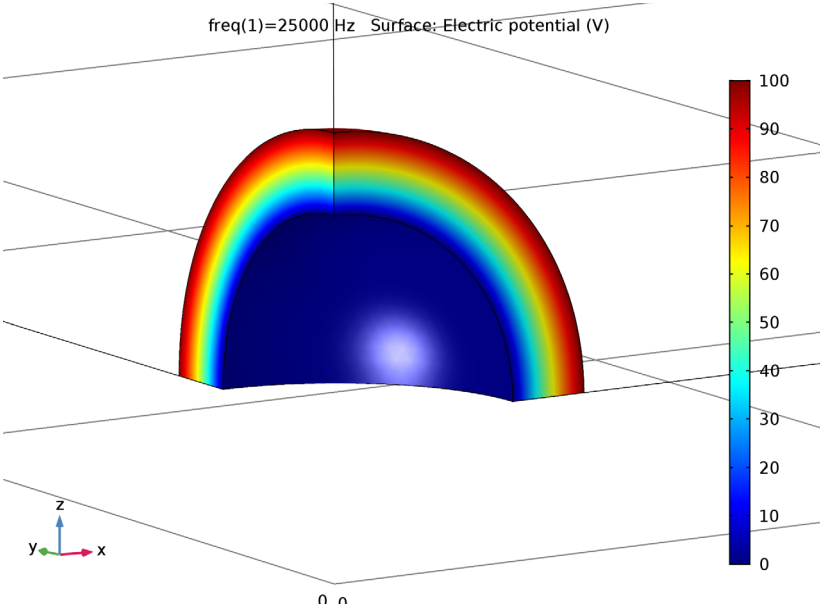


Figure 3: The electric potential, V, in the piezoelectric material.

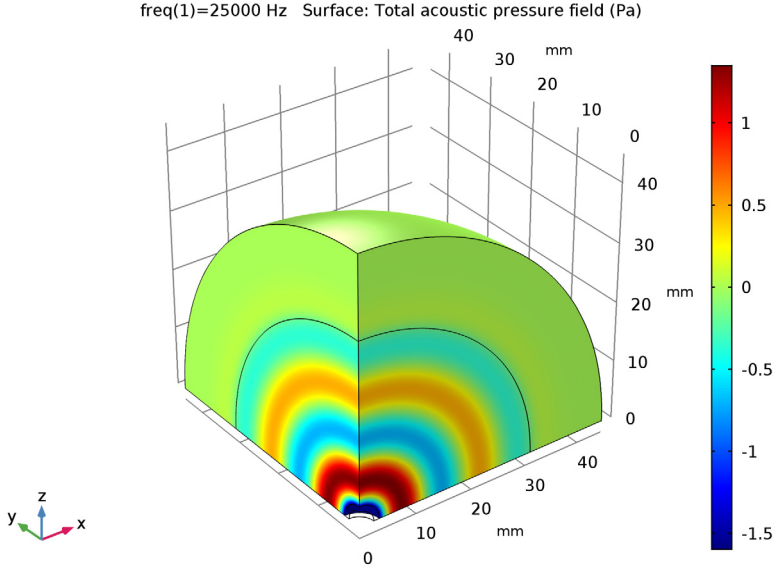


Figure 4: The acoustic pressure in air surrounding the piezoelectric material.

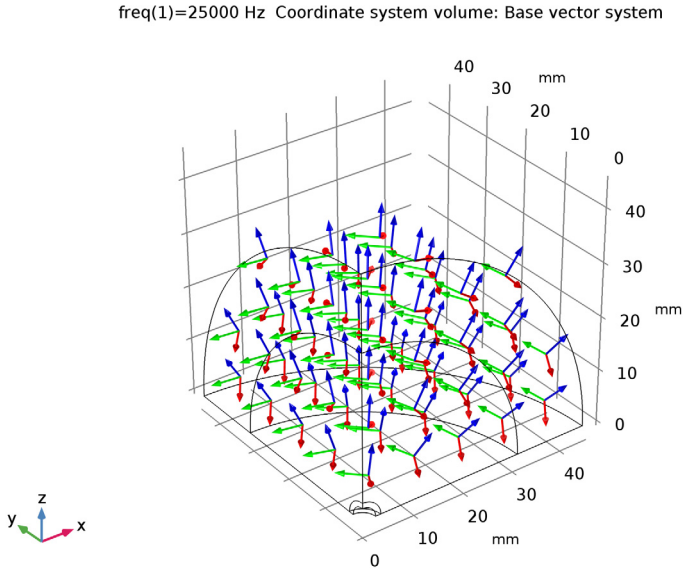


Figure 5: Arrow plot depicting the spherical coordinate system used to set up the poling direction in PZT-5H.

Figure 5 shows an arrow plot of the user-defined spherical coordinate system used to set up the poling direction in the piezoelectric material. Although this coordinate system is only active in the piezo domain, for visual clarity the arrows are plotted in the entire modeling geometry. The red, green and blue arrows correspond to the **x1**, **x2**, and **x3** directions, respectively, in the user-defined local coordinate system. In this case, **x1**, **x2**, and **x3** correspond to the polar, azimuthal, and radial directions, respectively. Because the **x3** direction corresponds to the radial direction, as indicated by the blue arrow, this setting helps to implement the idea that the piezo material is radially poled.

Figure 6 shows a plot of the acoustic pressure variation along the radius of the spherical air region including the PML. The pressure reaches a maximum at the piezo-air interface. The magnitude of the sinusoidally varying pressure decays with increasing distance from the piezo domain. Within the PML region, the pressure monotonically decays to zero, thereby demonstrating perfect absorption. Note that the pressure is continuous at the air-PML interface.

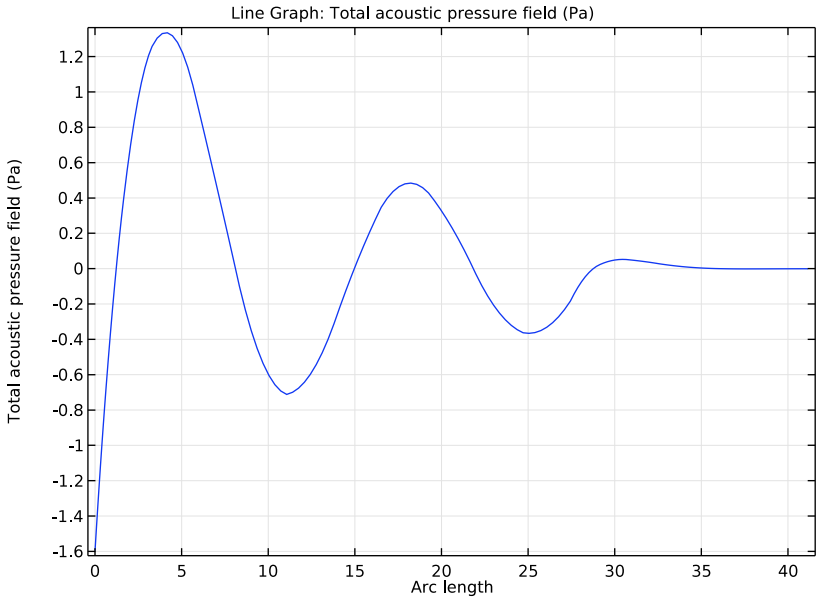


Figure 6: Pressure versus distance along the radius of the spherical air region.

Application Library path: `Acoustics_Module/Piezoelectric_Devices/piezoacoustic_spherical`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Piezoelectric Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
f0	25[kHz]	25000 Hz	Excitation frequency
c_fluid	343[m/s]	343 m/s	Speed of sound in fluid
lambda0	c_fluid/f0	0.01372 m	Wavelength
t_piezo	1[mm]	0.001 m	Thickness of piezo layer
r_piezo_inner	2.5[mm]	0.0025 m	Inner radius of piezo
r_tot	3*lambda0+ r_piezo_inner+ t_piezo	0.04466 m	Total radius of geometry
r_PML	lambda0	0.01372 m	Width of PML layer

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)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

4 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r_tot`.

4 In the **Sector angle** text field, type `90`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`r_PML`
`Layer 2`	`r_tot-r_PML-r_piezo_inner-t_piezo`
`Layer 3`	`t_piezo`

6 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Plane Geometry

On the **Work Plane** toolbar, click **Delete**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1> Work Plane 1 (wp1)>Plane Geometry** click **Delete Entities 1 (del1)**.

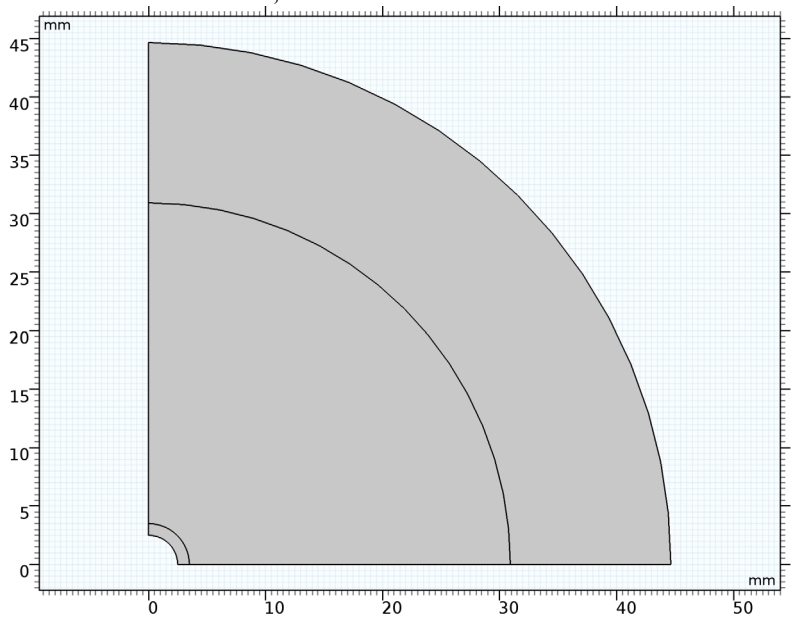
2 Select the object **c1** only.

3 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 On the object **c1**, select Domain 1 only.

6 On the **Work Plane** toolbar, click **Build All**.



7 In the **Model Builder** window, click **Geometry 1**.

Revolve 1 (rev1)

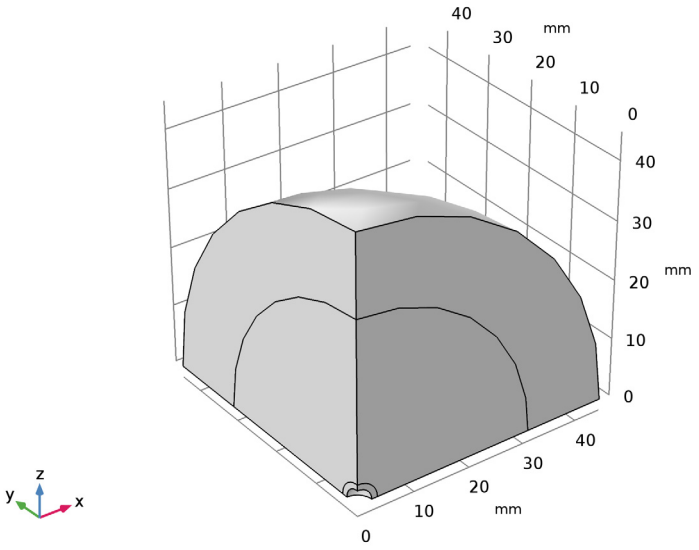
1 On the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `-90`.

5 Click **Build All Objects**.



ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-5H)**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Lead Zirconate Titanate (PZT-5H) (mat2)

Select Domain 1 only.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 3 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Spherical**.

Now, add a system of coordinates that represents the spherical system defined in Equation 2.

Base Vector System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Base Vector System**.

2 In the **Settings** window for **Base Vector System**, locate the **Settings** section.

3 Find the **Base vectors** subsection. In the table, enter the following settings:

	x	y	z
x1	`cos(atan2(Y,X))*cos(acos(Z/sqrt(X^2+Y^2+Z^2)))`	`sin(atan2(Y,X))*cos(acos(Z/sqrt(X^2+Y^2+Z^2)))`	`-sin(acos(Z/sqrt(X^2+Y^2+Z^2)))`
x2	`-sin(atan2(Y,X))`	`cos(atan2(Y,X))`	`0`
x3	`cos(atan2(Y,X))*sin(acos(Z/sqrt(X^2+Y^2+Z^2)))`	`sin(atan2(Y,X))*sin(acos(Z/sqrt(X^2+Y^2+Z^2)))`	`cos(acos(Z/sqrt(X^2+Y^2+Z^2)))`

4 Find the **Simplifications** subsection. Select the **Assume orthonormal** check box.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 Select Domains 2 and 3 only.

3 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Typical Wave Speed for Perfectly Matched Layers** section.

4 In the c_{ref} text field, type `acpr.c`.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 4, 5, and 12 only.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domain 1 only.

Piezoelectric Material 1

Select the new base vector system you have defined `sys2` as the local system of coordinates.

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 **Coordinate System Selection** section.

3 From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

4 In the **Model Builder** window, click **Solid Mechanics (solid)**.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 2, and 11 only.

The symmetry planes take care of the fact that you are modeling only 1/8th of the actual structure. Note that the symmetry boundary conditions that you just added in acoustics and solid mechanics only takes care of symmetry here. The electrical symmetry boundary condition is the same as the defaults Zero Charge. Since this boundary condition is applied on these boundaries by default, you do not need to do anything more explicitly.

ELECTROSTATICS (ES)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electrostatics (es)**.

2 Select Domain 1 only.

3 In the **Model Builder** window, click **Electrostatics (es)**.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundary 3 only.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 Select Boundary 6 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

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

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 `lambda0/5`.

5 In the **Minimum element size** text field, type `t_piezo/2`.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1 and 2 only.

Add a size node that defines the size inside of the piezo shell. Define a maximum size equal to half the thickness of the shell. The structural dynamics and electrostatics need to be resolved.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 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 `t_piezo/2`.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `8`.

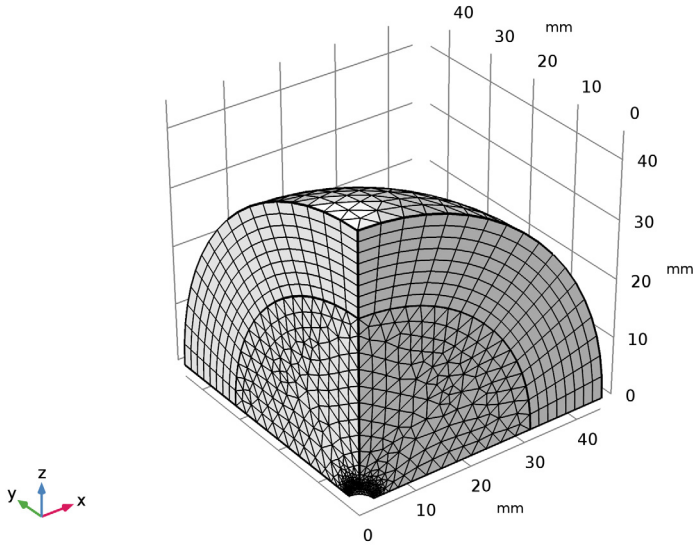
5 In the **Model Builder** window, click **Mesh 1**.

6 In the **Settings** window for **Mesh**, click **Build All**.

7 Click the **Go to Default View** button on the **Graphics** toolbar.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

The meshed geometry should look as shown in the figure below.



STUDY 1

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `f0`.

3 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

The plot should look like Figure 4.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

Acoustic Pressure, Isosurfaces (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure, Isosurfaces (acpr)**.

2 On the **Acoustic Pressure, Isosurfaces (acpr)** toolbar, click **Plot**.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Potential (es)** node.

2 Right-click **Multislice 1** and choose **Disable**.

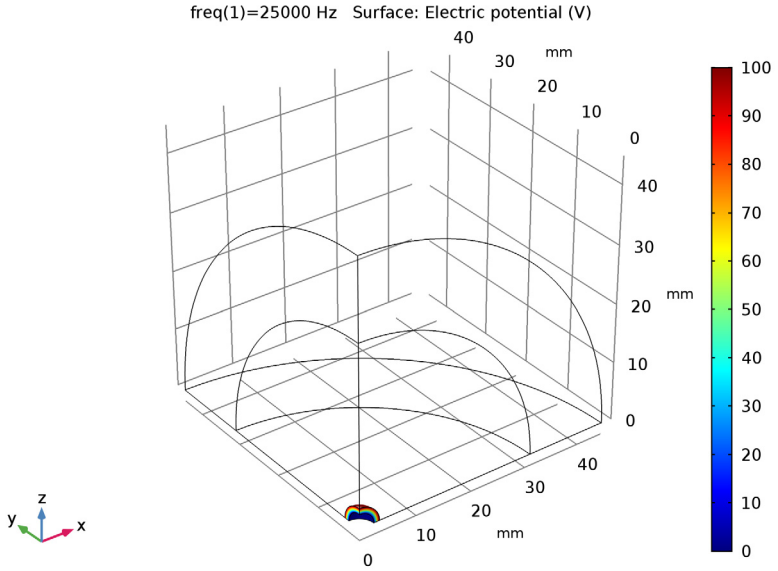
Surface 1

1 In the **Model Builder** window, under **Results** 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 **Model>Component 1>Electrostatics> Electric>V - Electric potential**.

3 On the **Electric Potential (es)** toolbar, click **Plot**.

The plot should look like the figure below. Use the **Zoom Box** tool to zoom on the figure to study it in more detail, this can look like Figure 3.



Save the view settings to apply them to the plots that contain the piezo domain only (for example, zoomed in on the piezo sphere).

DEFINITIONS

View 3

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

2 In the **Settings** window for **View**, type `Zoom View` in the **Label** text field.

3 Locate the **View** section. Select the **Lock camera** check box.

RESULTS

Electric Potential (es)

1 In the **Model Builder** window, under **Results** click **Electric Potential (es)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **Zoom View**.

4 On the **Electric Potential (es)** toolbar, click **Plot**.

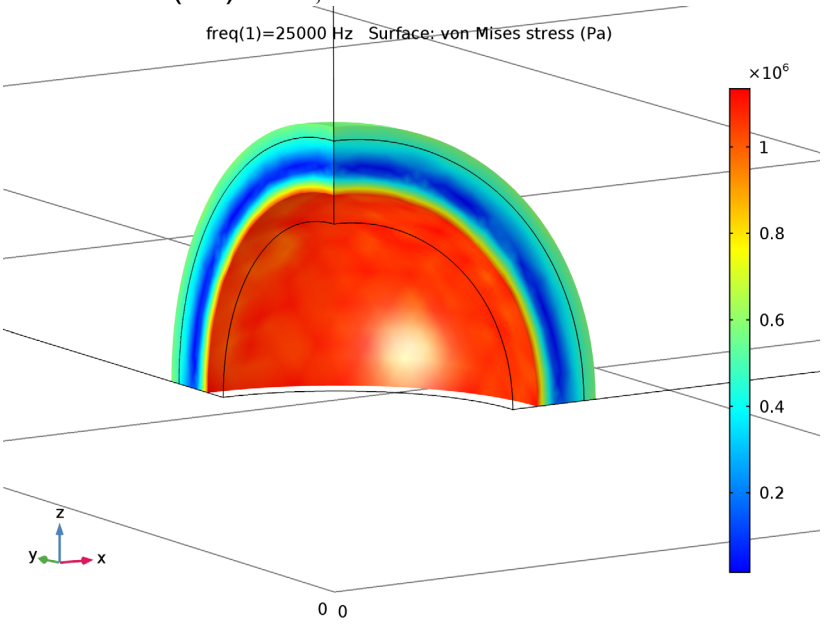
Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **Zoom View**.

4 On the **Stress (solid)** toolbar, click **Plot**.



3D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **Zoom View**.

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 **Model>Component 1>Solid Mechanics> Displacement>solid.disp - Total displacement**.

3 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

4 On the **Displacement** toolbar, click **Plot**.

The plot should look like Figure 2.

3D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Coordinate system` in the **Label** text field.

Coordinate System Volume 1

1 On the **Coordinate system** toolbar, click **More Plots** and choose **Coordinate System Volume**.

2 In the **Settings** window for **Coordinate System Volume**, locate the **Coordinate System** section.

3 From the **Coordinate system** list, choose **Base Vector System 2 (sys2)**.

4 Locate the **Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `5`.

5 Find the **y grid points** subsection. In the **Points** text field, type `5`.

6 Find the **z grid points** subsection. In the **Points** text field, type `5`.

7 On the **Coordinate system** toolbar, click **Plot**.

Go back to the default view.

8 Click the **Go to Default View** button on the **Graphics** toolbar.

The plot should look like Figure 5.

1D Plot Group 8

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `1D Pressure Plot` in the **Label** text field.

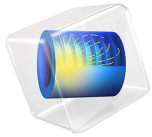
Line Graph 1

1 Right-click **1D Pressure Plot** and choose **Line Graph**.

2 Select Edges 20 and 21 only.

3 On the **1D Pressure Plot** toolbar, click **Plot**.

The plot should look like Figure 6.



Piezoacoustic Transducer

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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.

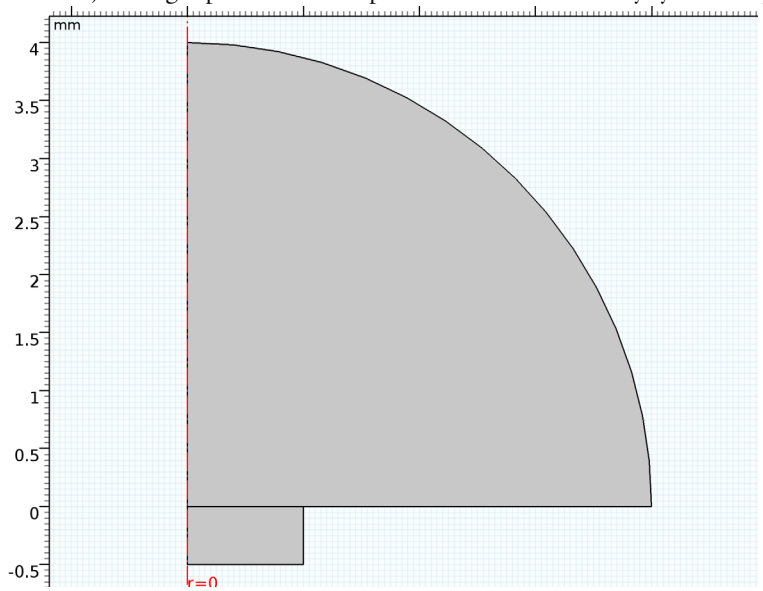


Figure 1: The model geometry.

PHYSICS IMPLEMENTATION IN DOMAINS

The model uses the built-in Acoustic-Piezoelectric Interaction, Frequency Domain multiphysics interface which comprises 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 that is a common material in piezoelectric transducers. The piezoelectric material is modeled by solving the Solid Mechanics and Electrostatics interfaces that are coupled via the 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 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 outwards 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.

Figure 2 shows the pressure distribution in the air domain.

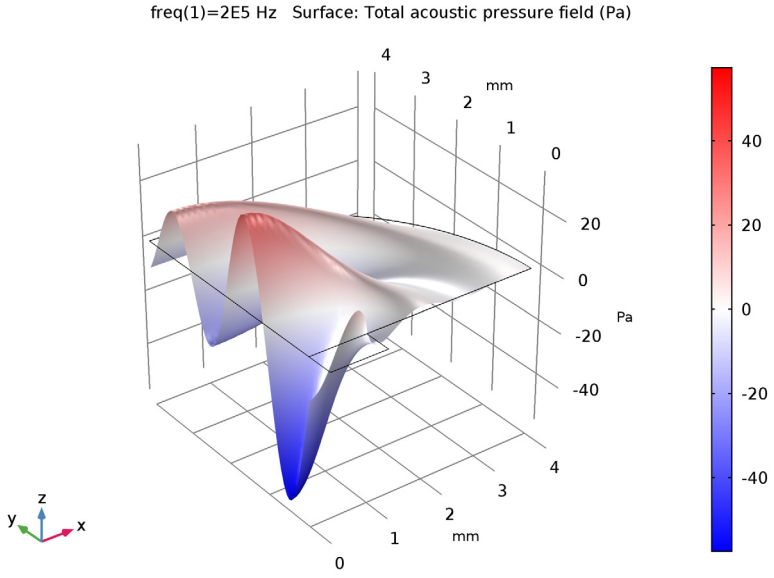


Figure 2: Surface and height plot of the pressure distribution.

Figure 3 shows the pressure distribution along the air-solid interface. Note that the acoustic pressure is small in comparison to the mechanical stress, which is plotted in Figure 4.

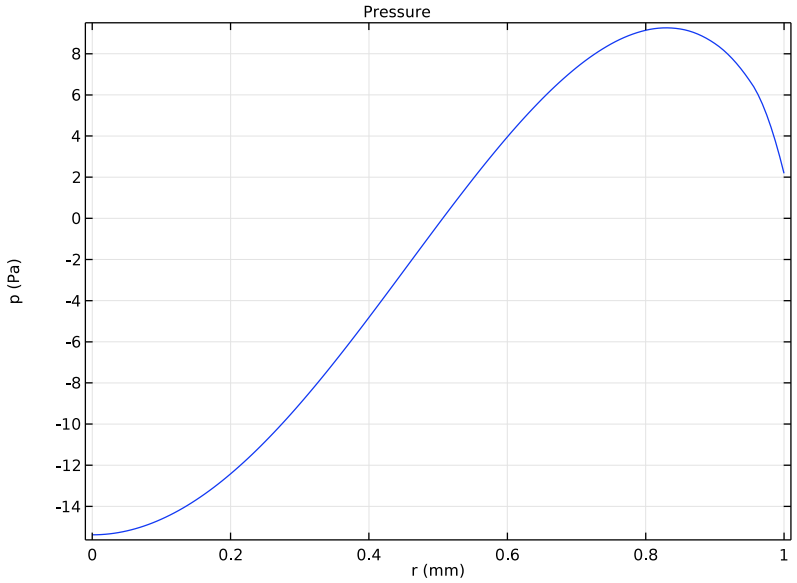


Figure 3: Acoustic pressure at the air-solid interface.

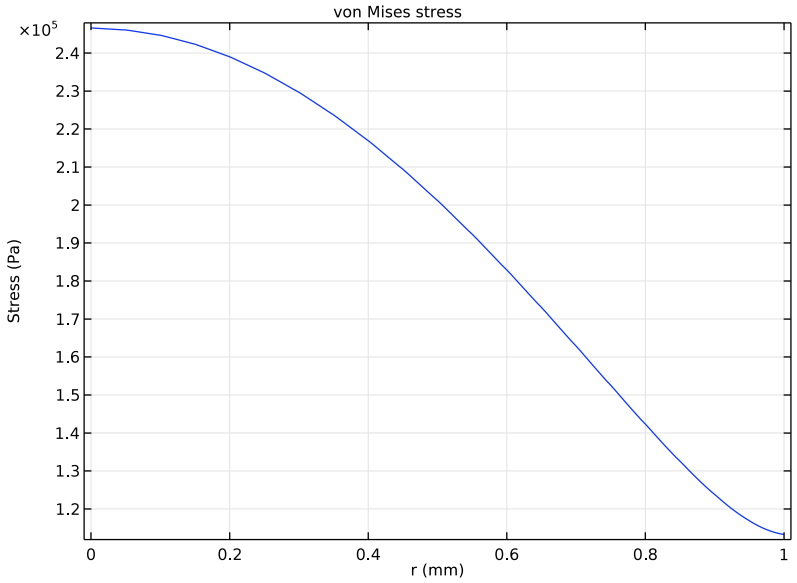


Figure 4: von Mises Stress along the air-solid interface.

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.

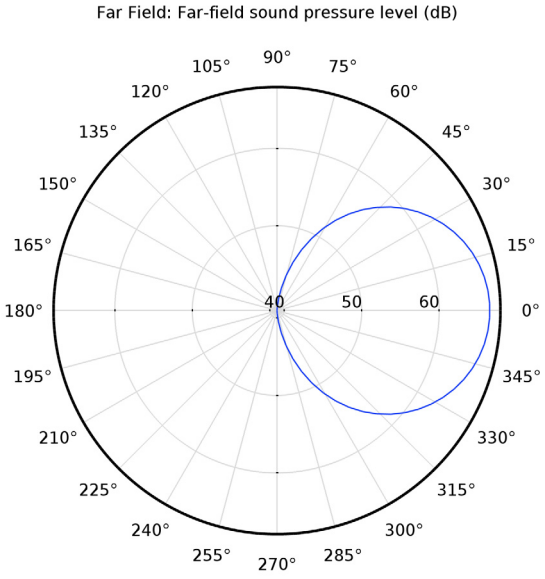


Figure 5: A polar plot of the far-field sound pressure level. The 0 degree axis coincides with the +z-direction of the rz-plane in the 2D axisymmetric model.

Application Library path: `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 **Preset Studies for Selected Physics Interfaces> 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 On the **Geometry** toolbar, click **Primitives** and choose **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 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Next, add the transducer, which is just a rectangle.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **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 on 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)

On the **Physics** toolbar, click **Pressure Acoustics, Frequency Domain (acpr)** and choose **Solid Mechanics (solid)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domain 1 only.

ELECTROSTATICS (ES)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **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

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 On 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 On 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.

Spherical Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **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 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Far-Field Calculation**, 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 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 On 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 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundary 2 only.

Electric Potential 1

1 On 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.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

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, under **Component 1 (comp1)>Mesh 1** 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 **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `200[kHz]`.

3 On 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.

Surface 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **WaveLight**.

4 Select the **Symmetrize color range** check box.

5 Right-click **Results>Acoustic Pressure (acpr)>Surface 1** and choose **Height Expression**.

6 Click the **Zoom Extents** button on 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 on the **Graphics** toolbar.

The third and fourth plots are the 3D revolved plots of acoustic pressure and sound pressure level.

Far-Field Sound Pressure Level (acpr)

The fifth default plot shows the far-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.

Far Field 1

1 In the **Model Builder** window, expand the **Far-Field Sound Pressure Level (acpr)** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, 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 On the **Far-Field Sound Pressure Level (acpr)** toolbar, click **Plot**.

Far-Field Pressure (acpr)

Next, do the same with the six default plot, which is the far-field pressure.

Far Field 1

1. In the **Model Builder** window, expand the **Far-Field Pressure (acpr)** node, then click **Far Field 1**.
2. In the **Settings** window for **Far Field**, 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. On the **Far-Field Pressure (acpr)** toolbar, click **Plot**.

Add a plot of the total displacement in the piezoelectric transducer.

2D Plot Group 11

1. On 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.

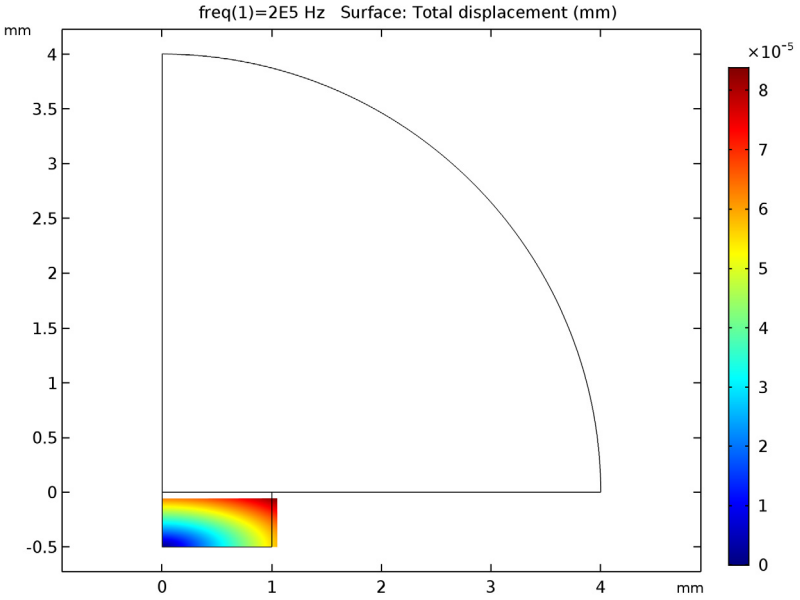
Surface 1

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 **Model>Component 1>Solid Mechanics> Displacement>solid.disp - Total displacement**.

Deformation 1

1. Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

2 On the **Displacement** toolbar, click **Plot**.



Next, create 1D plot groups to recreate Figure 3 and Figure 4.

1D Plot Group 12

1 On 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.

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 **Model>Component 1>Solid Mechanics> Stress>solid.mises - von Mises stress**.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `r`.

von Mises Stress

1 In the **Model Builder** window, under **Results** click **von Mises Stress**.

2 In the **Settings** window for **1D Plot Group**, click to expand 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 text field, type `r (mm)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Stress (Pa)`.

9 On the **von Mises Stress** toolbar, click **Plot**.

1D Plot Group 13

1 On 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.

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`.

Pressure

1 In the **Model Builder** window, under **Results** click **Pressure**.

2 In the **Settings** window for **1D Plot Group**, locate the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Pressure`.

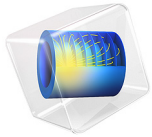
5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `r (mm)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `p (Pa)`.

9 On the **Pressure** toolbar, click **Plot**.



Porous Absorber

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of acoustic absorption by a porous acoustic open cell foam. In porous materials the sound propagates in a network of small interconnected pores. Because the dimensions of the pores are small, losses occur due to thermal conduction and viscous friction. Acoustic foams are used to sound proof rooms and ducts as well as to treat reverberation problems in rooms (see Ref. 1).

The aim of the model is to characterize the absorption properties—more specifically, the specific surface impedance and the absorption coefficient—of a layer of melamine foam in terms of sound incidence angle and frequency. The melamine foam contains an air inclusion. An analytical solution exist in the case where the layer is uniform. The model uses a 2D geometry of such a system.

Model Definition

Figure 1 depicts the geometry of the modeled system, in which an incident sound field hits the porous melamine foam layer at angle θ. An air inclusion, circular domain of radius a, is present in the porous layer. The incident wave has wave vector $\mathbf{k}$. In the figure, the dotted lined indicates the boundaries of the model domain. You only model a portion of width W and apply periodic Floquet boundary conditions on the left and right boundaries to extend the domain to infinity. The incident field is modeled by applying a background pressure field to the air domain. At the top, a perfectly matched layer (PML) domain is used to model an infinitely large air domain. The thickness of the porous melamine layer is $H_p = 10$ cm and the height of the modeled air region is $H = 30$ cm. The height of the PML domain is H_{pml}.

Model the melamine foam using the Pressure Acoustics interface's Poroacoustics domain feature using the Johnson-Champoux-Allard model with a rigid frame. This is a an equivalent fluid model for a rigid frame porous material, a so-called five parameter semi-empirical equivalent fluid model. See About the Poroacoustics Models in the *Acoustics Module User's Guide*. The surrounding fluid is air, and the material parameters for the foam are as listed in Table 1 (following Ref. 2, material sample number 31).

TABLE 1: MELAMINE FOAM MATERIAL PARAMETERS

SYMBOL	VALUE	DESCRIPTION
ε_p	0.995	Porosity
R_f	10,500 Pa·s/m^2	Flow resistivity
s	0.49	Viscous characteristic length parameter
L_{th}	470 μm	Thermal characteristic length

TABLE 1: MELAMINE FOAM MATERIAL PARAMETERS

SYMBOL	VALUE	DESCRIPTION
L_v	240 μm	Viscous characteristic length
τ	1.0059	Tortuosity factor

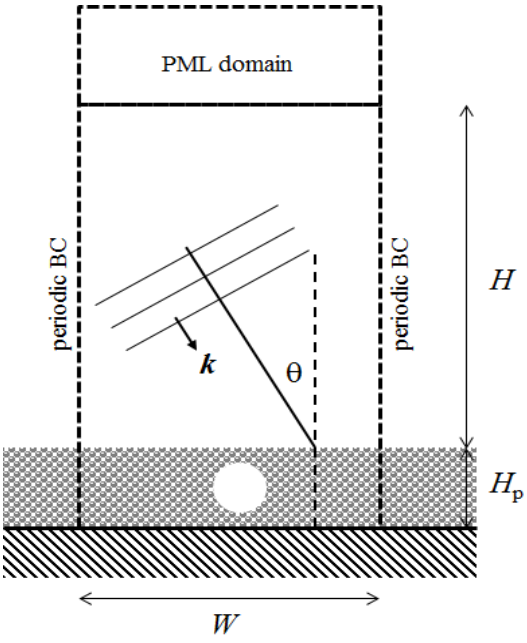


Figure 1: Geometry of the modeled system, the air inclusion has a radius a.

The incident background pressure field is given as

$$p_{\text{inc}} = e^{-i(\mathbf{k}\cdot\mathbf{x})} \qquad \mathbf{k} = k_0(\sin\theta,-\cos\theta) \tag{1}$$

where θ is the incidence angle and k_0 is the wave number in the free field (air domain). The pressure p solved for in the model is the total field and the scattered field p_{scat} is given as $p_{\text{scat}} = p - p_{\text{inc}}$. Note that this expression for the scattered field is only valid in the air domain, as the incident field is not know a priori in the porous material.

Two parameters that characterize the absorption properties of the porous absorber are the specific surface impedance Z and the absorption coefficient α (see Ref. 1). The absorption coefficient, which represents the ratio of the absorbed and incident energy, is defined as

$$\alpha = 1 - |R|^2 \qquad R = \frac{p_{\text{scat}}}{p_{\text{inc}}} \tag{2}$$

where R is the pressure reflection coefficient that gives the ratio of the scattered to the incident pressure. The surface specific impedance (normalized by the plane wave characteristic impedance) is defined as

$$Z = \frac{1}{\rho c}\frac{p}{u_{\text{n}}} \tag{3}$$

where ρ is the density of air, c is the speed of sound, and $u_{\text{n}} = \mathbf{u}\cdot\mathbf{n}$ is the normal velocity at the surface of the melamine layer. Both coefficients are dependent on frequency and on the incidence angle.

UNIFORM POROUS LAYER SOLUTION

In the case of a uniform porous layer (with no air inclusions) of thickness H_{p} backed by a sound hard wall an analytical solution exists for the surface impedance, reflection coefficient, and absorption coefficient (see Ref. 1). The surface normal impedance (normalized by the characteristic plane wave impedance) is given by

$$\begin{aligned} Z_{\text{ana}} &= \frac{1}{\rho c}\frac{-iZ_{\text{c}}k_{\text{c}}}{k_{\text{x}}\cot(k_{\text{x}}H_{\text{p}})} \\ k_{\text{x}} &= \sqrt{k_{\text{c}}^2 - k_{\text{y}}^2} \\ k_{\text{y}} &= k_0\sin(\theta) \end{aligned} \tag{4}$$

where a subscript "c" represents complex valued impedance and wave number variables from the Poroacoustic domain. From the normal impedance the absorption coefficient is deduced.

Figure 2 and Figure 3 plot the scattered and total fields for an incidence angle of 45° and the frequency f = 10 kHz. Note how the wave is absorbed in the porous layer.

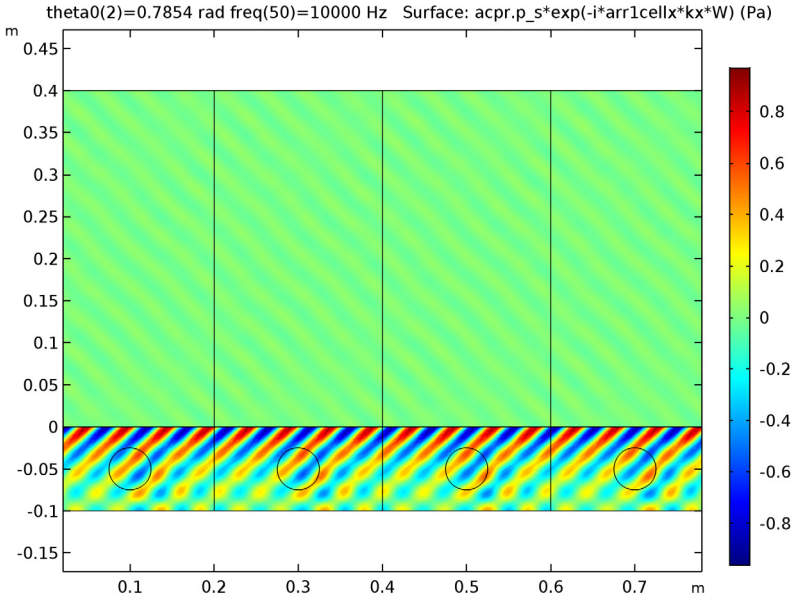


Figure 2: Scattered field for an incidence angle of 45^0 and frequency f = 10 kHz.

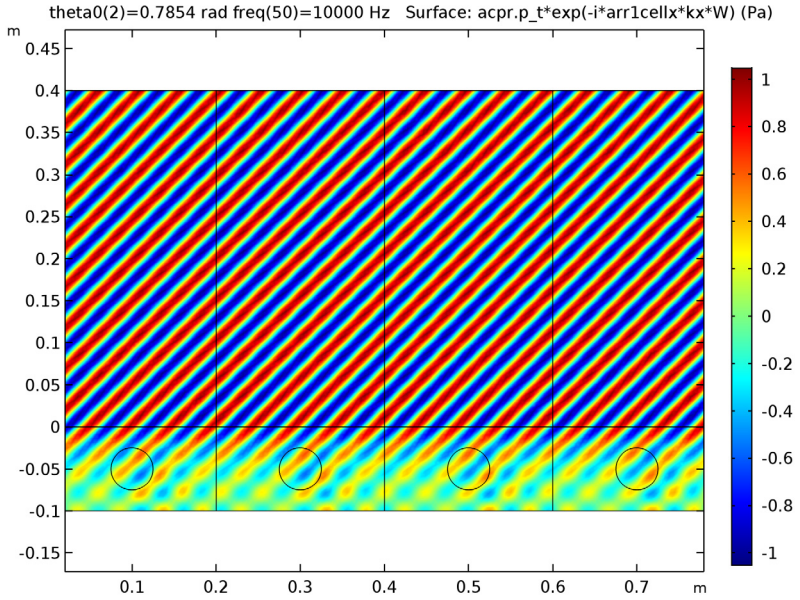


Figure 3: Total acoustic pressure for an incidence angle of 45^0 and frequency f = 10 kHz.

Figure 4 depicts the total sound pressure level at the surface of the porous melamine layer. Figure 5 plots the specific acoustic impedance at the surface of the porous absorber, and Figure 6 shows the absorption coefficient. The latter are compared to the analytical solution of a uniform porous layer.

The dependency of the surface specific impedance on incidence angle and frequency is important for modeling absorbers as impedance boundary conditions in, for example, a Ray Acoustics model. In larger model systems the present model could be used as a "submodel" to determine appropriate impedance boundary conditions. The real part of the impedance (the resistance) is associated with energy loss whereas the imaginary part (the reactance) is associated with phase changes of the field. The reciprocal value of the impedance is the admittance.

In this system, the absorption coefficient approaches 1 for increasing frequency. This corresponds to the frequency where the product between the porous absorber height H_p and $k_y \pi^{-1}$ of the incident wave is equal to one. This is where one half wavelength fits into the absorbing layer.

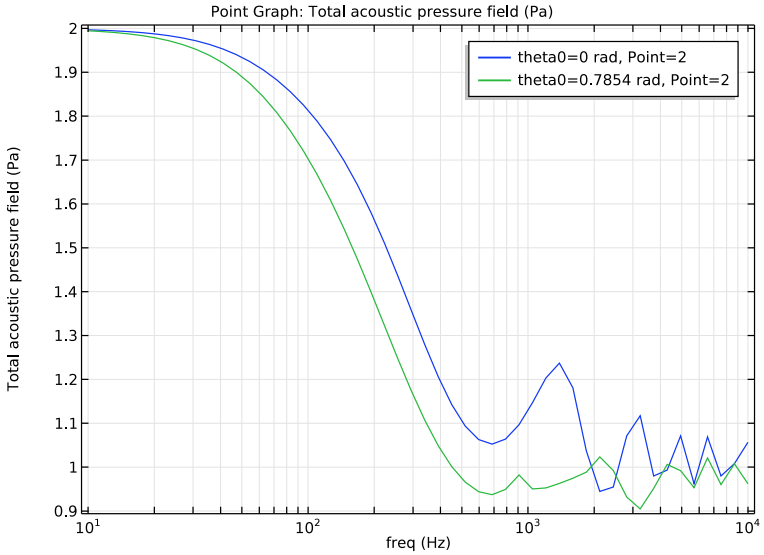


Figure 4: Sound pressure level at the surface of the porous absorber.

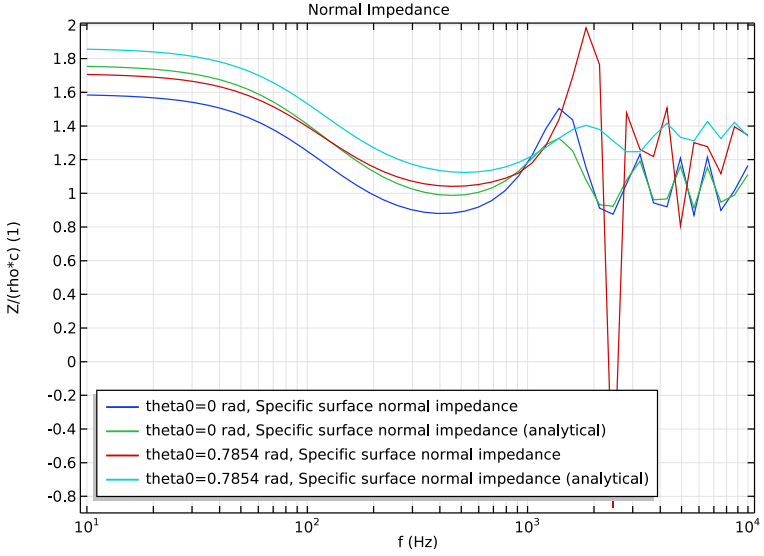


Figure 5: The specific acoustic impedance at the surface of the porous absorber.

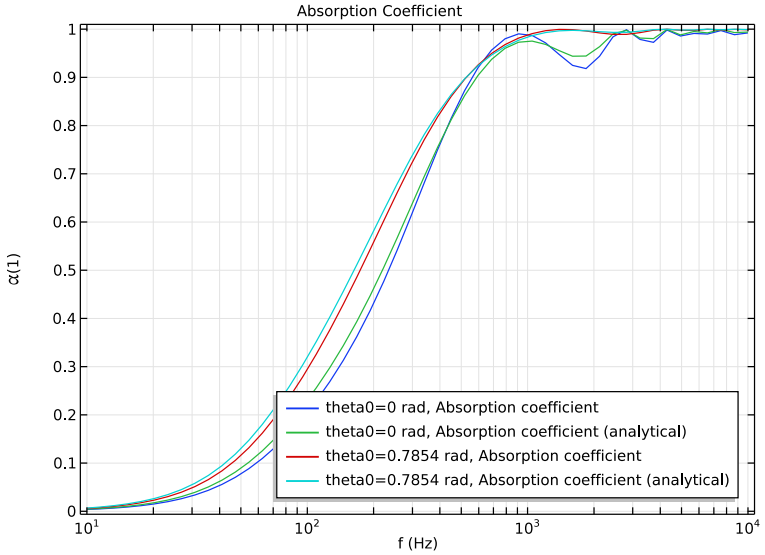


Figure 6: Absorption coefficient for the porous melamine absorber as function of frequency and incidence angle. Compared to the analytical solution of a uniform layer.

Notes About the COMSOL Implementation

PERIODIC FLOQUET BOUNDARY CONDITION

Apply a periodic Floquet boundary condition to model an infinite periodic structure. The periodicity is determined by the wave number of the background (incident) pressure field. The relation between the pressure at the left and right boundaries of the model domain is

$$p(\mathbf{x}) = p(\mathbf{x} + \mathbf{d})e^{-i(\mathbf{k} \cdot \mathbf{d})} \tag{5}$$

where $\mathbf{d} = (W, 0)$ is a vector extending from the left to the right boundary and $\mathbf{k}$ is the wave vector defined in Equation 1. COMSOL automatically calculates the vector $\mathbf{d}$ when applying the Floquet periodicity.

VISUALIZE PERIODIC SOLUTION

To visualize the periodic solution create an Array 2D data set and enable the special variables created by the dataset. The cell variable `arr1cellx` gives the cell number in the x-direction. Use this variable to plot, for example, the total field

```
acpr.p_t*exp(-i*arr1cellx,kx*W)
```

where the periodicity is included by multiplying with `exp(-i*arr1cellx*kx*W)`.

COMPARING TO THE ANALYTICAL SOLUTION

In the results section the simulated absorption coefficient and surface impedance are compared to the analytical solution of a uniform porous layer. To make a verification of the model simply select the inclusion (the circular air domain) as a Poroacoustic domain and run the model again. You will find that the analytical and model results show perfect agreement.

References

1. T.J. Cox and P. D'Antonio, *Acoustic Absorbers and Diffusers, Theory, Design and Applications,* 2nd ed., Taylor and Francis, 2009.

2. N. Kino and T. Ueno, "Comparison between characteristic lengths and fiber equivalent diameter in glass fiber and melamine foam materials of similar flow resistivity," *J. App. Acoustics*, vol. 69, pp. 325–331, 2008.

Application Library path: `Acoustics_Module/Building_and_Room_Acoustics/porous_absorber`

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**.
2. In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Load the parameters for the model. The list of parameters include geometry definitions, definitions used in the mesh, and material parameters for the melamine foam.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `porous_absorber_parameters.txt`.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `W`.

4 In the **Height** text field, type `H+Hpml`.

5 Click to expand the **Layers** section. Clear the **Layers on bottom** check box.

6 Select the **Layers on top** check box.

7 In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Hpml`

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `W`.

4 In the **Height** text field, type `Hp`.

5 Locate the **Position** section. In the **y** text field, type `-Hp`.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `a`.

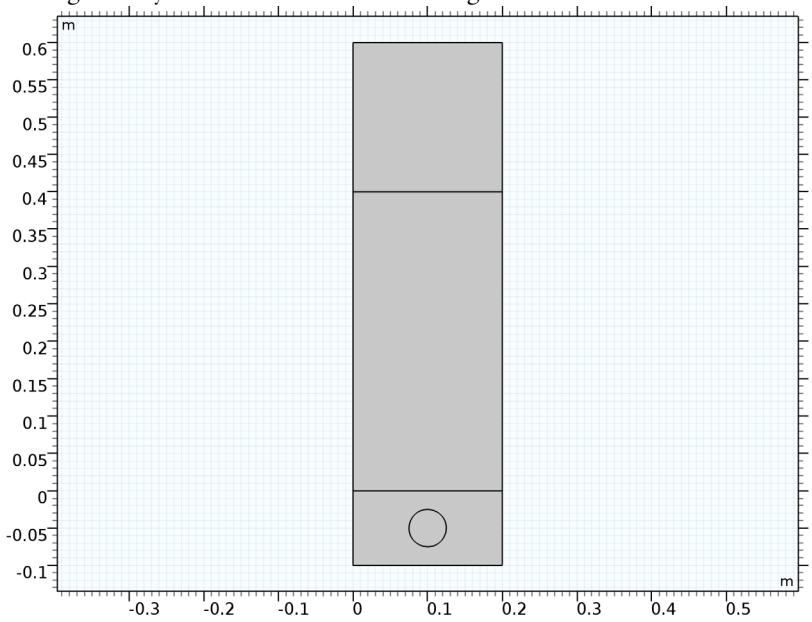
4 Locate the **Position** section. In the **x** text field, type `W/2`.

5 In the **y** text field, type `-Hp/2`.

6 On the **Geometry** toolbar, click **Build All**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

The geometry should look like that in the figure below.



DEFINITIONS

Load the expressions defining the background pressure field, see Equation 1, as well as the surface impedance and absorption coefficient, see Equation 2 and Equation 3, from a file.

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `porous_absorber_variables.txt`.

Load the expressions defining the analytical expressions for a single porous layer with a sound hard backing, see Equation 4.

Variables 2

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `porous_absorber_analytical.txt`.

Define two integration coupling operators that act on points in the geometry. You will use them later to map (or probe) values from these points. One in the porous domain and one in the air domain.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 1 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 3 only.

Define an average operator on the porous-air interface.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 4 only.

Now proceed to set up the material properties. Add air as the default domain material and create a new material to define the melamine foam porosity.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material 2 (mat2)

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 `Melamine Foam` in the **Label** text field.

3 Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Porosity**.

4 Click **Add to Material**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Porosity	epsilon	`epsilonP0`	1	Basic

6 In the **Settings** window for **Material**, click to expand the **Material properties** section.

7 Locate the **Material Properties** section. In the **Material properties** tree, select **Acoustics> Poroacoustics Model>Thermal characteristic length (Lth)**.

8 Click **Add to Material**.

9 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Flow resistivity	Rf	`Rf0`	Pa·s/m²	Poroacoustics model
Thermal characteristic length	Lth	`Lth0`	m	Poroacoustics model
Viscous characteristic length	Lv	`Lv0`	m	Poroacoustics model
Tortuosity factor	tau	`tau0`	1	Poroacoustics model

Notice that the parameter for the tortuosity is called tau0, not to be confused with the material property of the static viscous tortuosity.

Now set up the physics and the boundary conditions. First, define the incident background pressure field, see Equation 2, then the Floquet condition, see Equation 5, and finally porous material properties for the melamine foam.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 3 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Background Pressure Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Background Pressure Field**.

2 Select Domain 2 only.

3 In the **Settings** window for **Background Pressure Field**, locate the **Background Pressure Field** section.

4 From the **Pressure field type** list, choose **User defined**.

5 In the p_b text field, type `p_inc`.

Now, add the poroacoustic domain defining the melamine foam.

Poroacoustics 1

1 On the **Physics** toolbar, click **Domains** and choose **Poroacoustics**.

2 Select Domain 1 only.

3 In the **Settings** window for **Poroacoustics**, locate the **Poroacoustics Model** section.

4 From the **Poroacoustics model** list, choose **Johnson-Champoux-Allard**.

5 Locate the **Porous Matrix Properties** section. From the **Porous elastic material** list, choose **Melamine Foam (mat2)**.

It is good practice to add a periodic condition for each type of domain, in this case, one for the PML, one for the pressure acoustics, and one for the porous domain. This is especially the case when using a background pressure field.

Periodic Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 1 and 8 only.

3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 Specify the $\mathbf{k}_F$ vector as

kx	x
ky	y

Periodic Condition 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 3 and 9 only.

3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 Specify the $\mathbf{k}_F$ vector as

kx	x
ky	y

Periodic Condition 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 5 and 10 only.

3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 Specify the $\mathbf{k}_F$ vector as

kx	x
ky	y

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1, 2, and 4 only.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

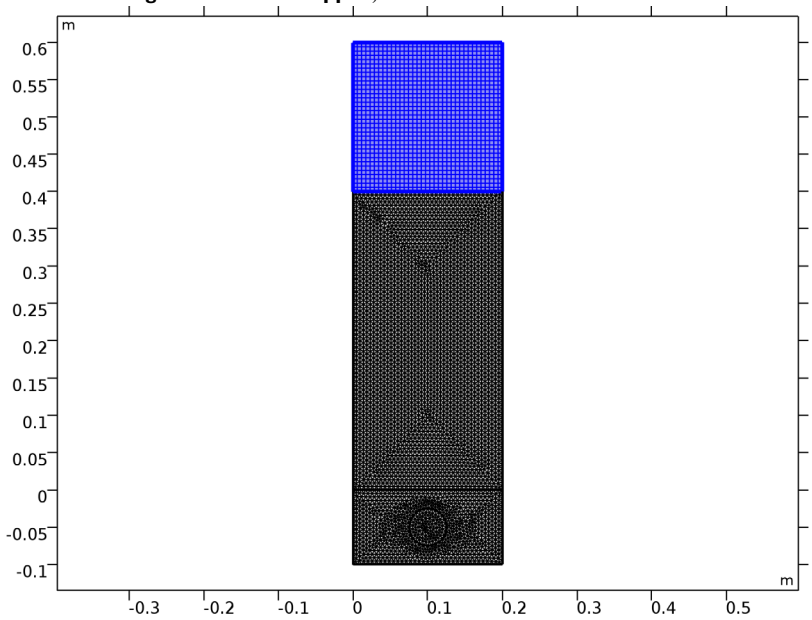
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambda_min/6`.

This mesh resolves the smallest wavelength of the study `lambda_min` with 6 elements.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.



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 **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `10^{range(1,3/49,4)}`.

You thus solve the model on a logarithmic frequency axis from 10 Hz to 10^4 Hz. Add a parametric sweep over the incidence angle `theta0` for the values 0^o and 45^o.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list
theta0 (Incident wave angle)	`0[deg] 45[deg]`

5 On the **Study** toolbar, click **Compute**.

Create an array data set that will help you plot the Floquet periodic solution on several unit cells. Add a selection to not show the unphysical solution in the PML domain.

RESULTS

Study 1/Parametric Solutions 1 (sol2)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/ Parametric Solutions 1 (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1, 2, and 4 only.

Array 2D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Array 2D**.

2 In the **Settings** window for **Array 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Array Size** section. In the **X size** text field, type `4`.

5 Click to expand the **Advanced** section. Select the **Define variables** check box.

Acoustic Pressure (acpr)

Modify the first default plot to depict the total pressure in the periodic geometry. For this purpose, use the special variable `arr1cellx` created by the dataset.

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **2D Plot Group**, type `Total Acoustic Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Array 2D 1**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Total Acoustic Pressure** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_t*exp(-i*arr1cellx*kx*W)`.

4 On the **Total Acoustic Pressure** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the resulting plot with that in Figure 3.

Now change the incidence angle from 45^o to 0^o.

Total Acoustic Pressure

1 In the **Model Builder** window, under **Results** click **Total Acoustic Pressure**.

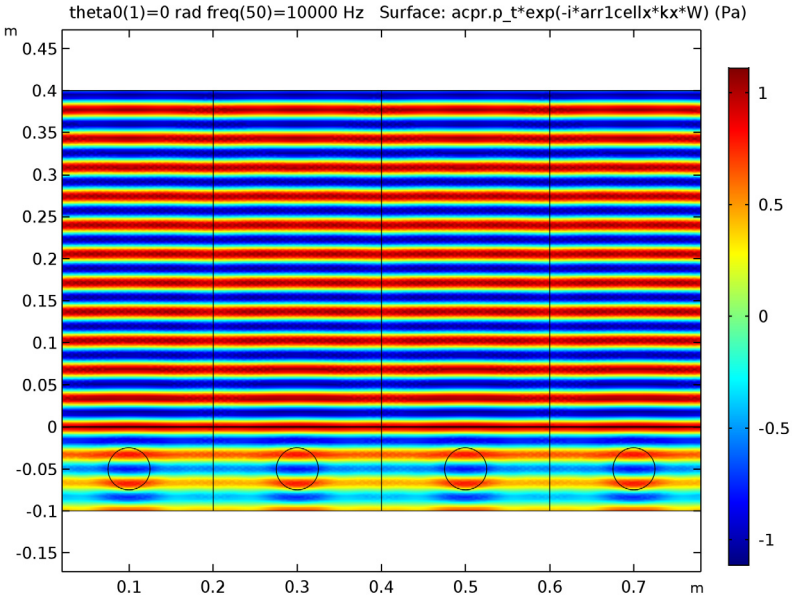
2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (theta0 (rad))** list, choose **0**.

4 On the **Total Acoustic Pressure** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The result should look like that in the following figure.



Total Acoustic Pressure 1

1 Right-click **Results>Total Acoustic Pressure** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Scattered Acoustic Pressure` in the **Label** text field.

Scattered Acoustic Pressure

Now, plot the scattered acoustic pressure.

Surface 1

1 In the **Model Builder** window, expand the **Results>Scattered Acoustic Pressure** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_s*exp(-i*arr1cellx*kx*W)`.

Scattered Acoustic Pressure

1 In the **Model Builder** window, under **Results** click **Scattered Acoustic Pressure**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (theta0 (rad))** list, choose **0.7854**.

4 On the **Scattered Acoustic Pressure** toolbar, click **Plot**.

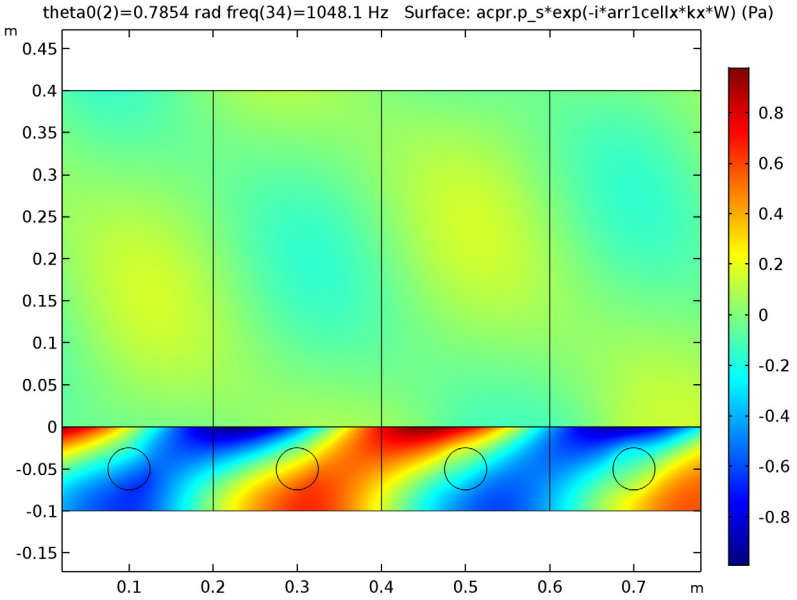
Compare the resulting plot with that in Figure 2.

Now change the frequency from 10 kHz to 1 kHz.

5 From the **Parameter value (freq (Hz))** list, choose **1048.1**.

6 On the **Scattered Acoustic Pressure** toolbar, click **Plot**.

The result should look like that in the figure below.



Total Acoustic Pressure

Next, plot the incident acoustic pressure field at 10 kHz for an incidence angle of 0^o

Total Acoustic Pressure 1

1 In the **Model Builder** window, under **Results** right-click **Total Acoustic Pressure** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Incident Acoustic Pressure` in the **Label** text field.

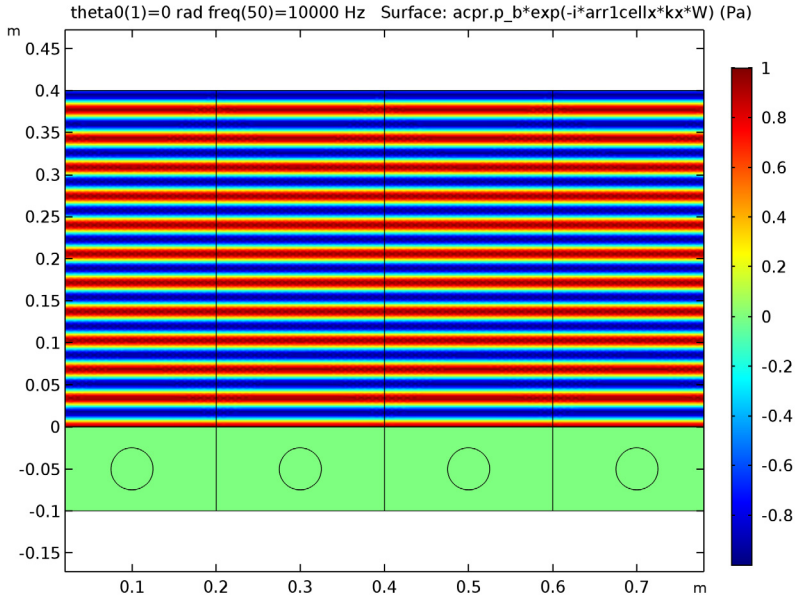
Surface 1

1 In the **Model Builder** window, expand the **Results>Incident Acoustic Pressure** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_b*exp(-i*arr1cellx*kx*W)`.

4 On the **Incident Acoustic Pressure** toolbar, click **Plot**.



Next, create 1D plots to depict the absorption properties of the melamine absorber.

First, reproduce the plot in Figure 4, which shows the sound pressure level at the surface of the porous melamine layer.

1D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Surface Sound Pressure Level` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

Point Graph 1

1 Right-click **Surface Sound Pressure Level** and choose **Point Graph**.

2 Select Point 2 only.

3 In the **Settings** window for **Point Graph**, click to expand the **Legends** section.

4 Select the **Show legends** check box.

5 On the **Surface Sound Pressure Level** toolbar, click **Plot**.

6 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Proceed by plotting the acoustic normal impedance at the surface of the porous melamine layer. The plot should look like that in Figure 5.

1D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Normal Impedance` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Normal Impedance`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `f (Hz)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Z/(rho*c) (1)`.

10 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 Right-click **Normal Impedance** 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
`Z`	1	`Specific surface normal impedance`
`Z_ana`	1	`Specific surface normal impedance (analytical)`

4 Click to expand the **Legends** section. On the **Normal Impedance** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.

Finally, plot the absorption coefficient of the porous melamine layer for the two studied incidence angles (Figure 6).

1D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Absorption Coefficient` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Absorption Coefficient`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `f (Hz)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `\alpha (1)`.

10 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 Right-click **Absorption Coefficient** and choose **Global**.

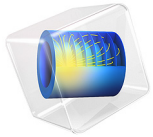
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`alpha`	1	`Absorption coefficient`
`alpha_ana`	1	`Absorption coefficient (analytical)`

4 Click to expand the **Legends** section. On the **Absorption Coefficient** toolbar, click **Plot**.

5 Click the **x-Axis Log Scale** button on the **Graphics** toolbar.



Probe Tube Microphone

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

It is often not possible to insert a normal microphone directly into the sound field being measured. The microphone may be too big to fit inside the measured system, such as for in-the-ear measurements for hearing aid fitting. The size of the microphone may also be too large compared to the wavelength, so that it disturbs the acoustic field. In these cases, a probe tube may be attached to the microphone case (see the sketch in Figure 1) in order to distance the microphone from the measurement point. In this model, the effect on the microphone's sensitivity due to the addition of this small probe is investigated, see Ref. 1 (Chapter 4, pp. 160–162) for further details.

This is a time-dependent model of a generic probe tube microphone setup consisting of an external acoustic domain, an elastic probe tube, and the cavity in front of the microphone diaphragm; see the sketch in Figure 1. The probe tube, modeled using the Pipe Acoustics, Transient interface, is connected to two separate 3D pressure acoustics domains, leading to a fully coupled acoustics simulation.

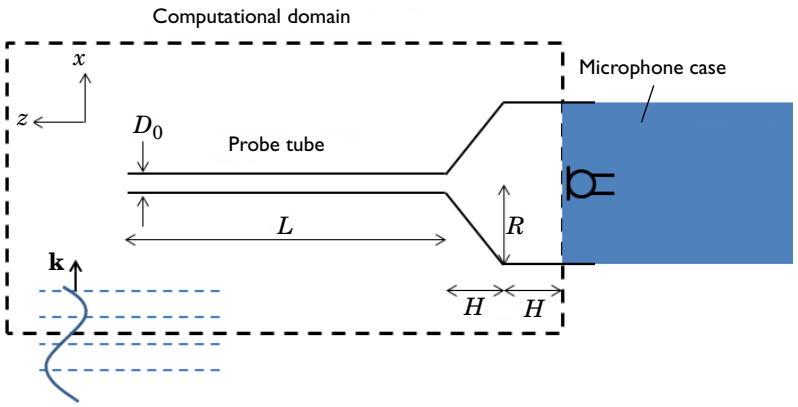


Figure 1: Sketch of the probe tube microphone setup, consisting of a probe tube of length L and outer diameter D_0. The tube is connected to a cavity (with the indicated dimensions) in front of the microphone diaphragm. An external sound field hits the probe tube with wave vector **k**.

Note: This application requires the Acoustics Module and the Pipe Flow Module.

Model Definition

The probe tube microphone geometry and the computational domain considered in this model are shown schematically in Figure 1. From right to left, the model consists of the cavity in front of the microphone, which takes the form of a cylinder of radius R and height H connected to a cone of bottom radius R and top radius D_0. D_0 is also the outer diameter of the probe tube of length L. The probe is made of an elastic material with inner diameter $2d_r = 2$ mm and wall thickness $d_w = 0.3$ mm. The tube material is set to have a Young's modulus of 0.1 GPa and a Poisson's ratio of 0.4.

Table 1 lists the model parameters,

TABLE 1: MODEL PARAMETERS

VARIABLE	VALUE	DESCRIPTION
f	500 Hz	Incident wave frequency
T	$1/f$	Incident wave period
ω	$2\pi f$	Incident wave angular frequency
c_0	343 m/s	Speed of sound at 20 °C and 1 atm
λ_0	c_0/f	Wave length of incident wave
k	$2\pi/\lambda_0$	Wave number of incident wave
L	20 mm	Probe tube length
R	5 mm	Microphone casing radius
H	5 mm	Cavity height
d_r	1 mm	Pipe inner radius
d_w	0.3 mm	Pipe wall thickness
D_0	$2d_r + 2d_w = 2.6$ mm	Probe tube outer diameter
kD_0	$k \cdot D_0 = 0.02$	Wave number times tube diameter
d_{visc}	0.1 mm	Viscous boundary layer thickness at 500 Hz

A sinusoidal wave moving in the positive x direction is incident on the probe tube microphone. The wave with an amplitude of 1 Pa is given by

$$p_{in} = 1\,\text{Pa} \cdot \sin(\omega t - kx) \tag{1}$$

where ω ισ the angular frequency, and k is the wave number (these are given in Table 1). Such a plane monochromatic wave exists as a built in option for the **Incident Pressure Field** feature in the interface.

Modeled using the Pipe Acoustics, Transient interface, the long probe tube is treated as 1D structure. This assumption is valid as long as the interaction between the probe tube and the incoming sound field can be neglected. This, in turn, requires that $k \cdot D_0 << 1$, which is true for the current choice of parameters because $k \cdot D_0 = 0.02$; see Table 1.

Another important assumption is that there are no significant thermal and viscous boundary losses inside the probe tube. Also this assumption holds for the current setup, where the incident field is a monochromatic wave. At a driving frequency of 500 Hz the viscous boundary layer thickness inside the probe tube is of the order 0.1 mm, which is 10 times smaller than the internal radius d_r. The thickness of the boundary layer is proportional to $1/\sqrt{f}$.

Because the diaphragm is not a fully rigid structure, assume it to have a resistive loss given by the impedance $Z = 100 \times 10^6$ N·s/m^5. This value is of the order of magnitude observed in common condenser microphones.

Results and Discussion

An important parameter of a probe tube extension to a microphone is the relation between the pressure at the tip and the pressure at the diaphragm (as measured by the microphone). This transfer function is necessary to calibrate the measurement system. The pressure at the tip of the probe and the pressure at the diaphragm are shown in Figure 2. After an initial transient, the solution becomes periodic after about 4 ms. Hereafter, the system has a gain of about 1.4 and a phase shift equal to $\phi = \Delta t \cdot \omega = 0.1 \text{ ms} \cdot 2\pi f = 9°$. The gain and phase shift depend on the frequency content of the applied signal — in this case a pure harmonic tone of 500 Hz.

A full transfer function could be obtained, for example, by using a Gaussian pulse as the incident signal and then performing a Fourier transform of the signal. This requires resolving the frequency content with an appropriate mesh that would become very dense. Moreover, the model assumptions might not hold for the full range of frequencies.

Figure 3 depicts the pressure distribution in the *xz*-plane at the end of the time interval shown in Figure 2.

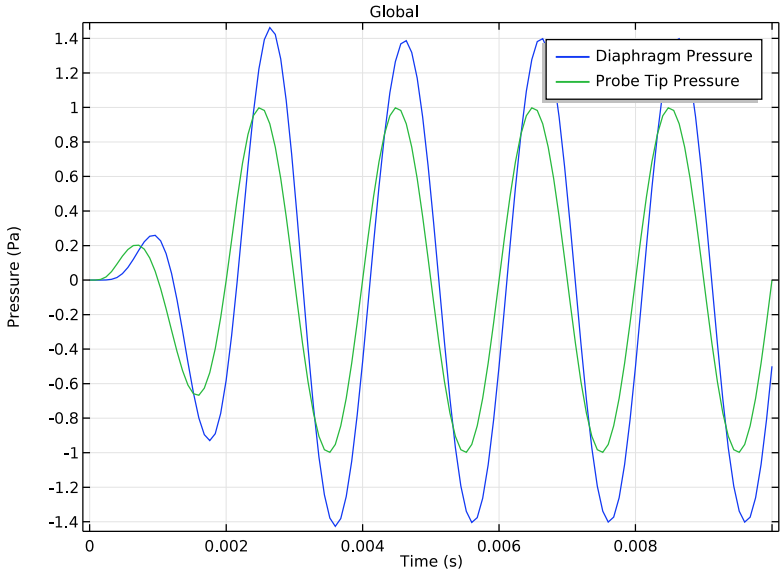


Figure 2: Pressure as function of time at the probes tip (green line) and at the microphone diaphragm (blue line).

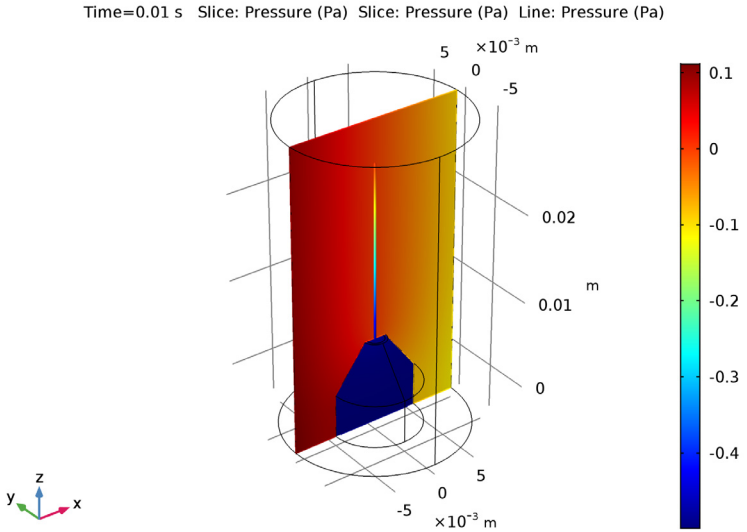


Figure 3: Pressure distribution in the xz-plane at time t = 10 ms.

COUPLING BETWEEN 1D AND 3D DOMAINS

To couple the 1D pipe acoustic domain with the two 3D pressure acoustic domains requires some manual steps. First of all, note that the dependent variables are p and u in the pipe domain, p_2 in the external air domain, and p_3 in the microphone cap.

At the tip of the probe tube, set the pressure inside the tube equal to the pressure at the outer domain

$$p = p_2 \tag{2}$$

COMSOL Multiphysics treats this type of Dirichlet boundary condition as a bidirectional constraint. This means that the pressure fields are coupled in both directions, so that the constraint only needs to be stated on one of the physics. To see this, choose **Show> Advanced Physics Options** and then expand the **Constraint Settings** panel. The default is **All physics (symmetric)**.

Continuity also needs to be imposed at the interface between the probe tube and the cone that tops the volume covering the microphone. This amounts to setting the velocity at the tube tip equal to the mean value on the connecting boundary as well as setting the normal acceleration on the boundary equal to the time derivative of the tangential velocity in the tube. This condition is best formulated using a so-called Lagrange multiplier λ_p, this is an extra variable solved for that enforces the boundary condition. This amounts to solving an extra weak equation and setting a boundary condition:

$$\begin{aligned} v_{\text{in}} &= \pm\frac{1}{A}\int \lambda_p dA \qquad \text{on pipe tip} \\ \frac{\partial \lambda_p}{\partial t}\tilde{p_3} + (p_3 - p)\tilde{\lambda_p} &\qquad \text{on boundary} \end{aligned} \tag{3}$$

Where a tilde above a variable represents its test function. In COMSOL this amounts to adding a weak contribution on the boundary (turn on view **Advanced Physics Options**) with the value: `d(lm_p,t)*test(p3)*actd2.delta+(p3-intop(p))*test(lm_p)`

In pipe acoustics set up a velocity condition with: `patd.exit*aveop(lm_p[m/s])`

Note: Further examples of coupling between 1D pipes and 3D domains is found in the tutorial model *Acoustics of a Pipe System with 3D Bend and Junction*, located in the Application Gallery: www.comsol.com/model/acoustics-of-a-pipe-system-with-3d-bend-and-junction-40831.

SOLVER SETUP

The only adjustments needed to be done to the solver is to change it from using a segregated approach to using a fully coupled approach. The default logic in the study is to use segregated groups for physics that do not have a built in multiphysics coupling. Here the coupling is set up manually. The settings for the transient solver are handled automatically by setting the **Maximum frequency to resolve** in the **Transient Solver Settings** section (as done in the instructions below.)

Note: This application requires the Acoustics Module and the Pipe Flow Module.

Reference

1. D.T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/probe_tube_microphone`

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 **Acoustics>Acoustic-Structure Interaction>Pipe Acoustics, Transient (patd)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Transient (actd)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Time Dependent**.

10 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `probe_tube_microphone_parameters.txt`.

GEOMETRY 1

Next, define the geometry of the model. Either build the geometry by following the instructions below (see **Geometry 1**) or import the geometry from a sequence, and then jump to the **Materials** section. To import the geometry sequence, click **Insert Sequence** in the **Geometry** toolbar. Browse to the model's Model Library folder and open the file `probe_tube_microphone_geom_sequence.mph` . The geometry should look like the one in the figure below the geometry instructions.

Use the wire frame rendering to more easily see and access faces and lines inside the geometry.

1 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Cone 1 (cone1)

1 On the **Geometry** toolbar, click **Cone**.

2 In the **Settings** window for **Cone**, locate the **Size and Shape** section.

3 In the **Bottom radius** text field, type `R`.

4 In the **Height** text field, type `H`.

5 In the **Top radius** text field, type `dr+dw`.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `R`.

4 In the **Height** text field, type `H`.

5 Locate the **Position** section. In the **z** text field, type `-H`.

Polygon 1 (pol1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the **x** text field, type `0 0`.

4 In the **y** text field, type `0 0`.

5 In the **z** text field, type `H H+L`.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `10[mm]`.

4 In the **Height** text field, type `35[mm]`.

5 Locate the **Position** section. In the **z** text field, type `-H`.

6 Right-click **Cylinder 2 (cyl2)** and choose **Build Selected**.

Work Plane 1 (wp1)

Now, create a work plane and draw a circle on top of the cone representing the inner part of the probe tube.

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Face parallel**.

4 Find the **Planar face** subsection. Select the **Active** toggle button.

5 On the object **cone1**, select Boundary 4 only.

6 Click **Show Work Plane**.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `dr`.

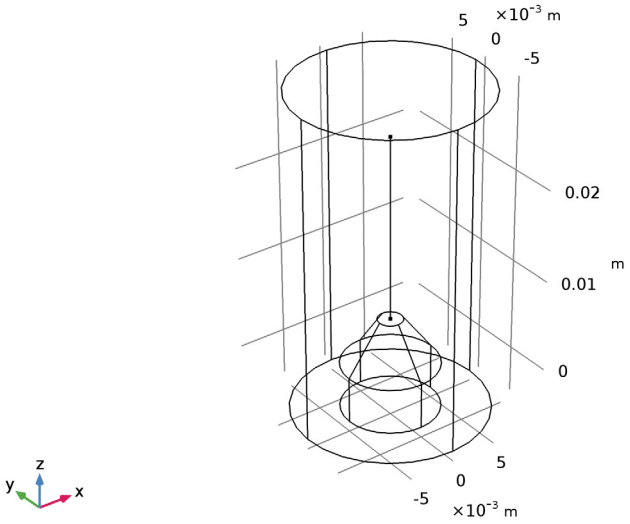
4 On the **Work Plane** toolbar, click **Build All**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

6 In the **Model Builder** window, click **Geometry 1**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

The geometry should look like the figure below.



ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

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 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air 1 (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Edge**.

3 Select Edge 25 only.

DEFINITIONS

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, type `aveop` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 12 only.

Average 2 (aveop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, type `aveop_mic` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 7 only.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

4 Select Point 13 only.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intop_tip` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

4 Select Point 14 only.

PIPE ACOUSTICS, TRANSIENT (PATD)

Proceed to set up the physics of the problem. Select the domains where the different physics are applied and add the appropriate boundary conditions to couple these.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pipe Acoustics, Transient (patd)**.

2 In the **Settings** window for **Pipe Acoustics, Transient**, locate the **Edge Selection** section.

3 Click **Clear Selection**.

4 Select Edge 25 only.

Pipe Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pipe Acoustics, Transient (patd)** click **Pipe Properties 1**.

2 In the **Settings** window for **Pipe Properties**, locate the **Pipe Shape** section.

3 From the list, choose **Circular**.

4 In the d_i text field, type `2*dr`.

5 Locate the **Pipe Model** section. From the **Pipe model** list, choose **Anchored at one end**.

6 From the E list, choose **User defined**.

7 In the associated text field, type `0.1[GPa]`.

8 From the ν list, choose **User defined**.

9 In the associated text field, type `0.4`.

10 From the Δw list, choose **User defined**.

11 In the associated text field, type `dw`.

12 In the **Model Builder** window, click **Pipe Acoustics, Transient (patd)**.

13 In the **Settings** window for **Pipe Acoustics, Transient**, locate the **Transient Solver Settings** section.

14 Find the **Changes made to these settings only take effect when the default solver is generated.** subsection. In the $f_{\text{max, sol}}$ text field, type `f`.

Pressure 1

1 On the **Physics** toolbar, click **Points** and choose **Pressure**.

2 Select Point 14 only.

3 In the **Settings** window for **Pressure**, locate the **Pressure** section.

4 In the p_{in} text field, type `p2`.

This is the boundary condition given in Equation 2.

Velocity 1

1 On the **Physics** toolbar, click **Points** and choose **Velocity**.

2 Select Point 13 only.

3 In the **Settings** window for **Velocity**, locate the **Velocity** section.

4 In the u_{in} text field, type `patd.exit*aveop(lm_p[m/s])`.

This was the first part of the boundary condition given in Equation 3.

PRESSURE ACOUSTICS, TRANSIENT (ACTD)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient (actd)**.

2 In the **Settings** window for **Pressure Acoustics, Transient**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 1 only.

5 Locate the **Transient Solver Settings** section. Find the **Changes made to these settings only take effect when the default solver is generated.** subsection. In the $f_{max,\,sol}$ text field, type `f`.

Cylindrical Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Cylindrical Wave Radiation**.

2 Select Boundaries 1, 2, 13, and 18 only.

3 In the **Settings** window for **Cylindrical Wave Radiation**, locate the **Cylindrical Wave Radiation** section.

4 Specify the $\mathbf{r}_{axis}$ vector as

0	x
0	y
1	z

Add an incident plane wave as defined in Equation 1 using the built in **Plane wave (monochromatic)** option. The wave is automatiocally multiplied with a smoothing ramp

function over the first period. This option can be seen by enabling view **Advanced Physics Options**.

Incident Pressure Field 1

1 On the **Physics** toolbar, click **Attributes** and choose **Incident Pressure Field**.

2 In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Transient (actd)>Cylindrical Wave Radiation 1** click **Incident Pressure Field 1**.

3 In the **Settings** window for **Incident Pressure Field**, locate the **Incident Pressure Field** section.

4 In the p_0 text field, type `1`.

5 From the c list, choose **From material**.

6 From the **Material** list, choose **Air (mat1)**.

7 Specify the $\mathbf{e}_k$ vector as

1	x
0	y
0	z

8 In the f_0 text field, type `f`.

PRESSURE ACOUSTICS, TRANSIENT 2 (ACTD2)

On the **Physics** toolbar, click **Pressure Acoustics, Transient (actd)** and choose **Pressure Acoustics, Transient 2 (actd2)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Transient 2 (actd2)**.

2 In the **Settings** window for **Pressure Acoustics, Transient**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 2 and 3 only.

5 Locate the **Transient Solver Settings** section. Find the **Changes made to these settings only take effect when the default solver is generated.** subsection. In the $f_{\text{max, sol}}$ text field, type `f`.

Impedance 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Impedance**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Impedance**, locate the **Impedance** section.

4 In the Z_i text field, type `100e6[N*s/m^5]*R^2*pi`.

Proceed to set up the remaining pieces of the boundary condition given in Equation 3. I order to do so turn on view **Advanced Physics Options** to be able to add user defined weak contributions to the model.

5 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Weak Contribution 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Weak Contribution**.

2 In the **Settings** window for **Weak Contribution**, type `Weak Contribution: Coupling` in the **Label** text field.

3 Select Boundary 12 only.

4 Locate the **Weak Contribution** section. In the **Weak expression** text field, type `d(lm_p, t)*test(p3)*actd2.delta+(p3-intop(p))*test(lm_p)`.

Auxiliary Dependent Variable 1

1 On the **Physics** toolbar, click **Attributes** and choose **Auxiliary Dependent Variable**.

2 In the **Settings** window for **Auxiliary Dependent Variable**, locate the **Auxiliary Dependent Variable** section.

3 In the **Field variable name** text field, type `lm_p`.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

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 `lambda0/10`.

5 In the **Minimum element size** text field, type `dr/2`.

6 In the **Resolution of narrow regions** text field, type `1`.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 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 **Boundary**.

4 Select Boundary 12 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 `dr/2`.

8 Click **Build Selected**.

Size 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

2 Right-click **Free Tetrahedral 2** and choose **Size**.

3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Edge**.

5 Select Edge 25 only.

6 Locate the **Element Size** section. Click the **Custom** button.

7 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

8 In the associated text field, type `L/10`.

9 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Times** text field, type `range(0,T/25,5*T)`.

The data will be recorded for 5 periods (5*T) with a sampling of 25 points per period (increase the number of points to getter better resolution in the stored solutiond).

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Time-Dependent Solver 1** node.

4 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Time-Dependent Solver 1** and choose **Fully Coupled**.

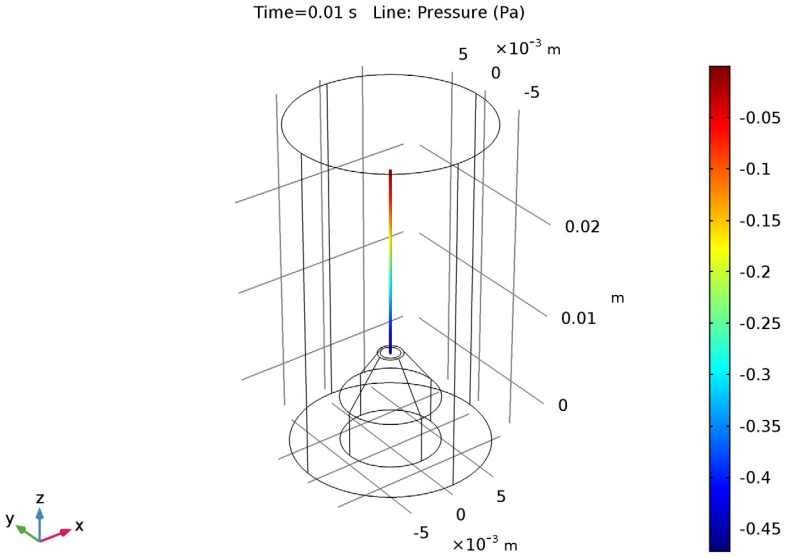
In this model the coupling between physics is set up manually (there is no built-in multiphysics coupling). The default behavior, in the study, is for COMSOL to generate solver suggestions for the individual constituting physics (notice the tags) and sets up a segregated solver suggestion. This model is best solved using the fully coupled approach.

Now, solve the model and proceed to look at the results. First, look at the default plots and then create two custom plot groups. The model only takes a few seconds to solve.

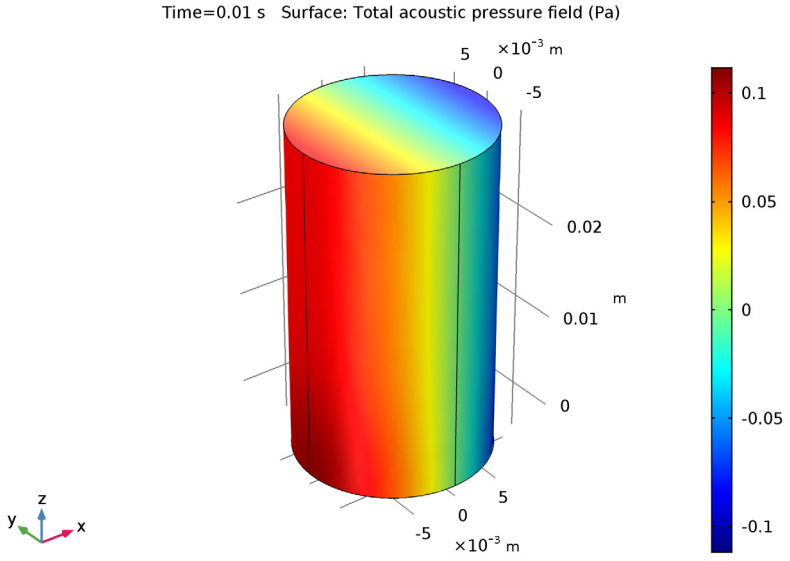
5 In the **Settings** window for **Time-Dependent Solver**, click **Compute**.

RESULTS

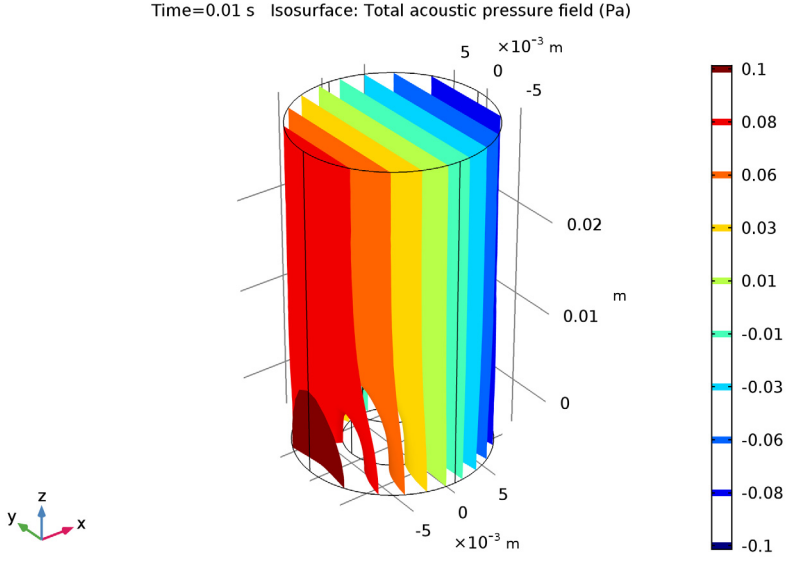
Pressure (patd)



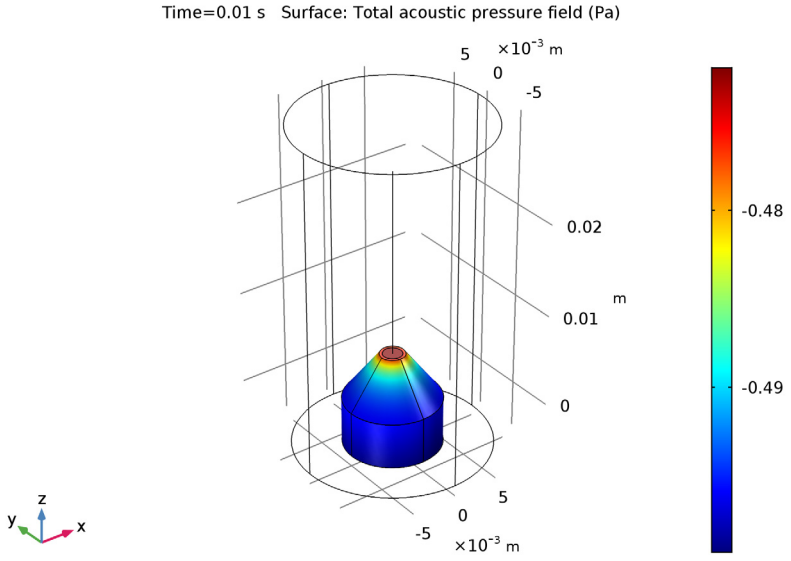
Acoustic Pressure (actd)



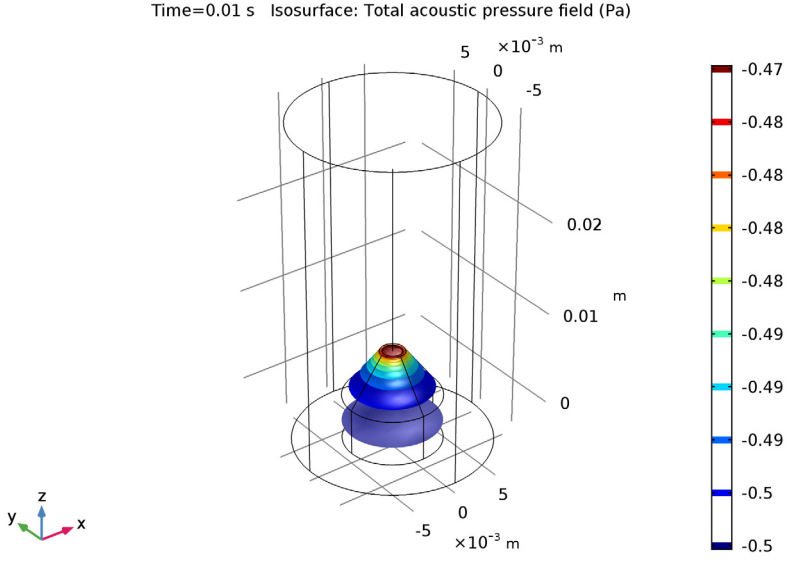
Acoustic Pressure, Isosurfaces (actd)



Acoustic Pressure (actd2)



Acoustic Pressure, Isosurfaces (actd2)



Follow the steps below to reproduce the plot in Figure 3.

Slice 1

1. On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2. In the **Model Builder** window, right-click **3D Plot Group 7** and choose **Slice**.
3. In the **Settings** window for **Slice**, locate the **Expression** section.
4. In the **Expression** text field, type `p2`.
5. Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.
6. In the **Planes** text field, type `1`.

Slice 2

1. In the **Model Builder** window, under **Results** right-click **3D Plot Group 7** and choose **Slice**.
2. In the **Settings** window for **Slice**, locate the **Expression** section.
3. In the **Expression** text field, type `p3`.
4. Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.
5. In the **Planes** text field, type `1`.
6. Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Line 1

1. Right-click **3D Plot Group 7** and choose **Line**.
2. In the **Settings** window for **Line**, locate the **Coloring and Style** section.
3. From the **Line type** list, choose **Tube**.
4. In the **Tube radius expression** text field, type `2*patd.dh`.
5. Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.
6. On the **3D Plot Group 7** toolbar, click **Plot**.

3D Plot Group 7

1. In the **Model Builder** window, under **Results** click **3D Plot Group 7**.
2. In the **Settings** window for **3D Plot Group**, type `Cross Sections` in the **Label** text field.

   The figure should look like the one depicted in Figure 3.

Global 1

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Model Builder** window, right-click **1D Plot Group 8** and choose **Global**.

3 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`aveop_mic(p3)`	`Pa`	`Diaphragm Pressure`
`intop_tip(p)`	`Pa`	`Probe Tip Pressure`

1D Plot Group 8

1 In the **Model Builder** window, under **Results** click **1D Plot Group 8**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** check box.

4 In the associated text field, type `Time (s)`.

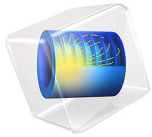
5 Select the **y-axis label** check box.

6 In the associated text field, type `Pressure (Pa)`.

7 On the **1D Plot Group 8** toolbar, click **Plot**.

8 In the **Label** text field, type `Pressure Profiles`.

The figure should look like the one in Figure 2.



Acoustic Reflections Off a Water-Sediment Interface

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This validation model computes the reflection coefficient of acoustic waves off a water-sediment interface. Homogeneous waves are incident from a fluid (water) domain and are reflected and transmitted at a water-sediment interface. The sediment domain is modeled using Biot's theory with the use of the Poroelastic Waves interface. It includes the detailed interaction between the pressure waves in the saturating fluid and the elastic waves in the porous matrix (the sediment). The model results are compared with those obtained by Stoll and Kan (Ref. 1). A visual comparison of the results with those presented in Ref. 1 shows good agreement.

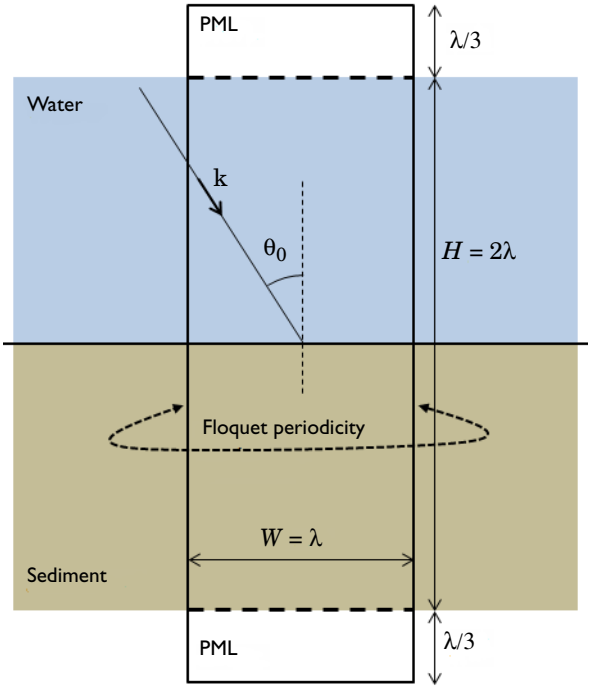


Figure 1: Sketch of the water-sediment system. The computational unit cell has the width W and the height H with perfectly matched layers at the top and bottom. λ is the acoustic wavelength in water. An acoustic wave with wave vector $\mathbf{k}$ is incident at an angle θ_0 on the water-sediment interface.

Knowledge about the plane wave reflection and refraction coefficients are important when interpreting acoustic data taken from marine seismology. The kind of model studied here can provide insight into the different phenomena at play. The sediment is modeled using Biot's theory with the addition of the porous matrix skeleton having viscoelastic properties. This is achieved by simply allowing the bulk modulus and the shear modulus

to have a small imaginary component (a constant attenuation factor). Inside the porous material three types of waves can propagate, a fast and a slowly attenuating pressure wave, and a shear wave. The interplay between these waves and the incoming homogeneous pressure wave in the fluid results in nontrivial behavior of the reflection coefficient. See Ref. 1 for further discussion.

Model Definition

A sketch of the modeled system is shown in Figure 1. An acoustic wave is incident on the water-sediment interface at an angle θ_0, defining the wave vector

$$\mathbf{k} = k_0(\sin(\theta_0), -\cos(\theta_0)) \qquad k_0 = \frac{\omega}{c}$$

where ω is the angular frequency $2\pi f$. The wave vector is used to define an incident plane wave of the form

$$P_{\text{in}}(\mathbf{x}) = 1\ \text{Pa} \cdot \exp(-i(\mathbf{k} \cdot \mathbf{x}))$$

where $\mathbf{x} = (x, y)$. The incident wave and wave vector are defined as variables in the model.

The fluid domain is modeled by classical pressure acoustics (lossless) solving the Helmholtz equation. The sediment domain is modeled by Biot's theory solving for both the pressure and the displacement field of the porous matrix. The coupled multiphysics problem is set up using the Acoustic-Poroelastic Waves Interaction multiphysics interface.

The problem is 2D and is modeled as a periodic structure using Floquet periodic conditions on both the acoustic and the porous domains. To ensure that the problem is well defined numerically, the domain size is set to depend on the wavelength in water (thus the frequency). The width of the computational unit cell W is equal to the wavelength and the height H is equal to two times the wavelength. The computational domain is truncated, in both the fluid and in the porous domain, using perfectly matched layers (PMLs). The thickness is set equal to one third of the wavelength.

The material parameters necessary for defining the properties of the porous domain are taken from Ref. 1 and some are calculated using these values. The porous material is here assumed to be sand. The most important parameters and their definitions are given in

Table 1 below.

TABLE 1: POROUS MATERIAL PARAMETERS FOR SAND

SYMBOL	VALUE	DESCRIPTION
μ_F	$1 \cdot 10^{-3}$ Pa·s	Dynamic viscosity of fluid (water)
ρ_F	1000 kg/m^3	Density of fluid (water)
ε_P	0.47	Porosity
κ_P	$1 \cdot 10^{-6}$ cm^2	Permeability
K	$4.36 \cdot 10^7$ Pa	Bulk modulus of frame (drained porous matrix)
K_s	$3.6 \cdot 10^{10}$ Pa	Bulk modulus of solid grains (constituting the frame)
α_B	$1 - K/K_s = 0.999$	Biot-Willis coefficient: Because it is very close to 1 the frame is said to be limp.
$\log_{dec}$	0.15	Logarithmic decrement factor
K_c	$(1+i \cdot \log_{dec}/\pi) \cdot K$	Complex bulk modulus of frame
G	$2.61 \cdot 10^7$ Pa	Shear modulus of frame (drained porous matrix)
G_c	$(1+i \cdot \log_{dec}/\pi) \cdot G$	Complex shear modulus of frame
ρ_s	2650 kg/m^3	Solid density (of material constituting the frame)
ρ_d	$(1-\varepsilon_P) \cdot \rho_s$ = 1404.5 kg/m^3	Drained density (of the porous matrix)
τ	1.25	Tortuosity (chosen, not specified in Ref. 1)
a	$4 \cdot 10^{-3}$ cm	Pore size parameter (chosen to have f_c = 100 Hz)
f_r	$\mu_F/(2\pi \cdot \rho_F \cdot a^2)$ = 99.5 Hz	Reference frequency: frequency at which the viscous boundary layer thickness (viscous penetration depth) is equal to the pore size parameter a. This value is automatically calculated bu COMSOL

Note: Further information about the physics solved by the Poroelastic Waves interface, about Biot's theory, and about the meaning of the material parameters is given in the documentation. Press Ctrl+F1 and browse to the *Acoustics Module* and the *User's Guide*. Theory is found in *Acoustic-Structure Interaction Interfaces > Theory for the Elastic Waves and Poroelastic Waves Interfaces*.

The reflection coefficient R and the absorption coefficient α are plotted for different combinations of the angle of incidence and the frequency (the model parameters). The reflection coefficient is the complex ratio of the reflected to the incident waves (at the interface) and is given by

$$R = \frac{P_{sc}}{P_{in}} = \frac{P_{tot} - P_{in}}{P_{in}}$$

where the subscript sc stands for scattered and tot for total. The absorption coefficient gives a measure of the relative absorbed energy and is given by

$$\alpha = 1 - |R|^2$$

The absolute value of the reflection coefficient R is shown in Figure 2 and Figure 3. In the first as function of the angle of incidence θ_0 and in the second as function of the frequency f. The absorption coefficient α corresponding the Figure 2 is shown in Figure 4. An interesting feature that can be deduced from Figure 3 is that the reflection at the interface acts as a filter with respect to a broadband signal, this is especially true at high angles of incidence.

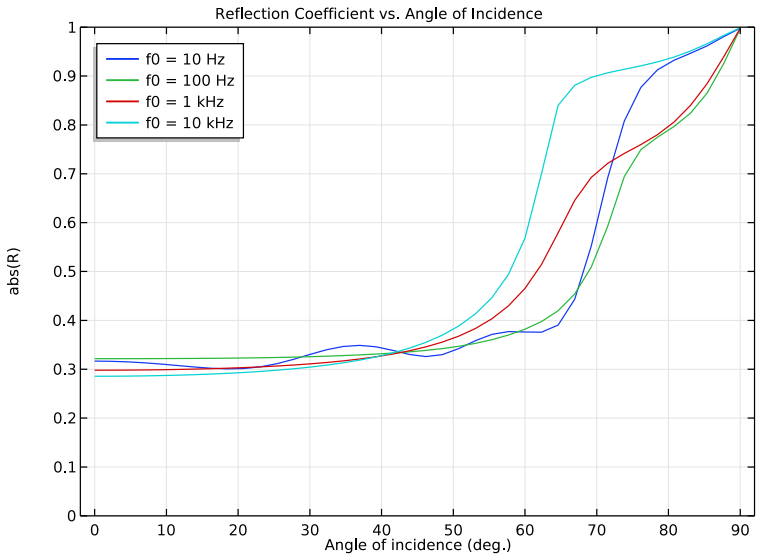


Figure 2: Absolute value of the reflection coefficient R as function of the angle of incidence for the driving frequencies 10 Hz, 100 Hz, 1 kHz, and 10 kHz.

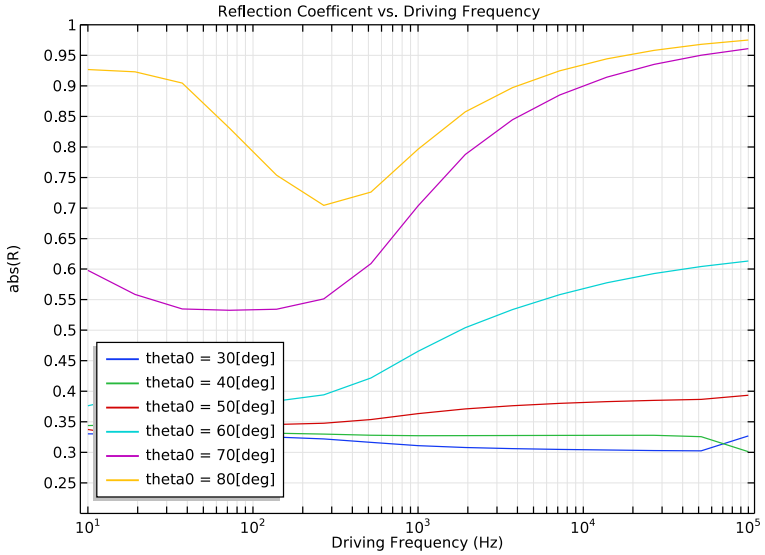


Figure 3: Absolute value of the reflection coefficient R as function of frequency for the angles of incidence 30°, 40°, 50°, 60°, 70°, and 80°.

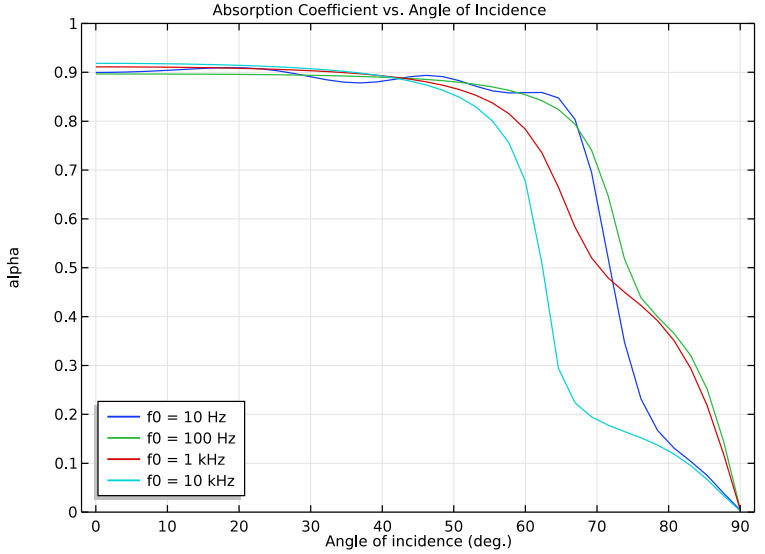


Figure 4: Absorption coefficient α as function of the angle of incidence for the driving frequencies 10 Hz, 100 Hz, 1 kHz, and 10 kHz.

As can be seen from Figure 2 the reflection coefficient has significantly different behavior for 3 of the driving frequencies at an incidence angle of, for example, 72°. The total pressure is shown in Figure 5 and the displacement field is shown in Figure 6. Note the different length scales as the computational domain (geometry) scales with the wavelength.

From the graphs in Figure 2 and Figure 4 it is evident that the behavior shifts for the different frequencies for increasing angles of incidence. At 90° the reflection coefficient is of course 1 and the absorption 0.

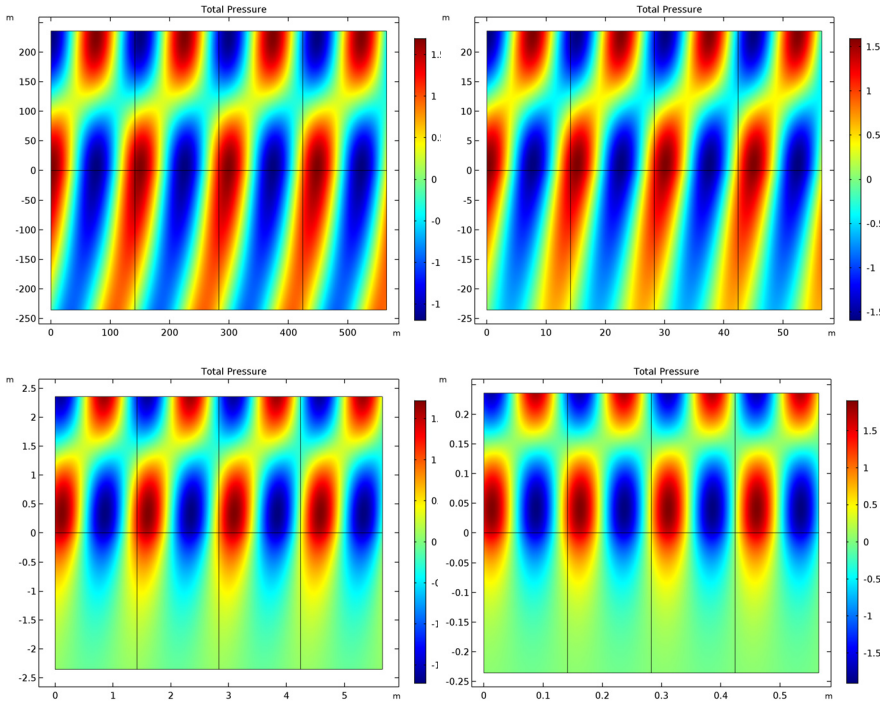


Figure 5: Total pressure distribution for an incidence angle of 72° for 10 Hz (top left), 100 Hz (top right), 1 kHz (bottom left), and 10 kHz (bottom right). Note that the computational domain size (geometry) scales with the wavelength.

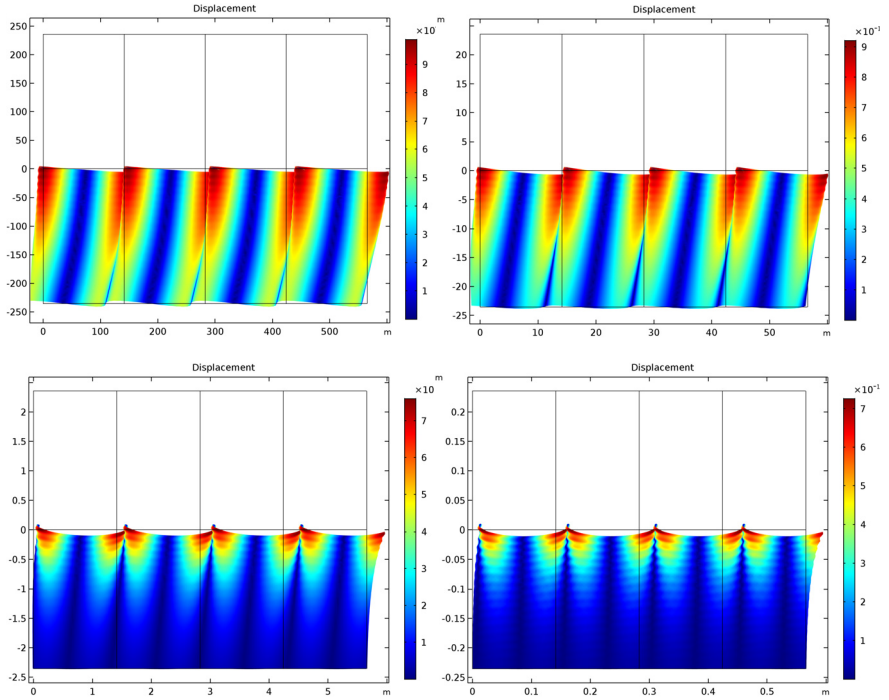


Figure 6: Displacement amplitude with deformation of the porous matrix for an incidence angle of 72° for 10 Hz (top left), 100 Hz (top right), 1 kHz (bottom left), and 10 kHz (bottom right). Note that the computational domain size (geometry) scales with the wavelength

Notes About the COMSOL Implementation

- In order for the model to be numerically well behaved for all frequencies studied, the geometry of the periodic unit cell is set to depend on the wavelength `lambda0`. This means that the geometry depends on the frequency. In COMSOL it is not permitted to have a frequency-dependent geometry in the Frequency Domain solver. To circumvent this problem, a parametric sweep is added. The sweep parameter is the frequency `f0`, which is then entered as the study frequency in the Frequency Domain study step.
- The periodic plots shown in Figure 5 and Figure 6 are set up using a Periodic 2D data set. The local cell variable (the cell number) is defined by the variable `arr1cellx` which exists in post processing when a periodic data set is set up.

- From the material parameters in Table 1 it is seen that the Biot-Willis coefficient α_B is close to 1. This means that the porous matrix is in a so-called limp configuration. At the other extreme, when α_B is close to the value of porosity ε_P, the configuration is named rigid (the skeleton is not moving). When a porous material is in one of these states it is possible to use an equivalent fluid model from the Poroacoustics domain feature (found in the Pressure Acoustics, Frequency Domain interface) to model the porous material. Use, for example, the Johnson-Champoux-Allard (limp frame option) for the case studied here. The advantage of using this model is that it is much less computationally expensive than the full Biot model. An equivalent fluid model generates or mimics the losses in the porous domain and therefore predicts the reflection and absorption coefficient. However, it does not account for the details of the movement of the skeleton and the propagation of the three different types of waves. Moreover, as soon as a porous material is neither limp nor rigid, the full Biot model of the Poroelastic Waves interface is necessary.

Reference

1. R.D. Stoll and T.-K. Kan, "Reflections of Acoustic Waves at a Water-Sediment Interface," *J. Acoust. Soc. Am.*, vol. 70, no. 1, pp. 149–156, 1981.

Application Library path: `Acoustics_Module/Underwater_Acoustics/reflections_water_sediment`

Modeling Instructions

From the **File** menu, choose **New**.

Start by adding the **Acoustic-Poroelastic Waves Interaction multiphysics** interface. The interface consists of the **Pressure Acoustics, Frequency Domain** interface, the **Poroelastic Waves** interface, and the necessary **Multiphysics** coupling.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Acoustics>Acoustic-Structure Interaction>Acoustic-Poroelastic Waves Interaction**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

6 Click **Done**.

GEOMETRY 1

First, load the model parameters from a file. The list contains definitions of material parameters (see Table 1), geometry lengths, and the two model parameters (frequency `f0` and angle of incidence `theta0`).

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `reflections_water_sediment_parameters.txt`.

Build the geometry, a sketch is given in Figure 1.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `W`.

4 In the **Height** text field, type `H`.

5 Click to expand the **Layers** section. Clear the **Layers on bottom** check box.

6 Select the **Layers on top** check box.

7 In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Hpml`

8 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `W`.

4 In the **Height** text field, type `H`.

5 Locate the **Position** section. In the **y** text field, type `-H`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Hpml

7 Click **Build All Objects**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

Under the **Definitions** node add variables as well as two coupling operators. The two operators act as probes picking up variable values. One samples a point in the water domain and the other evaluates the average over the water-sediment interface.

DEFINITIONS

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `reflections_water_sediment_variables.txt`.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 4 only.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 6 only.

Now, create selections to be used when setting up the physics.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Porous Domain` in the **Label** text field.

3 Select Domains 1 and 2 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Water Domain` in the **Label** text field.

3 Select Domains 3 and 4 only.

Proceed to set up the physics and boundary conditions of the problem. The condition at the water-sediment interface is defined under the **Multiphysics** node.

The properties of the porous material are edited after adding the porous material.

POROELASTIC WAVES (PELW)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Poroelastic Waves (pelw)**.

2 In the **Settings** window for **Poroelastic Waves**, locate the **Domain Selection** section.

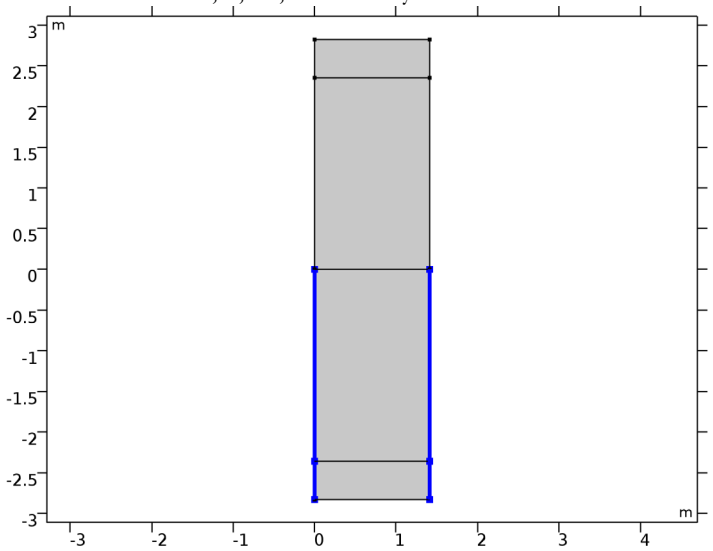
3 From the **Selection** list, choose **Porous Domain**.

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

Periodic Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 1, 3, 10, and 11 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 Specify the $\mathbf{k}_F$ vector as

k0x	X
k0y	Y

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

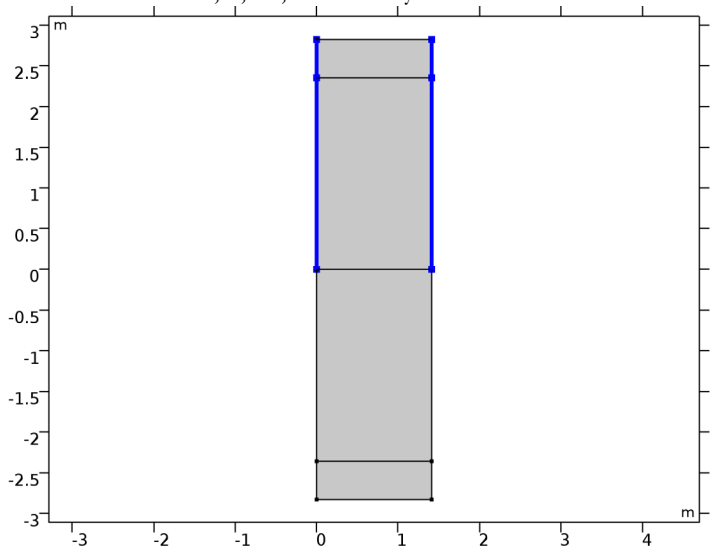
2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Water Domain**.

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

5 On the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

6 Select Boundaries 5, 7, 12, and 13 only.



7 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

8 From the **Type of periodicity** list, choose **Floquet periodicity**.

9 Specify the $\mathbf{k}_F$ vector as

k0x	x
k0y	y

10 In the **Model Builder** window, click **Pressure Acoustics, Frequency Domain (acpr)**.

11 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Typical Wave Speed for Perfectly Matched Layers** section.

12 In the c_{ref} text field, type `real(acpr.c_c)`.

Background Pressure Field 1

1 On the **Physics** toolbar, click **Domains** and choose **Background Pressure Field**.

2 In the **Settings** window for **Background Pressure Field**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Water Domain**.

4 Locate the **Background Pressure Field** section. From the **Pressure field type** list, choose **User defined**.

5 In the p_b text field, type `Pin`.

MATERIALS

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**, twice, to create two new blank materials.

Material 1 (mat1)

In the **Settings** window for **Material**, type `Water` in the **Label** text field.

Material 2 (mat2)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Material 2 (mat2)**.
2. In the **Settings** window for **Material**, type `Sediment` in the **Label** text field.

Now, go to the **Poroelastic Material** node and specify that the porous matrix material parameters be taken from the **Sediment** material that you just created. When done, return to the material, where it is now evident which parameters need to be defined.

POROELASTIC WAVES (PELW)

On the **Physics** toolbar, click **Pressure Acoustics, Frequency Domain (acpr)** and choose **Poroelastic Waves (pelw)**.

Poroelastic Material 1

1. In the **Model Builder** window, click **Poroelastic Material 1**.
2. In the **Settings** window for **Poroelastic Material**, locate the **Porous Matrix Properties** section.
3. From the **Porous elastic material** list, choose **Sediment (mat2)**.
4. Locate the **Fluid Properties** section. From the **Viscosity model** list, choose **Biot's high frequency range**.
5. From the **Specify** list, choose **Characteristic pore size**.
6. In the a text field, type `a`.

MATERIALS

Water (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water (mat1)**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rhoF	kg/m³	Basic
Speed of sound	c	c0	m/s	Basic
Dynamic viscosity	mu	muF	Pa·s	Basic
Compressibility of fluid	chif	1/Kf	1/Pa	Basic

Sediment (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Sediment (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk modulus	K	Kbc	N/m²	Bulk modulus and shear modulus
Shear modulus	G	Gc	N/m²	Bulk modulus and shear modulus
Density	rho	rhod	kg/m³	Basic
Porosity	epsilon	epsilonP	1	Basic
Permeability	kappa	kappaP	m²	Basic
Biot-Willis coefficient	alphaB	alphaB0	1	Poroelastic material
Tortuosity factor	tau	tau0	1	Poroacoustics model

MULTIPHYSICS

Now is a good time to inspect the **Multiphysics** node and look at the multiphysics couplings. Click on **Acoustic-Porous Boundary 1**, you can see that it is active on the interface between the water domain and the porous sediment. In this model there are no structures so the last multiphysics coupling is not used.

The last step before building the mesh is to set up and define the perfectly matched layers (PMLs). Define one for the poroelastic domain and another for the water domain.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 1 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

Perfectly Matched Layer 2 (pml2)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 4 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **Rational**.

5 From the **Physics** list, choose **Pressure Acoustics, Frequency Domain (acpr)**.

Build the mesh by following the instructions below. When modeling systems with interfaces between porous domains and fluid domains the mesh is important; it should be very fine at the interface and it is good practice to use a single boundary layer mesh element in the porous domain. It is also important for the periodic conditions to have the same mesh at source and destination. Ideally, a mesh sensitivity analysis should be made for models of this kind.

MESH 1

Edge 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Boundaries 11 and 12 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Edge 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. Select the **Maximum element size** check box.

5 In the associated text field, type `lambda0/15`.

Size 2

1 Right-click **Edge 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 **Point**.

4 Select Point 8 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 `lambda0/40`.

8 Click **Build Selected**.

Copy Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Copy Edge**.

2 Select Boundaries 11 and 12 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundaries 3 and 5 only.

6 Click **Build Selected**.

Free Triangular 1

1 Right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 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. Select the **Maximum element size** check box.

5 In the associated text field, type `lambda0/15`.

6 Select the **Maximum element growth rate** check box.

7 In the associated text field, type `1.1`.

Size 2

1 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 **Boundary**.

4 Select Boundary 6 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 `lambda0/40`.

Size 3

1 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 **Point**.

4 Select Points 3 and 8 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 `lambda0/40`.

8 Click **Build Selected**.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundaries 1 and 7 only.

4 In the **Settings** window for **Distribution**, locate the **Distribution** section.

5 In the **Number of elements** text field, type `20`.

6 Click **Build Selected**.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.

3 In the **Number of boundary layers** text field, type `1`.

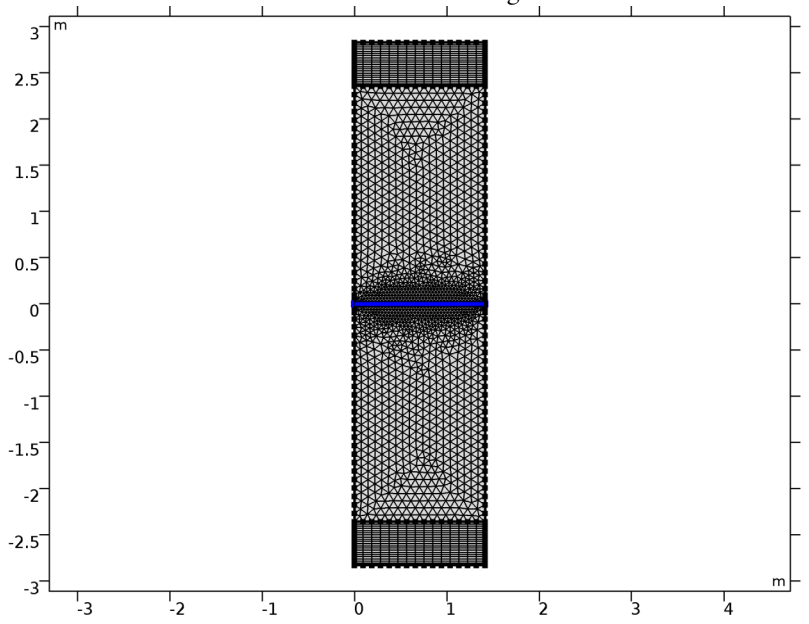
4 From the **Thickness of first layer** list, choose **Manual**.

5 In the **Thickness** text field, type `lambda0/400`.

6 Select Boundary 6 only.

7 Click **Build All**.

The finalized mesh should look like that in the figure below.



STUDY 1

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then 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 `f0`.

4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 From the **Sweep type** list, choose **All combinations**.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Incidence angle)	`range(0,pi/2/39,pi/2)`	

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0	10 100 1000 10000	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Add a selection to the data set to only plot results in the physical domains (not in the PMLs). You can study the behavior of the PMLs by removing this selection again or creating a separate data set without the selection.

Data Sets

In the **Model Builder** window, expand the **Results** node.

Study 1/Parametric Solutions 1 (sol2)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/ Parametric Solutions 1 (sol2)**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 3 only.

Now, also add an array dataset in order to easily plot the result in the periodic structure.

Array 2D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Array 2D**.

2 In the **Settings** window for **Array 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Locate the **Array Size** section. In the **X size** text field, type `4`.

5 Click to expand the **Advanced** section. Select the **Define variables** check box.

Now, create four different 2D plots representing the incident pressure (background pressure), the reflected pressure (scattered pressure), the total pressure, and the displacement in the porous matrix.

2D Plot Group 1

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Incident Pressure` in the **Label** text field.

Surface 1

1 Right-click **Incident Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_b`.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Incident Pressure** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.

3 In the **X component** text field, type `k0x`.

4 In the **Y component** text field, type `k0y`.

5 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. In the **Points** text field, type `5`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Incident Pressure

1 In the **Model Builder** window, under **Results** click **Incident Pressure**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (theta0)** list, choose **0.12083**.

Select any of the desired combinations of the frequency `f0` and the angle of incidence `theta0` and plot to have a look at the results. You may need to use the **Zoom Extent** tool as the geometry depends on the frequency.

5 On the **Incident Pressure** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

2D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Reflected Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (theta0)** list, choose **0.12083**.

Surface 1

1 Right-click **Reflected Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_s`.

4 On the **Reflected Pressure** toolbar, click **Plot**.

Surface 2

1 In the **Model Builder** window, under **Results** right-click **Reflected Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `pelw.p_s`.

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

5 On the **Reflected Pressure** toolbar, click **Plot**.

Note that this gives the scattered field in the water domain while it is equal to the total acoustic field in the porous domain.

In the next figure, plot the total pressure (air and porous domains). Use the periodic data set to display the periodic nature of the modeled system.

The total pressure is written in a general way for the N^{th} unit cell number given by the cell variable `arr1cellx`.

2D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Total Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Array 2D 1**.

4 From the **Parameter value (theta0)** list, choose **0.12083**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Total Pressure`.

Surface 1

1 Right-click **Total Pressure** and choose **Surface**.

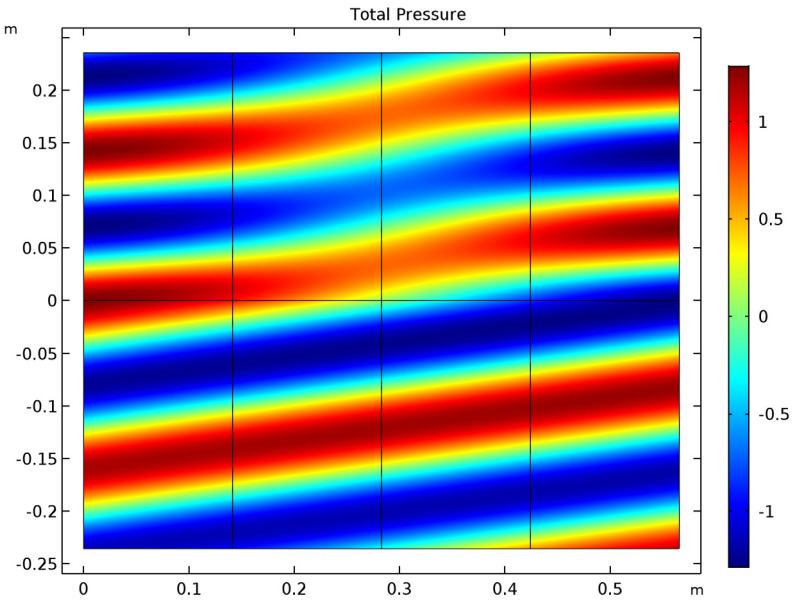
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `if(dom==3,acpr.p_t,pelw.p_t)*exp(-i*arr1cellx*k0x*W)`.

The `if()` statement plots `acpr.p_t` if in the water domain (domain 3) else it plots `pelw.p_t` (the sediment domain).

4 On the **Total Pressure** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.



Now, plot the total displacement of the porous matrix. Again use the periodic data set to represent the periodic nature of the modeled system, just as for the total pressure.

In the first plot, the displacement amplitude is written in a general way for the N^{th} unit cell.

2D Plot Group 4

1 On 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.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Displacement`.

5 Locate the **Data** section. From the **Data set** list, choose **Array 2D 1**.

6 From the **Parameter value (theta0)** list, choose **0.12083**.

Surface 1

1 Right-click **Displacement** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `sqrt(u^2*exp(-i*arr1cellx*k0x*W)^2+v^2*exp(-i*arr1cellx*k0x*W)^2)`.

Deformation 1

1 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **x component** text field, type `u*exp(-i*arr1cellx*k0x*W)`.

4 In the **y component** text field, type `v*exp(-i*arr1cellx*k0x*W)`.

5 Locate the **Scale** section. Select the **Scale factor** check box.

6 In the associated text field, type `4e9`.

7 On the **Displacement** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

The total pressure and the displacement are depicted in Figure 5 and Figure 6 for several combinations of the driving frequency and the angle of incidence.

Create Figure 2 from Ref. 1 by plotting the absolute value of the reflection coefficient as function of the angle of incidence for four values of the frequency `f0`.

1D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Reflection Coefficient vs theta0` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Angle of incidence (deg.)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `abs(R)`.

8 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

9 In the **Title** text area, type `Reflection Coefficient vs. Angle of Incidence`.

10 Locate the **Axis** section. Select the **Manual axis limits** check box.

11 In the **x minimum** text field, type `-2`.

12 In the **x maximum** text field, type `92`.

13 In the **y minimum** text field, type `0`.

14 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Reflection Coefficient vs theta0** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (f0)** list, choose **From list**.

5 In the **Parameter values (f0)** list, select **10**.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
absR	1	Absolute value of reflection coefficient

7 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.

8 From the **Parameter** list, choose **Expression**.

9 In the **Expression** text field, type `theta0/pi*180`.

10 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
f0 = 10 Hz

12 On the **Reflection Coefficient vs theta0** toolbar, click **Plot**.

Global 2

1 Right-click **Results>Reflection Coefficient vs theta0>Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 In the **Parameter values (f0)** list, select **100**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
f0 = 100 Hz

Global 3

1 Right-click **Results>Reflection Coefficient vs theta0>Global 2** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 In the **Parameter values (f0)** list, select **1000**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
f0 = 1 kHz

Global 4

1 Right-click **Results>Reflection Coefficient vs theta0>Global 3** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 In the **Parameter values (f0)** list, select **10000**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
f0 = 10 kHz

5 On the **Reflection Coefficient vs theta0** toolbar, click **Plot**.

The plot should look like the one in Figure 2.

Next, create a second study to sweep over the frequency for a given number of angles of incidence.

ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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 `f0`.

4 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 From the **Sweep type** list, choose **All combinations**.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta0 (Incidence angle)	`30[deg] 40[deg] 50[deg] 60[deg] 70[deg] 80[deg]`	

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
f0	`10^{range(1,4/14,5)}`	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Create Figure 3 from Ref. 1 by plotting the absolute value of the reflection coefficient as function of the frequency `f0` for six values of the angle of incidence.

1D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Reflection Coefficient vs f0` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **None**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Reflection Coefficent vs. Driving Frequency`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Driving Frequency (Hz)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `abs(R)`.

10 Locate the **Axis** section. Select the **Manual axis limits** check box.

11 On the **Reflection Coefficient vs f0** toolbar, click **Plot**.

12 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 Right-click **Reflection Coefficient vs f0** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Parametric Solutions 2 (sol8)**.

4 From the **Parameter selection (theta0)** list, choose **Manual**.

5 In the **Parameter indices (1-6)** text field, type `1`.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
absR	1	Absolute value of reflection coefficient

7 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

8 From the **Parameter** list, choose **Expression**.

9 In the **Expression** text field, type `f0`.

10 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
theta0 = 30[deg]

Global 2

1 Right-click **Results>Reflection Coefficient vs f0>Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 In the **Parameter indices (1-6)** text field, type `2`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
theta0 = 40[deg]

Global 3

1 Right-click **Results>Reflection Coefficient vs f0>Global 2** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 In the **Parameter indices (1-6)** text field, type `3`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`theta0 = 50[deg]`

Global 4

1 Right-click **Results>Reflection Coefficient vs f0>Global 3** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 In the **Parameter indices (1-6)** text field, type `4`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`theta0 = 60[deg]`

Global 5

1 Right-click **Results>Reflection Coefficient vs f0>Global 4** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 In the **Parameter indices (1-6)** text field, type `5`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`theta0 = 70[deg]`

Global 6

1 Right-click **Results>Reflection Coefficient vs f0>Global 5** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 In the **Parameter indices (1-6)** text field, type `6`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`theta0 = 80[deg]`

5 On the **Reflection Coefficient vs f0** toolbar, click **Plot**.

The plot should look like the one in Figure 3.

Finally, plot the absorption coefficient as function of the frequency `f0` for six values of the angle of incidence. Simply duplicate the plot with the absorption coefficient, change `absR` to `alpha` in the plots, and rename titles where necessary.

Reflection Coefficient vs theta0.1

1 In the **Model Builder** window, under **Results** right-click **Reflection Coefficient vs theta0** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Absorption Coefficient vs theta0` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Absorption Coefficient vs. Angle of Incidence`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `alpha`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 In the **Model Builder** window, expand the **Results>Absorption Coefficient vs theta0** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`alpha`	1	

Global 2

1 In the **Model Builder** window, under **Results>Absorption Coefficient vs theta0** click **Global 2**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`alpha`	1	

Global 3

1 In the **Model Builder** window, under **Results>Absorption Coefficient vs theta0** click **Global 3**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`alpha`	1	

Global 4

1 In the **Model Builder** window, under **Results>Absorption Coefficient vs theta0** click **Global 4**.

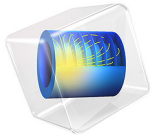
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`alpha`	1	

4 On the **Absorption Coefficient vs theta0** toolbar, click **Plot**.

The plot should look like the one in Figure 4.



Surface Acoustic Wave Gas Sensor

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

A surface acoustic wave (SAW) is an acoustic wave propagating along the surface of a solid material. Its amplitude decays rapidly, often exponentially, with the depth of the material. SAWs are featured in many kinds of electronic components, including filters, oscillators, and sensors. SAW devices typically use electrodes on a piezoelectric material to convert an electric signal to a SAW, and back again.

In this model, you investigate the resonance frequencies of a SAW gas sensor. The sensor consists of an interdigitated transducer (IDT) etched onto a piezoelectric $LiNbO_3$ (lithium niobate) substrate and covered with a thin polyisobutylene (PIB) film. The mass of the PIB film increases as PIB selectively adsorbs CH_2Cl_2 (dichloromethane, DCM) from air. This causes a shift in resonance which slightly lowers the resonance frequency for the same SAW mode.

Model Definition

Figure 1 shows a conceptual view of the gas sensor in this model.

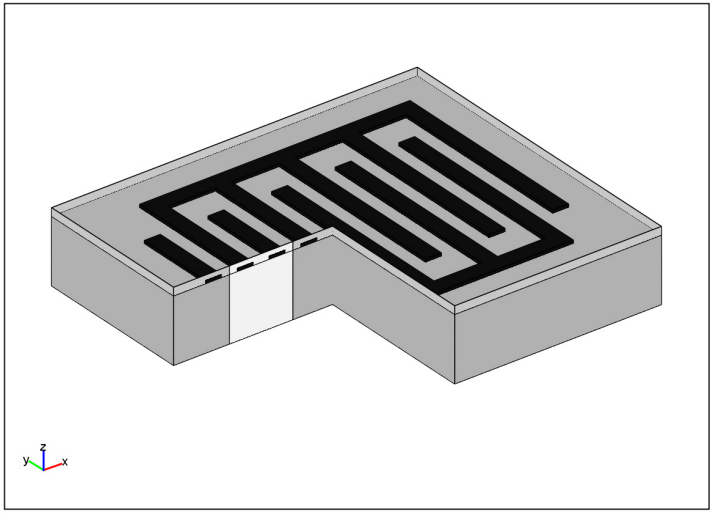


Figure 1: SAW gas sensor, showing the IDT electrodes (in black), the thin PIB film (light gray), and the $LiNbO_3$ substrate (dark gray). For the sake of clarity, the dimensions are not to scale and the IDT has fewer electrodes than in common devices. A slice of the geometry is removed to reveal the modeled unit cell (in white).

IDTs used in SAW devices may have hundreds of identical electrodes, and each electrode can be about 100 times longer than it is wide. You can therefore neglect the edge effects and reduce the model geometry to the periodic unit cell shown in Figure 2. The height of this cell does not have to extend all the way to the bottom of the substrate but only a few wavelengths down, so that the SAW has almost died out at the lower boundary.

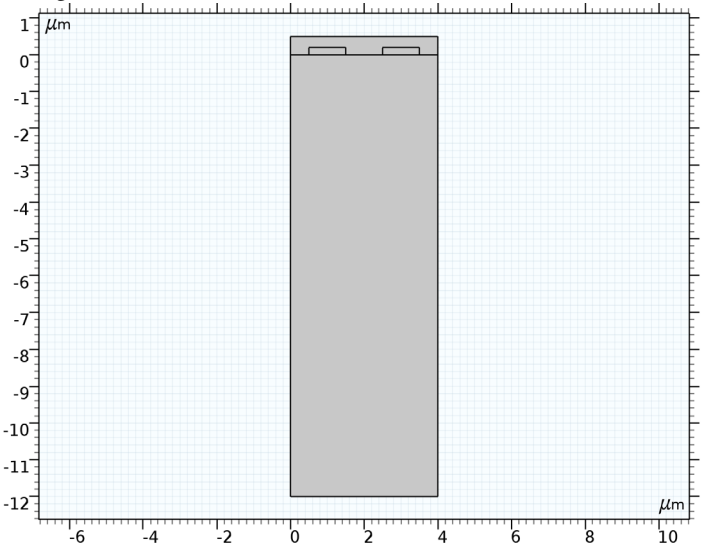


Figure 2: The geometry of the SAW unit cell used in this model. A 500 nm PIB film covers two 1 μm-wide electrodes on top of the $LiNbO_3$ substrate. The substrate domain has a total height of 12 μm.

Set up the model using the predefined Piezoelectric Devices multiphysics interface. In 2D, a Plane Strain assumption is used for the Solid Mechanics interface. Hence the out-of-plane strain component is zero. This should be a valid assumption, considering that the SAW is generated in the plane of the model and hence any variation in the out-of-plane direction can be considered minimal.

BOUNDARY CONDITIONS

In order to define the model, you need to apply structural and electrical boundary conditions.

As one can assume that the surface wave dies off within two to three wavelengths from the surface, the lower boundary is fixed. This enforces a zero structural displacement but does not contribute to any significant reflection from the lower boundary back into the bulk of the substrate as long as we are observing surface waves and in particular Rayleigh waves.

The electrodes have a much higher electrical conductivity compared to PIB and $LiNbO_3$. Hence, one can expect each of the electrodes to be isopotential. This is why you do not need to model the domains that constitute the electrodes but can simply use appropriate boundary conditions on all the outer boundaries of each electrode to indicate what type of isopotential state it is in. The boundaries of the left terminal is set to electrical ground, and those of the right terminal are assigned to a Floating Potential with zero surface charge accumulation. This combination of electrical boundary conditions corresponds to an open circuit configuration, which is typically suitable for sensing applications.

Use periodic boundary conditions to dictate that the electric potential and displacements are the same along both vertical boundaries of the geometry. When using periodic boundary conditions, one needs to ensure that the mesh on the vertical boundaries on the left of the unit cell and the vertical boundaries on the right of the unit cell are identical. This is achieved by first creating the mesh on the vertical boundaries on the left and then using the Copy Edge feature to create the exact same mesh on the vertical boundaries on the right.

All other boundaries are left to the default boundary conditions which are Free for the Solid Mechanics interface and Zero Charge for the Electrostatics interface, respectively.

MATERIAL PROPERTIES

The substrate used in the simulation is YZ-cut $LiNbO_3$ with the following properties (cited in Ref. 2):

- *Elasticity matrix:*

$$\begin{bmatrix} 2.424e11 & 0.752e11 & 0.752e11 & 0 & 0 & 0 \\ & 2.03e11 & 0.573e11 & 0 & 0.085e11 & 0 \\ & & 2.03e11 & 0 & -0.085e11 & 0 \\ & & & 0.752e11 & 0 & 0.085e11 \\ & & & & 0.595e11 & 0 \\ & & & & & 0.595e11 \end{bmatrix}$$

- *Coupling matrix:*

$$\begin{bmatrix} 1.33 & 0.23 & 0.23 & 0 & 0 & 0 \\ 0 & 0 & 0 & -2.5 & 0 & 3.7 \\ 0 & -2.5 & 2.5 & 0 & 3.7 & 0 \end{bmatrix}$$

- *Relative permittivity:*

$$\begin{bmatrix} 28.7 & 0 & 0 \\ & 85.2 & 0 \\ & & 85.2 \end{bmatrix}$$

The density of the PIB film is from Ref. 1. The Poisson's ratio is considered to be 0.48, and the Young's modulus is set to 10 GPa.

The adsorption of DCM gas is represented as a slight increase of the overall density of the PIB film as shown in the following expression.

$$\rho = \rho_{PIB} + switch \cdot \rho_{\mathrm{DCM,PIB}}$$

In this model, you use a parameter, *switch*, whose value can be either 0 or 1. This allows to solve the model for two cases; once without the effect of adsorbed gas and once with the effect of the adsorbed DCM gas in PIB.

When the sensor is exposed to 100 ppm of DCM in air at atmospheric pressure and room temperature, the "partial density" of DCM in the PIB film can be calculated from

$$\rho_{\mathrm{DCM,PIB}} = KMc$$

where $K = 10^{1.4821}$ (Ref. 1) is the air/PIB partition coefficient for DCM, M is its molar mass, and c

$$c = (c_0 p)/(RT)$$

is its concentration in air. The DCM concentration, c in moles/m^3 is computed using the Gas Law. Here c_0 is the concentration in parts per million, p is the pressure, T is the temperature, and R is the gas constant. Any effects of the DCM adsorption on the material properties other than the density are neglected.

Most of the material properties and factors affecting them have been parametrized as shown in Table 1. This easily allows the model to be adapted for other materials and operating conditions.

TABLE 1: LIST OF PARAMETERS

NAME	EXPRESSION	DESCRIPTION
p	1[atm]	Air pressure
T	25[degC]	Air temperature
c0	100	DCM concentration in ppm
c_DCM_air	1e-6*c0*p/(R_const*T)	DCM concentration in air
M_DCM	84.93[g/mol]	Molar mass of DCM
K	10^1.4821	PIB/air partition constant for DCM
rho_DCM_PIB	K*M_DCM*c_DCM_air	Mass concentration of DCM in PIB
rho_PIB	0.918[g/cm^3]	Density of PIB
E_PIB	10[GPa]	Young's modulus of PIB
nu_PIB	0.48	Poisson's ratio of PIB
eps_PIB	2.2	Relative permittivity of PIB
switch	0	Switch for adding DCM density
vR	3488[m/s]	Rayleigh wave velocity
width	4[um]	Width of unit cell
f0	vR/width	Estimated SAW frequency
t_PIB	0.5[um]	PIB thickness

ESTIMATING THE SAW FREQUENCY

The use of periodic boundary condition implies that the frequencies of interest correspond to wavelengths that are integer fractions of the width of the geometry. The lowest SAW eigenmode has its wavelength equal to the width of the geometry, that is, 4 µm. Using this along with the Rayleigh wave velocity for the given piezoelectric substrate material, one can find an estimate of the resonance frequency of interest. The information can be used in the eigenfrequency solver, which helps it to find out the resonance frequencies close to this estimated number. In this model, you use a YZ-cut $LiNbO_3$ whose Rayleigh wave velocity (vR) is around 3488 m/s. This gives an estimate of the lowest SAW frequency (f0) to be 872 MHz.

Results and Discussion

The anti-resonance and resonance frequencies evaluate to approximately 850 MHz and 855 MHz, respectively. Figure 3 and Figure 4 show the corresponding SAW modes. Figure 5 and Figure 6 show the electric potential distribution characteristics for these solutions.

Exposing the sensor to a 100 ppm concentration of DCM in air leads to a resonance frequency shift of approximately 200 Hz downwards. This is computed by evaluating the resonance frequency before and after adding the density of adsorbed DCM to that of the PIB domain.

Note that the computational mesh is identical in both these solutions. This implies that the relative error of the frequency shift is similar to that of the resonance frequency itself. Thus, the shift is accurately evaluated despite being a few magnitudes smaller than the absolute error of the resonance frequency.

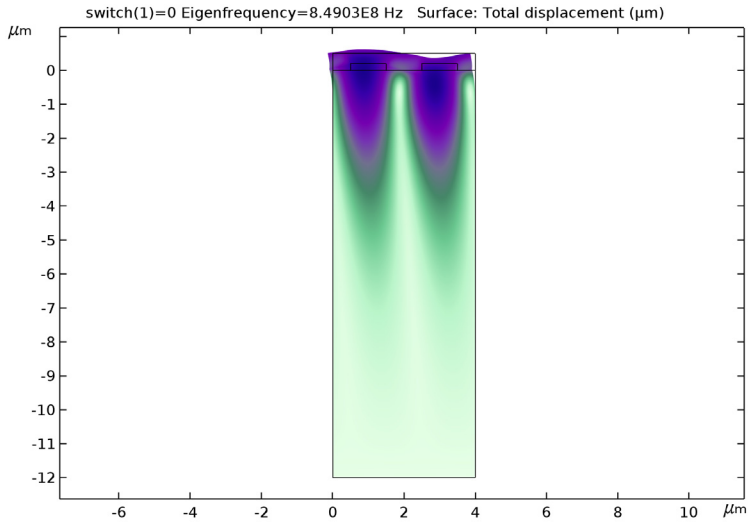


Figure 3: Deformed shape plot of the anti-resonance SAW mode.

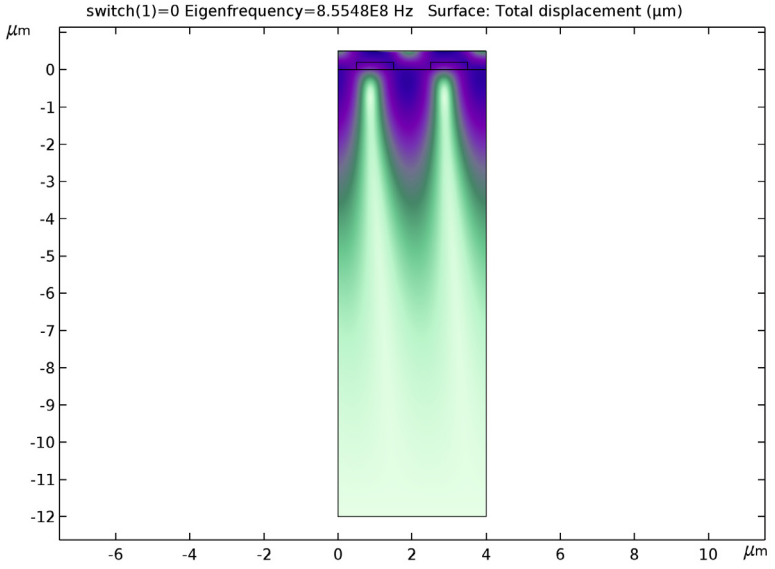


Figure 4: Deformed shape plot of the resonance SAW mode.

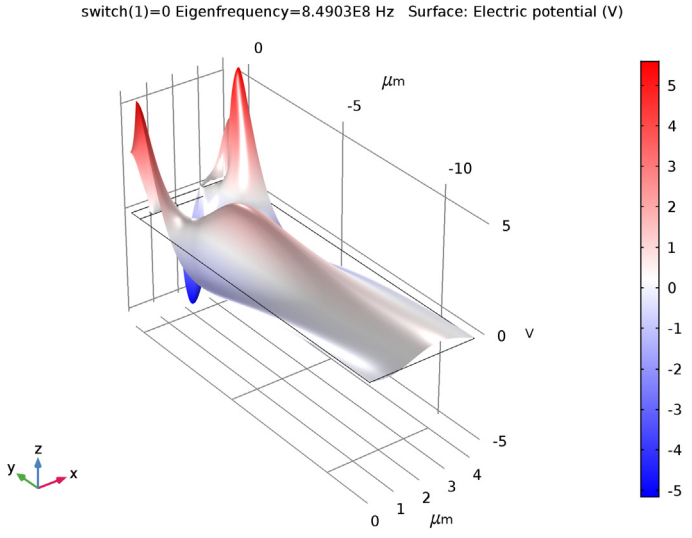


Figure 5: Electric potential distribution and deformations at anti-resonance, anti-symmetric with respect to the center of each electrode.

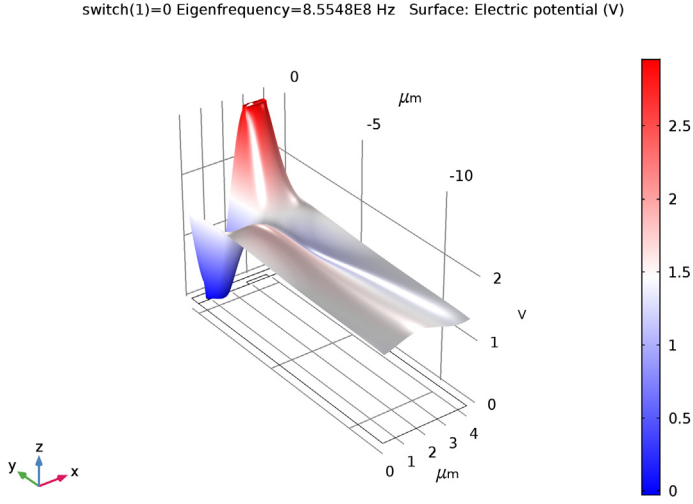


Figure 6: Electric potential distribution at resonance, symmetric with respect to the center of each electrode.

References

1. K. Ho, E. R. Lindgren, K. S. Rawlinson, L. K. McGrath, and J. L. Wright, "Development of a Surface Acoustic Wave Sensor for In-Situ Monitoring of Volatile Organic Compounds," *Sensor*s vol. 3, pp. 236–247, 2003.

2. S. Ahmadi, F. Hassani, C. Korman, M. Rahaman, and M. Zaghloul, "Characterization of Multi- and Single-layer Structure SAW Sensor," *Sensors 2004*, *Proceedings of IEEE*, vol. 3, pp. 1129–1132, 2004.

Application Library path: `Acoustics_Module/Ultrasound/saw_gas_sensor`

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**.

2 In the **Select Physics** tree, select **Structural Mechanics>Piezoelectric Devices**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

For quicker modeling, load the parameters from a file.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `saw_gas_sensor_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 **µm**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `width`.

4 In the **Height** text field, type `3*width+t_PIB`.

5 Locate the **Position** section. In the **y** text field, type `-3*width`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (µm)
`Layer 1`	`t_PIB`

7 Select the **Layers on top** check box.

8 Clear the **Layers on bottom** check box.

9 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `width/4`.

4 In the **Height** text field, type `0.4*t_PIB`.

5 Locate the **Position** section. In the **x** text field, type `width/8`.

6 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

Copy 1 (copy1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **r2** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `width/2`.

5 Click **Build All Objects**.

Before creating materials, define the domains where each physics apply.

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)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Electrostatics (es)**.

The electrical equations are not solved in the aluminum electrodes because they are assumed to be perfect conductors compared to LiNbO3 and PIB, and hence we consider them to be equipotential regions.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electrostatics (es)**.

2 Select Domains 1 and 2 only.

Charge Conservation, Piezoelectric 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electrostatics (es)** click **Charge Conservation, Piezoelectric 1**.

2 Select Domain 1 only.

MATERIALS

Material 1 (mat1)

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 `LiNbO3` 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)	cE	{242.4[GPa], 75.2[GPa],203[GPa], 75.2[GPa],57.3[GPa], 203[GPa],0,0,0, 75.2[GPa],0,8.5[GPa], -8.5[GPa],0, 59.5[GPa],0,0,0, 8.5[GPa],0,59.5[GPa]}	Pa	Stress-charge form
Coupling matrix	eES	{1.33,0,0,0.23,0,-2.5,0.23,0,2.5,0,-2.5,0,0,0,3.7,0,3.7,0}	C/m²	Stress-charge form
Relative permittivity	epsilonrS	{28.7,85.2,85.2}	1	Stress-charge form
Density	rho	4647	kg/m³	Basic

Alternately, you can click the **Edit** button below the **Output properties** table and use the matrix inputs to enter `cE`, `e`, and `epsilonrS` according to the material data shown under the **Model Definition** section.

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `PIB` 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	E_PIB	Pa	Basic
Poisson's ratio	nu	nu_PIB	1	Basic
Density	rho	rho_PIB+switch*rho_DCM_PIB	kg/m³	Basic
Relative permittivity	epsilonr	eps_PIB	1	Basic

ADD MATERIAL

1 On 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>Aluminum**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Aluminum (mat3)

1 Select Domains 3 and 4 only.

2 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

Periodic Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 1, 3, 16, and 17 only.

This defines periodic condition for the **Solid Mechanics** physics. By default, the periodicity type is set to **Continuity** for all dependent variables.

ELECTROSTATICS (ES)

Ground the left electrode by applying boundary condition on its edges.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electrostatics (es)**.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 Select Boundaries 6–9 only.

Apply a **Floating Potential** on the edges of the right electrode.

Floating Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Floating Potential**.

2 Select Boundaries 11–14 only.

Define periodic boundary condition for the **Electrostatics** physics.

Periodic Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 1, 3, 16, and 17 only.

MESH 1

Edge 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Edge**.

2 Select Boundaries 1 and 3 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Edge 1** and choose **Distribution**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 4.

Edge 1

Define an increasing mesh size along the thickness of the piezoelectric substrate.

Distribution 2

1 Right-click **Edge 1** and choose **Distribution**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** text field, type 25.

6 In the **Element ratio** text field, type 25.

7 Select the **Reverse direction** check box.

8 Click **Build Selected**.

Copy Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Copy Edge**.

2 Select Boundaries 1 and 3 only.

3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.

4 Select the **Active** toggle button.

5 Select Boundaries 16 and 17 only.

Free Quad 1

1 Right-click **Mesh 1** and choose **Free Quad**.

2 In the **Settings** window for **Free Quad**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2–4 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Quad 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

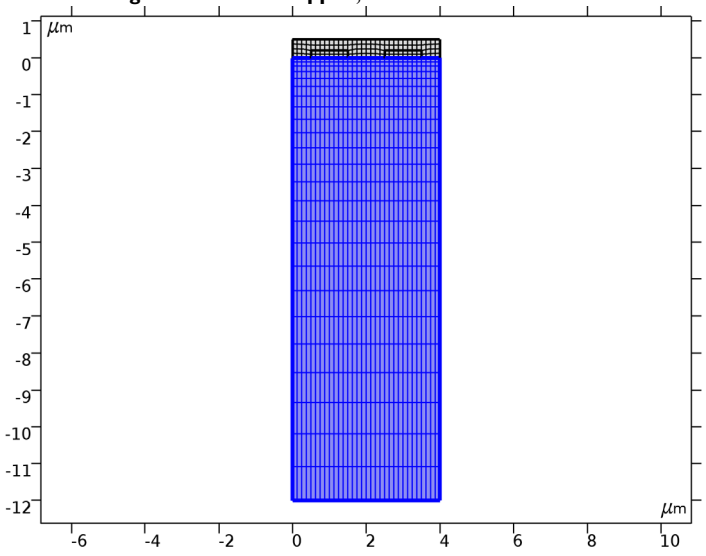
4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

5 In the associated text field, type `t_PIB/4`.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.



STUDY 1

Set up a **Parametric Sweep** to solve the model with and without the adsorbed species on the sensor. The parameter switch is used to solve the model once without adding the density of DCM and once with the added density of adsorbed DCM in PIB.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
switch (Switch for adding DCM density)	0 1	

Search for eigenfrequencies near the estimated resonance frequency.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Search for eigenfrequencies around** check box.

4 In the associated text field, type `f0`.

5 On the **Study** toolbar, click **Compute**.

RESULTS

Mode Shape (solid)

The default plot shows the anti-resonance SAW mode near 850 MHz. Adjust the deformed shape plot to see the effect of the wave localization near the surface.

1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (switch)** list, choose **0**.

4 In the **Model Builder** window, expand the **Mode Shape (solid)** node.

Deformation

1 In the **Model Builder** window, expand the **Results>Mode Shape (solid)>Surface 1** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `100`.

5 On the **Mode Shape (solid)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the resulting plot to that in Figure 3.

Mode Shape (solid)

Plot the second mode shape near 855 MHz, which is the resonance SAW mode.

1 In the **Model Builder** window, under **Results** click **Mode Shape (solid)**.

2 Click **Plot**.

3 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

4 From the **Eigenfrequency (Hz)** list, choose **8.5548E8**.

5 On the **Mode Shape (solid)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the resulting plot to that in Figure 4.

Electric Potential (es)

To visualize the electric potential distribution for the eigenmodes, follow these steps.

Surface 1

1 In the **Model Builder** window, expand the **Results>Electric Potential (es)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **WaveLight**.

Electric Potential (es)

1 Right-click **Results>Electric Potential (es)>Surface 1** and choose **Height Expression**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Parameter value (switch)** list, choose **0**.

4 On the **Electric Potential (es)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

6 From the **Eigenfrequency (Hz)** list, choose **8.5548E8**.

7 On the **Electric Potential (es)** toolbar, click **Plot**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

Derived Values

To see all computed eigenfrequencies as a table, follow these steps.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Table columns** list, choose **Outer solutions**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Solver>freq - Frequency**.

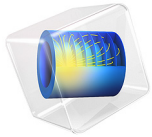
6 Click **New Table**.

TABLE

1 Go to the **Table** window.

2 Click **Full Precision** in the window toolbar.

The first 6 digits of the eigenfrequency are the same. Subtracting the new value from the previous value shows that the eigenfrequency with gas exposure is lower by 200 Hz.



Test Bench Car Interior

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

In this application, a point source generates a pressure wave in the Sound Brick, a test bench car interior (Ref. 1). The sound pressure level is measured at another point and at a range of frequencies high enough that a good mesh resolution is required to properly resolve the wave. To get an idea of the accuracy of the model, the sound pressure level response is studied using four different mesh resolutions. Finally, the model is also solved using an iterative solver.

Model Definition

The geometrical and material parameters in this model come from Ref. 2. The geometry of the Sound Brick is shown in Figure 1. The model solves for the acoustic pressure p in the frequency domain using the Pressure Acoustics, Frequency Domain interface.

A monopole point flow source of strength $Q_S = 10^{-5}$ m^3/s located at the point $\mathbf{R}_0 = (0.21,\ 0,\ 1.28)$ drives the system. The walls of the Sound Brick are assumed to be perfectly reflecting and have the default sound hard boundary condition. The sound pressure level is measured at the point $\mathbf{R}_1 = (1.34,\ 1.22,\ 0.8)$ at a range of frequencies from 490 Hz to 500 Hz.

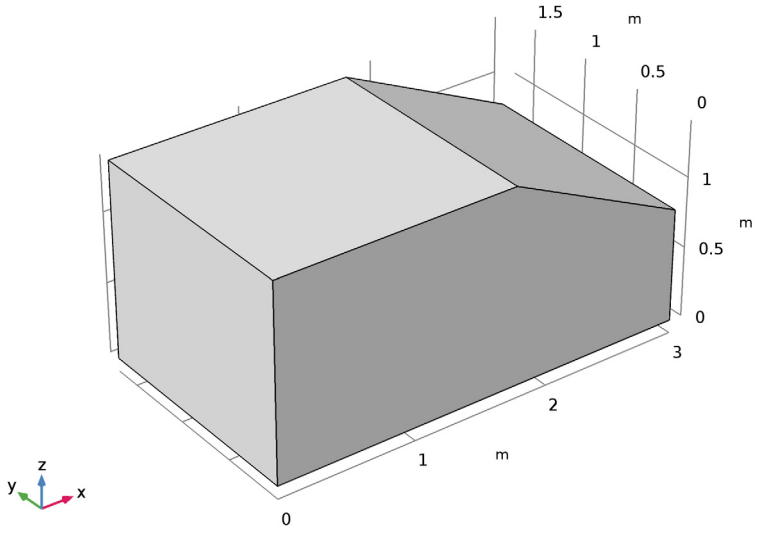


Figure 1: The geometry of the test bench car interior.

The dimensions of the test bench car are length × height × depth = 3.0 m × 1.4 m × 1.7 m. The windshield has its lower end 0.8 m above the floor and with an inclination such that the entire volume of the geometry is 6.5 m^3.

Results and Discussion

Figure 2 shows the instantaneous pressure distribution in the car interior at 500 Hz.

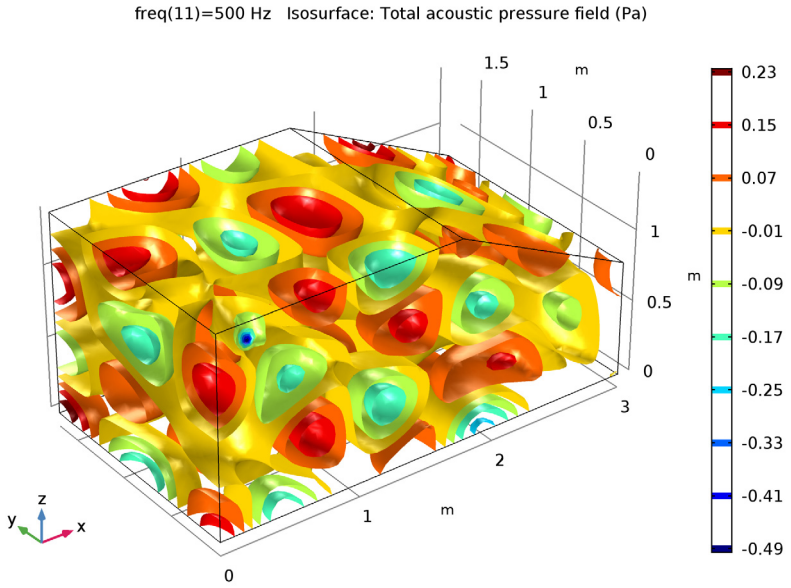


Figure 2: Isosurface plot of the pressure distribution at 500 Hz, for the coarsest one of the 4 meshes used in the model.

A direct comparison of the computational results with those published in Ref. 2 is difficult for two reasons:

- The sampling frequency is too small to resolve some of the resonances at the higher frequencies
- The amplitude for the point source is not given

Nevertheless, it is possible to examine the accuracy of the results by studying how they converge when using successively greater mesh resolutions.

To provide an idea of the accuracy of the various solutions, Figure 3 shows the measured sound pressure level between 490 Hz and 500 Hz, computed at 0.1 Hz intervals for the

listed mesh resolutions. The plot can be reproduced by changing the frequency resolution from 1 Hz to 0.1 Hz when following the Modeling Instructions. As you can see in the plot, all the curves follow each other closely except at the dips and peaks, where the curves for the two coarsest meshes are off somewhat in frequency and substantially in amplitude. The two finest meshes agree very well on the location of the resonances but differ regarding the amplitude. This is because without energy losses in the system, the resonances are singular. A perfect resolution in space and time would result in infinite sound levels in the peaks and absolutely no sound in the dips. In real-life applications, the extremes would be much less pronounced than in the plot because the sound sources are rarely point-like and the walls are usually not perfectly reflecting. In such cases, you can expect a similar solution accuracy at the peaks and the dips as in between them.

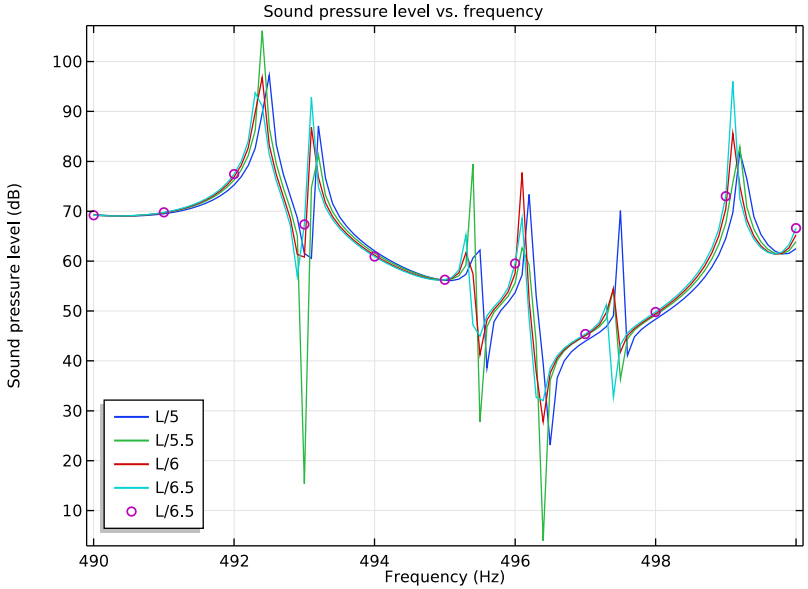


Figure 3: Sound pressure level (dB) at the point of measurement as a function of frequency (Hz) for a few different mesh densities. The legend shows the maximum mesh element size in fractions of the wavelength L.

Notes about Modeling in COMSOL Multiphysics

In the case treated in the step-by-step instructions, you generate the initial mesh by specifying a maximum element size of $\lambda/5$, where $\lambda = c/f$ is the free-space wavelength of the sound waves at 500 Hz. After showing how you to set up, run, and analyze the model with this mesh, the instructions take you through the mesh convergence study. Sometimes,

as a rule of thumb, the mesh is set with a resolution of $\lambda/5$. As it is seen from this model this is not always fine enough to get accurate resonances. When a model has sharp resonances or it has small geometry details, the mesh should be finer. Therefore it is always recommended to do a mesh convergence study when performing simulations, as is done in this model. Moreover, as it is clear from this model having a fine enough frequency resolution is also important as it is clear that the two solutions match well between the resonances.

Note: It is good simulation practice to perform a mesh convergence analysis to be sure that the obtained results have converged.

As a final step in the instructions it is shown how to solve the model using an iterative solver. This approach is seen to be both faster and more memory efficient than using the default direct solver.

Note: When solving large 3D acoustic models it is often advantageous to use an iterative solver (GMRES of FGMRES) with the multigrid preconditioner. More details are found in the *Acoustics Module User's Guide* in the *Modeling with the Pressure Acoustics Branch* section under the *Pressure Acoustics Interface*. See *Solving Large Acoustics Problems Using Iterative Solvers*. In this model (the final instructions in the step-by-step) it is illustrated how you can switch to an iterative solver using one of the predefined iterative solver suggestions.

References

1. The Sound Brick is located in Acoustic Competence Centre, Graz, Austria http://portal.tugraz.at/portal/page/portal/TU_Graz

2. A. Hepberger, H. Priebsch, W. Desmet, B. Van Hal, B. Pluymers, and P. Sas, "Application of the Wave-Based Method for the Steady-state Acoustic Response Prediction of a Car Cavity in the Mid-frequency Range," *Proceedings of the International Conference on Noise and Vibration Engineering*, ISMA2002, Leuven, Belgium, pp. 877–884, 2002.

Application Library path: `Acoustics_Module/Automotive/test_bench_car_interior`

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **Preset Studies>Frequency Domain**.
6. Click **Done**.

GLOBAL DEFINITIONS

Now, either enter the parameters used for the model manually or load them from the file `test_bench_car_interior_parameters.txt`.

Parameters

The values used in Ref. 2 for air density and speed sound are slightly different from those in the default air material available in the COMSOL Multiphysics Material Library. This model uses the values from Ref. 2.

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
rho0	1.2[kg/m^3]	1.2 kg/m³	Air density
c0	343.8[m/s]	343.8 m/s	Speed of sound in air
f0	500[Hz]	500 Hz	Highest frequency used in the model
L	c0/f0	0.6876 m	Shortest wavelength
S	1e-5[m^3/s]	1E-5 m³/s	Flow source strength

You will use L, the wavelength corresponding to the highest frequency, when defining the mesh size.

Import the geometry of the test bench car interior from a file.

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file test_bench_car_interior.mphbin.

5 Click **Import**.

Create two points, one for the source and one for the receiver.

Point 1 (pt1)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Point**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type 0.21.

4 In the **z** text field, type 1.28.

Point 2 (pt2)

1 On the **Geometry** toolbar, click **More Primitives** and choose **Point**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type 1.34.

4 In the **y** text field, type 1.22.

5 In the **z** text field, type 0.8.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Form Union (fin)** and choose **Build Selected**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho0	kg/m³	Basic
Speed of sound	c	c0	m/s	Basic

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Monopole Point Source 1

1 On the **Physics** toolbar, click **Points** and choose **Monopole Point Source**.

2 Select Point 5 only.

Point 5 is the one on the geometry's front wall.

3 In the **Settings** window for **Monopole Point Source**, locate the **Monopole Point Source** section.

4 In the Q_S text field, type `i*S`.

Introducing a phase shift by multiplying the source strength with the imaginary unit gives nonzero pressure plots at the default zero value for the complex pressure variable's phase. You can also here simply enter `S` in the Q_S text field and then enter `pi/2` for the phase ϕ.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

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 `L/5`.

5 Click **Build All**.

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 **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(490,1,500)`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

The first two of the default plots show the pressure and sound pressure level distributions on the walls of the car at 500 Hz. The third plot gives a closer look at the standing waves inside the car in the form of isosurfaces for the pressure (see Figure 2).

Point Graph 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Point Graph**.

3 In the **Settings** window for **Point Graph**, locate the **Selection** section.

4 Select the **Active** toggle button.

5 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

6 Select Point 6 only.

Wireframe rendering makes it easier to select Point 6, which is in the interior of the car.

7 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain> Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

8 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Global>acpr.freq - Frequency**.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
L/5

12 On the **1D Plot Group 4** toolbar, click **Plot**.

This concludes the coarse mesh version of the model. The rest of the instructions show you how to perform a mesh convergence study, by repeating the solution procedure with three consecutively finer meshes and comparing the results. The approach in this model is useful if you want to keep all the results in the model and easily compare them afterwards. However, if you systematically want to process solutions on many different meshes or geometries, a batch run would be more convenient, as it avoids the need to perform any of the settings more than once.

To prepare for your convergence study, start by creating the meshes. Each one should have a successively smaller maximum element size.

COMPONENT 1 (COMP1)

Mesh 2

On the **Mesh** toolbar, click **Add Mesh**.

MESH 2

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Mesh 2** and choose **Free Tetrahedral**.

2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

3 In the **Maximum element size** text field, type `L/5.5`.

COMPONENT 1 (COMP1)

Mesh 3

On the **Mesh** toolbar, click **Add Mesh**.

MESH 3

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Mesh 3** and choose **Free Tetrahedral**.

2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

3 In the **Maximum element size** text field, type `L/6`.

COMPONENT 1 (COMP1)

Mesh 4

On the **Mesh** toolbar, click **Add Mesh**.

MESH 4

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Mesh 4** and choose **Free Tetrahedral**.

2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

3 In the **Maximum element size** text field, type `L/6.5`.

You need a separate study for each mesh.

ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, click to expand the **Mesh selection** section.

2 Locate the **Mesh Selection** section. In the table, enter the following settings:

Geometry	Mesh
Geometry 1	Mesh 2

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

3 Click **Add Study** in the window toolbar.

STUDY 3

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, click to expand the **Mesh selection** section.

2 Locate the **Mesh Selection** section. In the table, enter the following settings:

Geometry	Mesh
Geometry 1	Mesh 3

ADD STUDY

1 Go to the **Add Study** window.

2 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Frequency Domain**.

3 Click **Add Study** in the window toolbar.

4 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Mesh Selection** section.

2 In the table, enter the following settings:

Geometry	Mesh
Geometry 1	Mesh 4

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 2** 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 `range(490,1,500)`.

STUDY 3

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 3** 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 `range(490,1,500)`.

STUDY 4

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 4** 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 `range(490,1,500)`.

   Having set up a Frequency Domain study step for each study, you are now ready to start the solution process. The first solution already exists, so start by solving the second one.

STUDY 2

On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr) 1

When the solver is done, three more plots will appear, this time for the second one of your meshes. Proceed to solve for the next mesh.

STUDY 3

Click **Compute**.

RESULTS

Acoustic Pressure, Isosurfaces (acpr) 1

Finally, solve for the finest mesh case. This solution takes a little longer time to compute than the others. Go to the end of the instructions to learn how to set up an iterative solver that is both faster and more memory efficient than the default direct solver.

STUDY 4

Click **Compute**.

RESULTS

Point Graph 2

1. In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Point Graph**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
4. Locate the **Selection** section. Select the **Active** toggle button.

5 Select Point 6 only.

6 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain> Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

7 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **acpr.freq - Frequency**.

8 Locate the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
L/5.5

Point Graph 3

1 Right-click **1D Plot Group 4** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 3/Solution 3 (sol3)**.

4 Locate the **Selection** section. Select the **Active** toggle button.

5 Select Point 6 only.

6 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain> Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

7 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **acpr.freq - Frequency**.

8 Locate the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
L/6

Point Graph 4

1 Right-click **1D Plot Group 4** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Study 4/Solution 4 (sol4)**.

4 Locate the **Selection** section. Select the **Active** toggle button.

5 Select Point 6 only.

6 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain> Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

7 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **acpr.freq - Frequency**.

8 Locate the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
L/6.5

1D Plot Group 4

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `SPL Response` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Sound pressure level vs. frequency`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

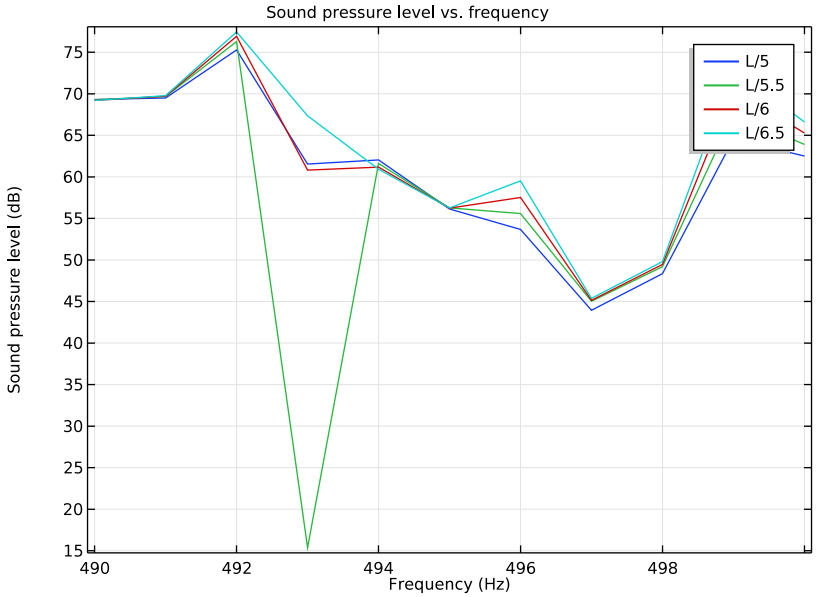
6 In the associated text field, type `Frequency (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Sound pressure level (dB)`.

9 On the **SPL Response** toolbar, click **Plot**.

The plot should look like the figure below. A version of the plot with a higher frequency resolution of 0.1 Hz is depicted in Figure 3.



The next instructions are optional. They show how to use an iterative solver using a multigrid approach. This solver configuration is both faster (by a factor 3, depending on the hardware) and more memory efficient.

STUDY 4

In the **Model Builder** window, collapse the **Study 4>Solver Configurations** node.

ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 5

Step 1: Frequency Domain

1. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
2. In the **Frequencies** text field, type `range(490,1,500)`.
3. In the **Model Builder** window, click **Study 5**.
4. In the **Settings** window for **Study**, type `Study 5 - Iterative Solver` in the **Label** text field.
5. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 5 - ITERATIVE SOLVER

Generate and show the default solver sequence and then expand the nodes. You can then enable one of the suggested iterative solvers that are predefined. Use the one based on the FGMRES solver.

Solution 5 (sol5)

1. On the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 5 (sol5)** node.
3. In the **Model Builder** window, expand the **Study 5 - Iterative Solver> Solver Configurations>Solution 5 (sol5)>Stationary Solver 1** node.
4. Right-click **Suggested Iterative Solver (FGMRES with GMG) (acpr)** and choose **Enable**.
5. On the **Study** toolbar, click **Compute**.

RESULTS

Point Graph 4

In the **Model Builder** window, expand the **SPL Response** node.

Point Graph 5

1. Right-click **Point Graph 4** and choose **Duplicate**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. From the **Data set** list, choose **Study 5 - Iterative Solver/Solution 5 (sol5)**.
4. Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
5. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
6. From the **Positioning** list, choose **In data points**.

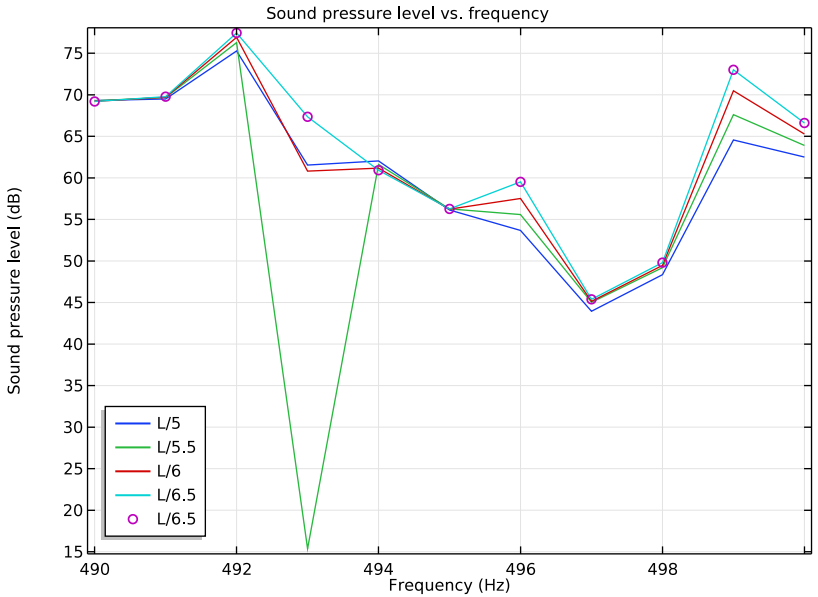
SPL Response

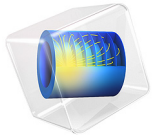
1 In the **Model Builder** window, under **Results** click **SPL Response**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Legend** section.

3 From the **Position** list, choose **Lower left**.

The plot should look like the figure below. It shows that the iterative solutions (circles on the plot) exactly matches the direct solver.





Piezoelectric Tonpilz Transducer

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

The tonpilz (sound mushroom) transducer is a transducer for relatively low frequency, high power sound emission (Ref. 1 and Ref. 2). It is one of the popular acoustic transducer designs that are used for underwater SONAR applications. The tonpilz transducer modeled in this tutorial consists of piezoceramic rings stacked between an aluminum head mass and a steel tail mass connected by a steel bolt. This central bolt could be prestressed to control the transducer response. The tail and head mass are used to lower the resonance frequency of the device to the desired level.

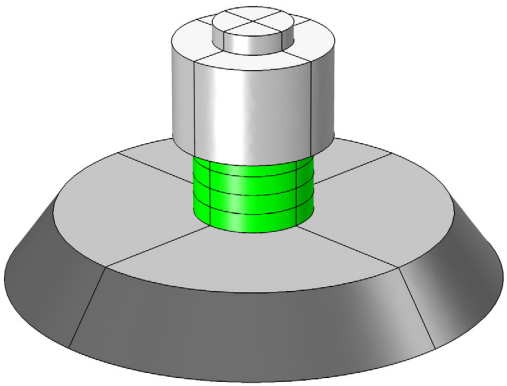


Figure 1: A tonpilz transducer. The aluminum head mass is shown in dark gray, the central steel bolt and steel tail mass are shown in light gray and the piezostack actuator with four disks of PZT-4 is shown in green.

Model Definition

In this model, the frequency response of the transducer shown in Figure 1 is studied. In this version of the tutorial, the effect of prestress in the bolt is not considered. The outer curved surface of the steel tail mass is assumed to be fixed. Each of the piezo disks are excited with a 1 V RMS electrical signal. The model determines the deformation in the device, the radiated pressure field and sound pressure level, as well as the spatial far-field sensitivity, the transmitting voltage response (TVR) curve of the transducer, and the directivity index (DI) of the sound beam within the frequency range of 1 kHz to 40 kHz.

The parameters used in this model are shown in Table 1.

TABLE 1: LIST OF MODELING PARAMETERS

NAME	EXPRESSION	DESCRIPTION
Rwater	40[mm]	Water domain radius
Rpml	10[mm]	PML layer thickness
a	25[mm]	Piston head radius
Zeval	-10[m]	Directivity evaluation distance
Vrms	1[V]	RMS drive voltage
V0	sqrt(2)*Vrms	Zero-to-peak drive voltage
f0min	1[kHz]	Minimum operating frequency
f0max	40[kHz]	Maximum operating frequency
f0step	1[kHz]	Frequency step

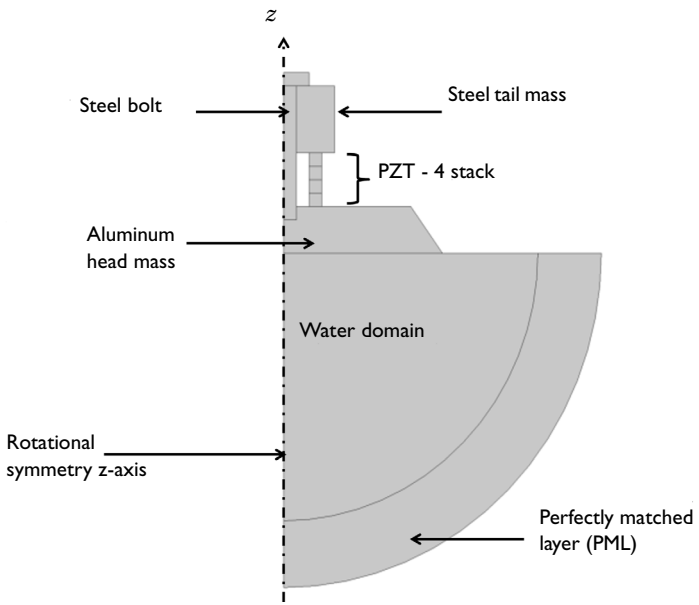


Figure 2: An axisymmetric section of the model geometry.

Figure 2 shows an axisymmetric view of the actual 3D model. The head mass is exposed to an unbounded region of water. A Perfectly Matched Layer (PML) is used to model the absorption of sound waves as they propagate far away from the sound source. Note that

although the modeling geometry has a rotational symmetry, using a full 3D geometry allows us to capture any vibration mode of the transducer and acoustic mode of the fluid region that do not possess such a symmetry in the solution.

PHYSICS IMPLEMENTATION

The Acoustic-Piezoelectric Interaction, Frequency Domain interface available in the Acoustics Module is used to simulate the multiphysics interactions. This predefined multiphysics interface includes the necessary fundamental physics which are Pressure Acoustics, Solid Mechanics, and Electrostatics. The Pressure Acoustics interface is used to solve the wave equation in the water domain. The Solid Mechanics interface is solved on all structural materials including the PZT-4 disks. The Electrostatics interface is solved on the PZT-4 disks. The multiphysics couplings necessary to model this system are available as predefined nodes under the Multiphysics branch. These couplings are:

Acoustic-Structure Boundary: This node is active on the boundaries that are at the interface of the water domain and transducer head mass. On these boundaries a bidirectional coupling is automatically set up. The fluid pressure evaluated by the Pressure Acoustics interface is applied as a mechanical load in the Solid Mechanics interface. Furthermore, the normal component of the structural acceleration is used as a sound source.

Piezoelectric Effect: This node is active on the PZT-4 domains only and couple the Solid Mechanics and Electrostatics equations solved in these domains via the linear constitutive equations that model the piezoelectric effect by coupling stresses and strains with electric field and electric displacement.

MATERIAL ORIENTATION

The piezoelectric disks are stacked in a way such that alternate disks are poled along opposite directions as shown in Figure 3. This allows us to use a single electrical terminal at the interface of each pair of disks and obtain the piezoelectric actuation effect in each of

the disks along the same direction. Having the piezoelectric strain in-phase in all the disks maximizes the actuation.

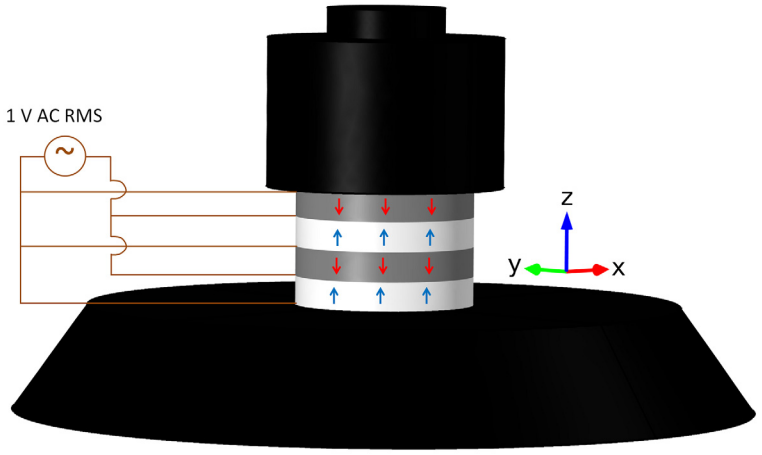


Figure 3: Schematic representation of the poling direction and electrical connections in the piezo disks. The blue arrows in the light gray disks indicate a +Z polarization. The red arrows in the dark gray disks indicate a -Z polarization.

In this model, the PZT-4 disks actuate in the d_{33}-mode. Hence two of the disks are poled along the +Z-direction while the other two are poled along the -Z-direction. The default definition of the piezoelectric material properties in COMSOL's Global Coordinate System automatically creates a +Z polarization. In order to create a -Z polarization, a user-defined Rotated Coordinate System is used. In this coordinate system, the Euler angles are set to $\alpha = 0$, $\beta = \pi$, and $\gamma = 0$. Note that COMSOL uses the Z-X-Z convention for Euler angles.

FAR FIELD CALCULATION

A far-field calculation is set up on the interface boundaries between the inner water domain and the PML domains. The far-field integral type is set to *full integral* which allows computation of both amplitude and phase of the acoustic pressure and sound pressure level (SPL) at any point in space outside the computational domain. These quantities are later used in postprocessing to plot the on-axis pressure variation outside the computational domain as well as to visualize the beam pattern in 2D and 3D polar plots.

MESHING CONSIDERATIONS

In order to accurately resolve the pressure waves within the inner water domain, the maximum mesh element size is specified as $1/5^{\text{th}}$ of the smallest wavelength of interest.

The smallest wavelength is the speed of sound in water (1500 m/s) divided by the largest frequency used in the frequency sweep. The PML is meshed using the Swept feature to create five layers of structured mesh. Additionally a single layer of Boundary Layer Mesh is created within the inner water domain adjacent to the far-field boundaries. The thickness of this layer is set to $1/100^{\text{th}}$ of the smallest wavelength of interest. The single boundary layer creates a smooth transition between the inner free tetrahedral mesh and the outer structured prism mesh elements thereby yielding a more precise far-field calculation.

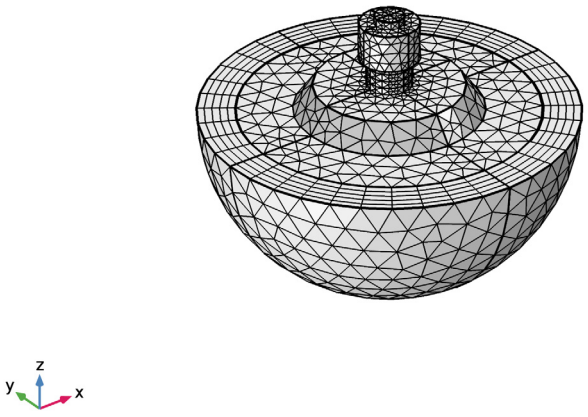


Figure 4: The modeling geometry with the computational mesh.

COMPUTING TRANSDUCER CHARACTERISTICS

The frequency response of the following important transducer characteristics are investigated in this tutorial.

Specific Acoustic Impedance

The specific acoustic impedance Z_{aco} is computed as the ratio of the impedance of the head mass surface exposed to water to the characteristic impedance Z_0 of water as shown in Equation 1. The value of Z_0 is computed from the product of density and speed of sound in water at a temperature of 293.15 K.

$$Z_{\text{aco}} = \frac{\left(\int_A p\,da\right) / \left(\int_A v_n\,da\right)}{Z_0} \tag{1}$$

The impedance of the head mass surface is computed from the ratio of the area integral of the acoustic pressure p to the area integral of the normal component of the structural velocity v_n of the surface. In this case the normal velocity is the same as the Z-component of the velocity v_z which in frequency domain can be represented as the product of the Z-component of the structural displacement w, the variable j which is the imaginary number square root of -1 and the angular frequency of vibration ω. The area integral is computed by introducing an integration coupling operator that is assigned to the surface of interest. To avoid division by zero, it is advised to add the built-in constant **eps**, which denotes the machine precision limit, to the variable w as shown in Table 2 in the definition of the variable Z_{aco}.

Transmitting Voltage Response (TVR)

The TVR represents the sensitivity of the transducer measured at a distance of 1 m and driven at 1 V RMS. This definition can be mathematically expressed using Equation 2.

$$TVR = 20\log\frac{p_{\text{RMS}} / V_{\text{RMS}}}{1\mu Pa\,V^{-1}} \tag{2}$$

The RMS pressure at 1 m from the transducer head mass surface can be computed from Equation 3.

$$p_{\text{RMS}} = \sqrt{\frac{1}{2} p p^*} \tag{3}$$

In Equation 3, the pressure p at 1 m ahead of the transducer is obtained using the far-field pressure variable pfar by using an expression pfar(0,0,-1). Note that the computational domain is much smaller than 1 m but by performing the far-field calculation you are able to compute the pressure and phase at any point outside the domain. The variable p^* is the complex conjugate of the pressure p and can be expressed in COMSOL using the expression `conj(pfar(0,0,-1))`.

Directivity Index (DI)

The directivity index (DI) gives a measure of how directional the transmitted acoustic beam is (Ref. 3). It is a measure of the ratio of the intensity I_{front} in front of the transducer at a certain distance in the far-field to the intensity I_{ave} transmitted by an omni-directional

source of the same strength. The DI is evaluated at a distance of 10 m in front of the transducer. You can alter the value of the parameter Z_{eval} to compute the DI at a different location. The DI can be computed from Equation 4.

$$DI = 10\log\frac{I_{\text{front}}}{I_{\text{ave}}} \tag{4}$$

The quantity I_{front} can be computed from the ratio of the RMS pressure at the desired distance from the transducer to the characteristic impedance Z_0. The quantity I_{ave} can be computed from the ratio of total radiated power P_{tot}, to the surface area of a sphere having the same radius as the distance from the transducer at which the DI is computed.

The user-defined variables used to compute the transducer characteristics are shown in Table 2.

TABLE 2: LIST OF VARIABLES

NAME	EXPRESSION	DESCRIPTION
rho0	intop3(acpr.rho)	Density of water at room temperature
c0	intop3(acpr.c)	Speed of sound in water at room temperature
Zaco	intop2(p)/intop2(acpr.iomega*(w+eps))/(rho0*c0)	Specific acoustic impedance
pfar_1	pfar(0,0,-1)	Far field pressure at 1 m
prms	sqrt(0.5*pfar_1*conj(pfar_1))[Pa]	RMS pressure at 1 m
TVR	20*log10(prms/Vrms/1[uPa/V])	Transmitting Voltage Response (TVR)
pfar_Zeval	pfar(0,0,Zeval[1/m])	Far field pressure at Zeval
Ifront	0.5*pfar_Zeval*conj(pfar_Zeval)[Pa^2]/(rho0*c0)	On-axis intensity at Zeval
Ptot	intop1(down(acpr.lx)*acpr.nx+down(acpr.ly)*acpr.ny+down(acpr.lz)*acpr.nz)	Total radiated power
Iave	Ptot/(4*pi*Zeval^2)	Average intensity of monopole source at Zeval
Di	Ifront/Iave	Intensity directivity
DI	10*log10(Di)	Directivity index of the tonpilz transducer
k0	2*pi*freq/c0	Wave number

TABLE 2: LIST OF VARIABLES

NAME	EXPRESSION	DESCRIPTION
DI_fl_pist	10*log10((k0*a)^2/(1-2*besselj(1,2*k0*a)/(2*k0*a)))	Directivity index of flanged piston
SPL_Zeval	intop3(subst(acpr.ffc1.Lp_pfar,x,0,y,0,z,Zeval))	SPL at Zeval
SPL_rel	acpr.ffc1.Lp_pfar-SPL_Zeval	SPL relative to 0 dB at Zeval

Results and Discussion

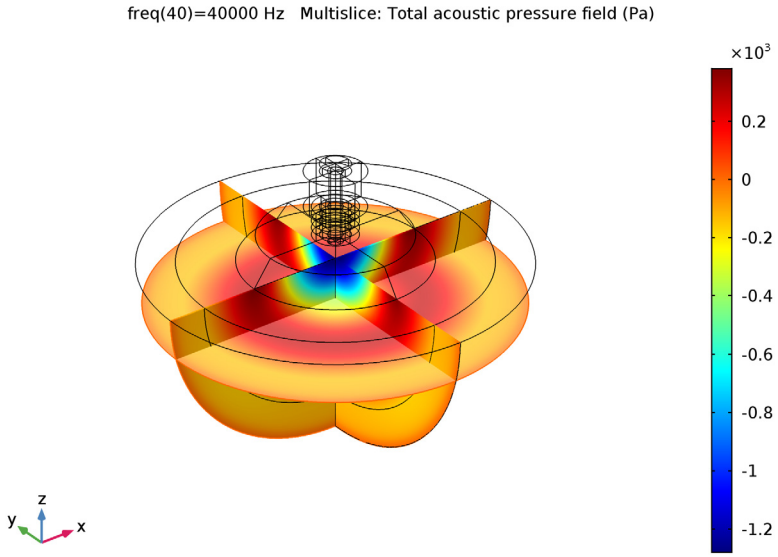


Figure 5: A multislice plot showing the total acoustic pressure variation in the water domain and PML at 40 kHz.

Figure 5 shows the total acoustic pressure in the water domain for 40 kHz excitation. The inner water domain in the model captures roughly half a wavelength at this frequency. As expected, the outer surface of the PML layer shows zero pressure which confirms that the PML layer effectively absorbs the outgoing waves. Setting the PML stretching type to rational allows us to effectively use the PML over a large range of wavelengths and angles of incidence of the pressure waves as encountered in this model.

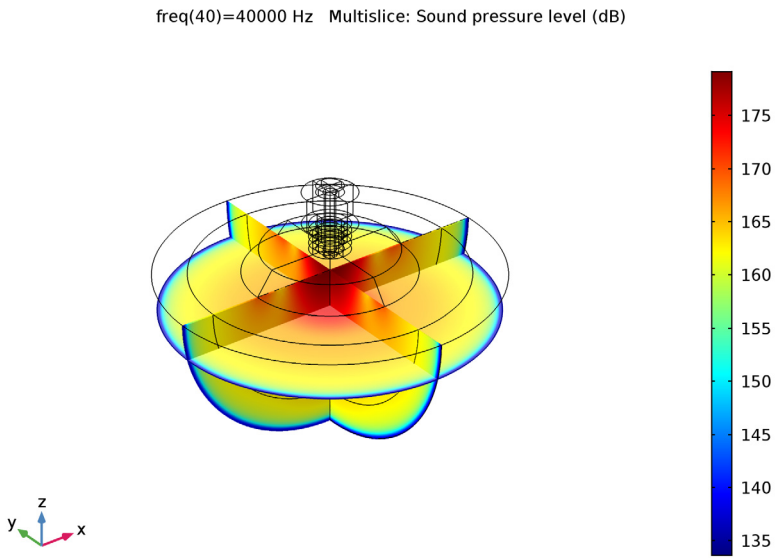


Figure 6: A multislice plot showing the sound pressure level (SPL) in the water domain and PML at 40 kHz.

Figure 6 shows the sound pressure level (SPL) in the water domain for 40 kHz excitation. Note the 45 dB difference between the SPL near the transducer head mass and the outer surface of the PML layer. This once again confirms the effectiveness of the damping induced by the PML.

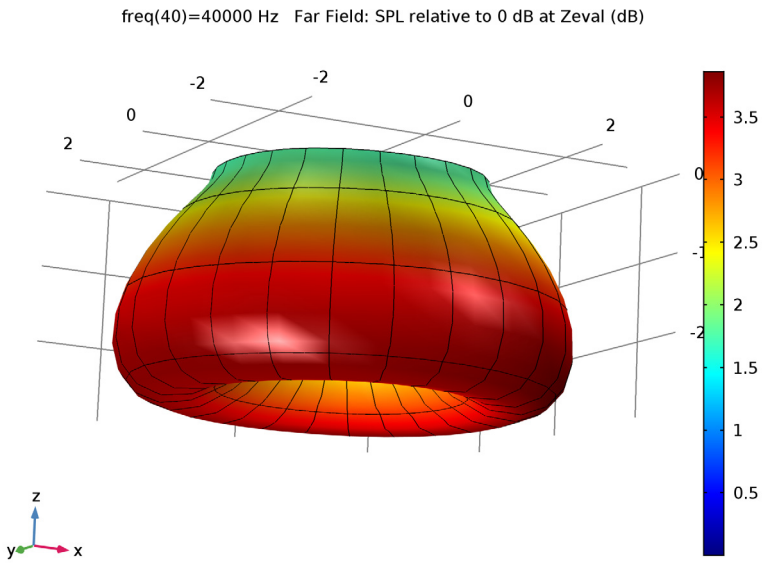


Figure 7: A 3D far-field plot showing the sound pressure level (SPL) for 40 kHz excitation at a distance of 10 m from the head mass. The SPL is computed relative to 0 dB at an on-axis distance of 10 m from the head mass.

Figure 7 shows a 3D polar plot of the far-field SPL computed at a radial distance of 10 m from the transducer head mass for 40 kHz excitation. The SPL shown here is computed relative to a value of 0 dB at an on-axis distance of 10 m directly ahead of the transducer head mass. This baseline value is computed by the user-defined variable `SPL_Zeval` and the SPL relative to this quantity is computed by the variable `SPL_rel`. Note that you can control the baseline value by changing the value of the parameter `Zeval`. You could also create similar plot for any of the other frequencies solved for and also at other distances outside the computational domain. The later can be controlled by manually specifying the radius of the sphere as the absolute value of the parameter `Zeval` in the Evaluation section of the Far-Field plot settings.

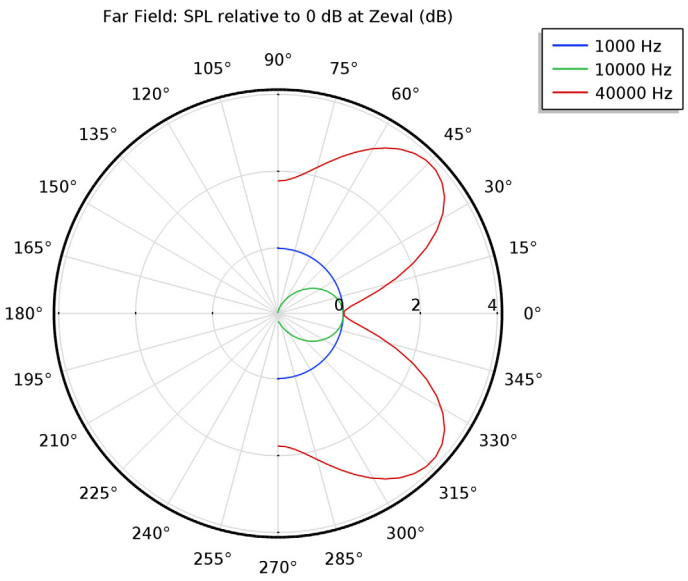


Figure 8: A polar beam sensitivity plot showing the sound pressure level (SPL) for 1, 10 and 40 kHz excitation at a distance of 10 m from the head mass. The SPL is computed relative to 0 dB at an on-axis distance of 10 m from the head mass.

Figure 8 shows a polar beam sensitivity plot which is also known as the beam pattern. Here the relative far-field SPL is computed at a radial distance of 10 m from the transducer head mass for 1, 10, and 40 kHz excitation. The polar plot here shows the radiation pattern in the XZ-plane. The right half of the circle from 270 ^{o} through 0 ^{o} up to 90 ^{o} correspond to the -Z hemisphere. The +Y direction is directed into the plane. The beam pattern shows that at a relatively lower frequency of 1 kHz, the sound emitted is fairly omni-directional. This is because at low frequencies the transducer works in the piston mode.

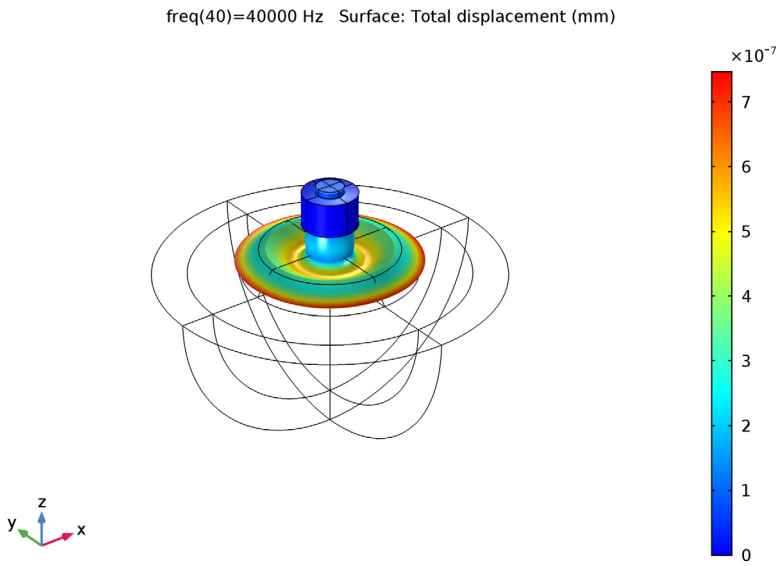


Figure 9: Total structural displacement of the transducer at 40 kHz. An exaggerated deformation has been used for better visualization.

Figure 9 shows the total structural displacement of the tonpilz transducer at 40 kHz excitation. At this frequency, the head mass vibrates in a mode whose shape is somewhat toroidal. This produces the lobes in the sound radiation pattern as can be seen in Figure 7 and Figure 8. If you observe the vibration mode for relatively lower frequencies, say 1 kHz, observe that the head mass vibrates mainly along its axis similar to a flanged piston.

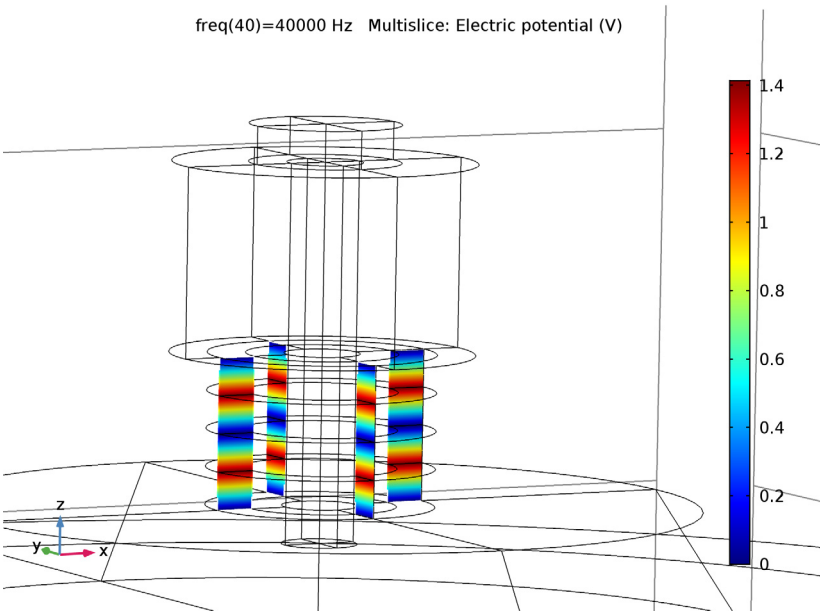


Figure 10: A multislice plot of the electric potential distribution within the four PZT-4 disks.

Figure 10 zooms on the piezo stacks. It shows the electric potential distribution through the thickness of the PZT-4 disks. The color bands indicate a successful implementation of the idea described in Figure 3.

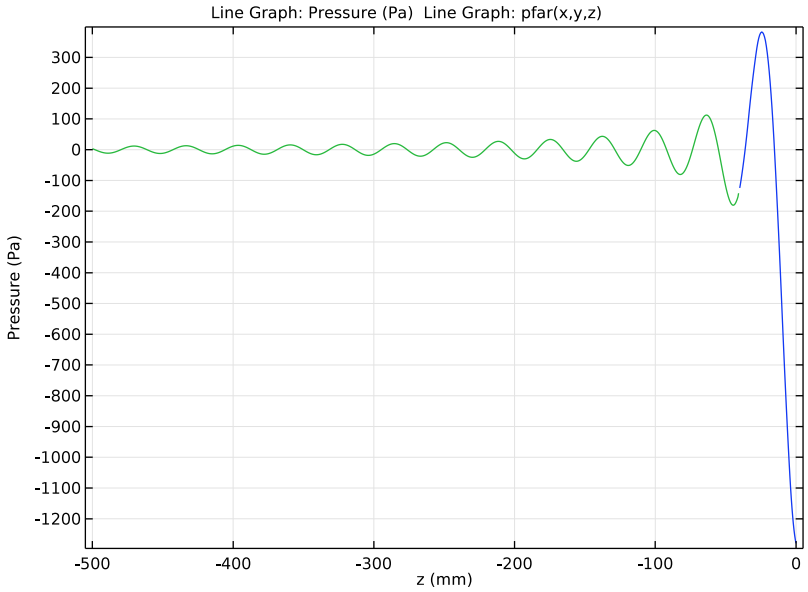


Figure 11: A line plot of acoustic pressure variation directly ahead of the transducer head mass up to an on-axis distance of 500 mm.

Figure 11 shows a plot of acoustic pressure vs. z-coordinate along the axis of the transducer for an excitation frequency of 40 kHz. The blue curve shows the pressure variable p that is solved for vs. the z-coordinate of the geometric edge that passes through the radius of the inner water domain vertically downwards from the head mass along the transducer axis. The green curve shows how the pressure outside the computational domain can be computed using the far-field variable `pfar` by using an expression `pfar(x,y,z)`. The variable is evaluated using a parametrized curve data set to evaluate the variable outside the computational mesh.

This approach shows how to use a relatively small computational domain for efficient modeling and still use the results from the far-field calculation in postprocessing to visualize both the magnitude and phase of the pressure outside the computational domain.

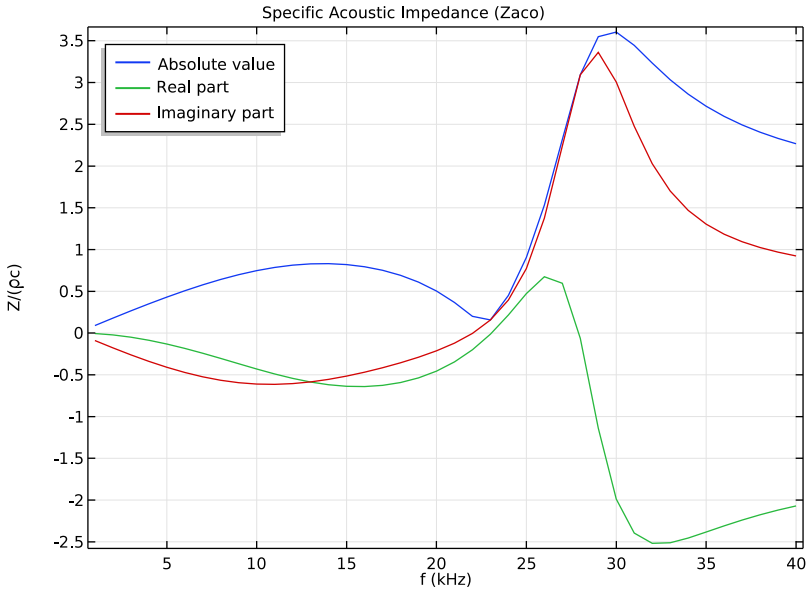


Figure 12: Frequency response plot of the absolute value, real and imaginary components of the specific acoustic impedance at the interface between the head mass and water.

Figure 12 shows the frequency response of the specific acoustic impedance of the head mass surface that is exposed to water. A resonance and anti-resonance is observed within the frequency range of 20 kHz and 35 kHz.

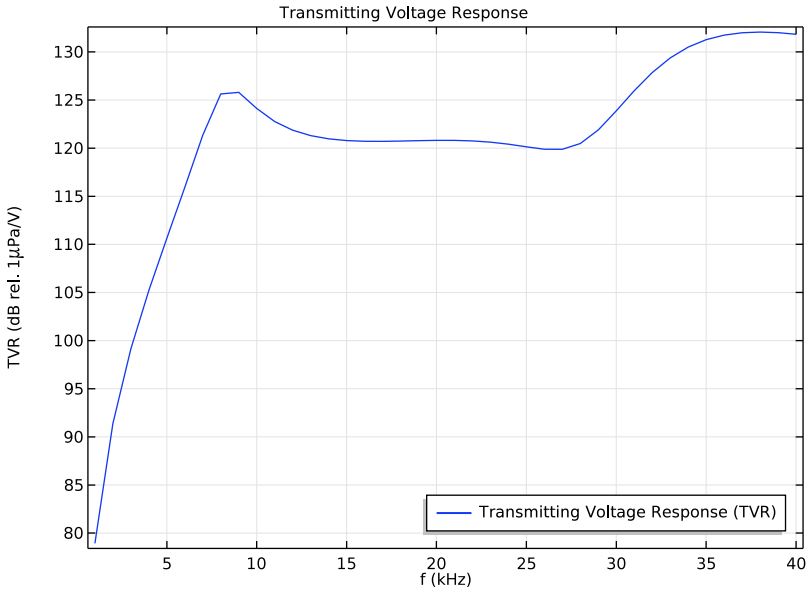


Figure 13: Transmitting Voltage Response (TVR) as a function of frequency obtained at an on-axis distance of 1 m ahead of the head mass and computed relative to 1 µPa/V.

Figure 13 shows the variation in the TVR of the transducer as a function of operating frequency. The fairly flat region between 15 kHz and 25 kHz can be particularly useful for sensing applications.

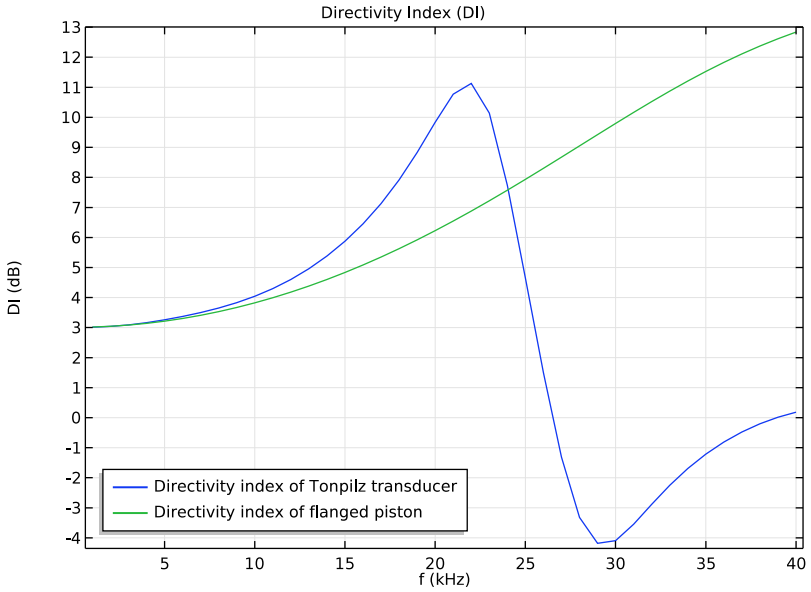


Figure 14: Frequency response of the Directivity Index (DI) computed at an on-axis distance of 10 m from the head mass. The DI of the tonpilz transducer is compared to that of a flanged piston.

Figure 14 shows the Directivity Index (DI) of the tonpilz transducer (blue curve) and compares it with the DI of a flanged piston (green curve). The latter can be computed from analytical expression as shown in Table 2. It is defined by the variable `DI_fl_pist`. Note that when the tonpilz transducer operates like a piston at lower frequencies, its DI becomes very similar to that of a flanged piston. Another feature worth noting is that within the range of 15 kHz and 25 kHz, the DI of the tonpilz transducer changes from nearly 0 dB to 11 dB while its TVR remains nearly constant. This can make the transducer quite versatile within this operating range.

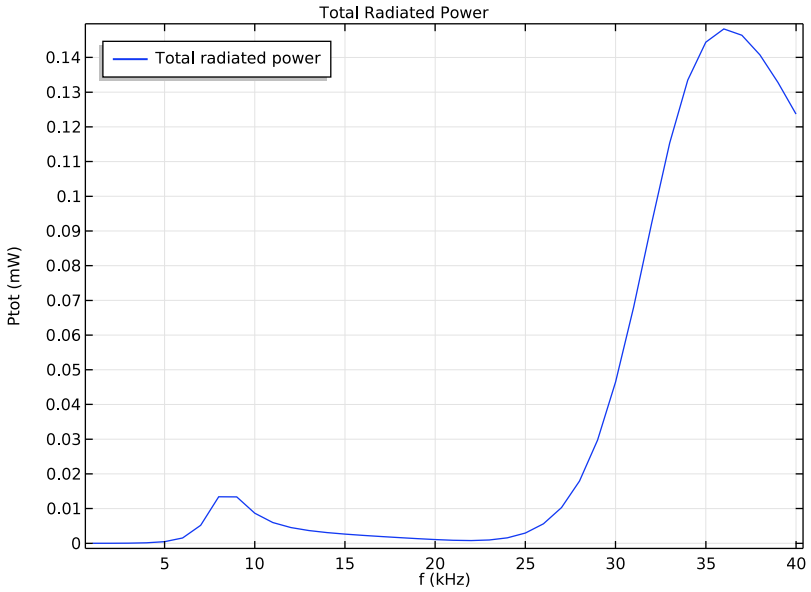


Figure 15: Total radiated power from the tonpilz transducer within the operating frequency range of 1 kHz to 40 kHz.

Figure 15 shows the total radiated power as a function of the operating frequency of the tonpilz transducer. Note that the acoustic radiated power should be always positive. In order to correctly compute the radiated power at the low frequency range you need to make some changes to the default settings of the Perfectly Matched Layer (PML).

The default settings produce negative radiated power at low frequencies. This happens because at low frequencies the evanescent waves extend into the PML layer. The interaction between the scaled coordinate system in the PML and these waves may create an erroneous energy contribution in the model (can be either positive or negative).

A good way to investigate the performance of the PML is to make a sensitivity analysis on some parameter (for example, the total radiated acoustic power) with respect to changes in the PML parameters. In this model such a rigorous sensitivity analysis is not performed. In general, increasing the curvature factor effectively shifts the resolving power of the PML towards the physical domain, which is necessary in this case since the evanescent components decay in only a fraction of a wavelength. However, if you increase it too much, you may lose resolution in the other end, that is, of the free space wavelength. Assuming that the PMLs work properly at high frequencies for a curvature parameter of 1, you can

in principle perform a convergence study by increasing the value of the curvature parameter until the low-frequency (for 1 kHz) result converges while making sure that the high-frequency (for 40 kHz) result is not affected. It turns out that a value of 5 for the curvature parameter yields good results in this model. A scaling factor of 0.5 further improves the results, but only by a small amount. Decreasing the scaling factor corresponds to compressing the PML layer (shortening it), which in turn effectively increases the mesh resolution within the PML region.

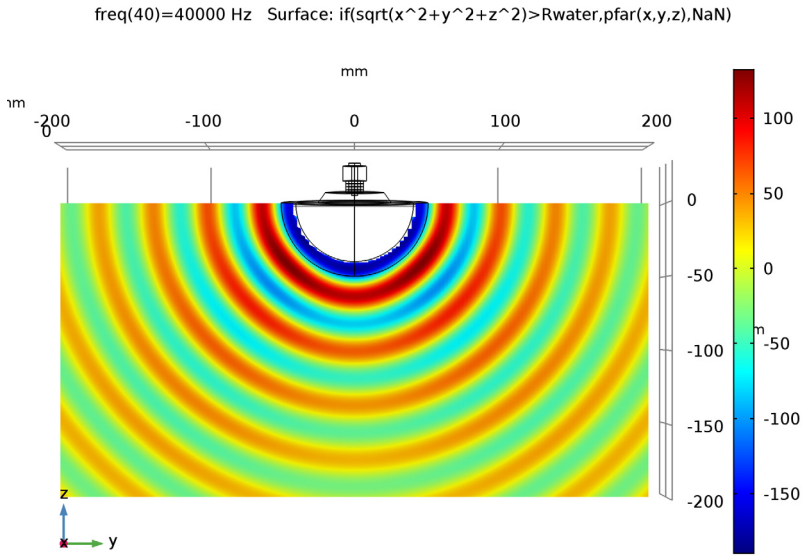


Figure 16: Pressure evaluated outside the computational mesh using the far-field operator pfar(x,y,z).

As discussed above, the pressure (including phase) is evaluated outside the computational domain using the far-field computation feature. The feature name can be a bit misleading since this feature actually evaluates the pressure at any distance outside the far-field boundary (as already seen in Figure 11). The pressure outside the computational mesh is depicted in Figure 16 using the Grid 3D data set. The sound pressure level is depicted in Figure 17.

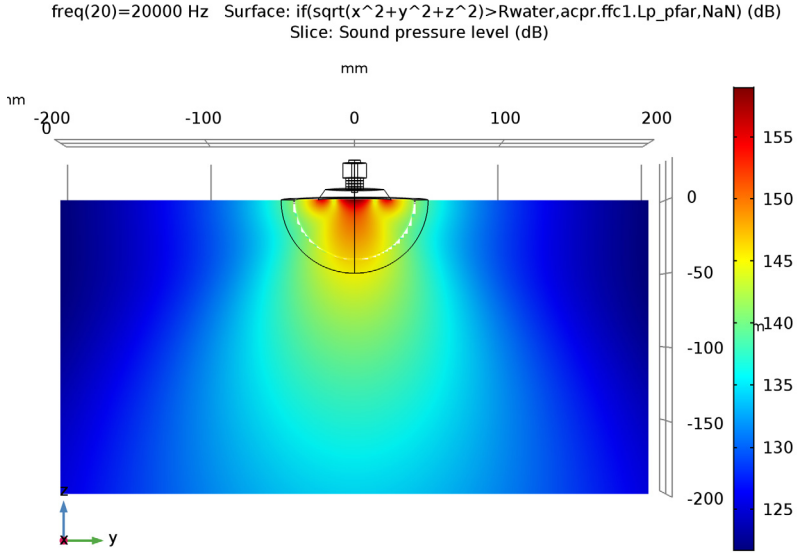


Figure 17: Sound pressure level evaluated outside the computational domain using the far-field feature as well as the sound pressure level inside the domain.

References

1. C. H. Sherman and J. L. Butler, *Transducers and Arrays for Underwater Sound*, Springer, New York, 2007.

2. M. Lasky, "Review of Undersea Acoustics to 1950," *J. Acoust. Soc. Am.*, vol. 61, pp. 283–297, 1976.

3. D. T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.

Application Library path: `Acoustics_Module/Piezoelectric_Devices/tonpilz_transducer`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Piezoelectric Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `tonpilz_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**.

Build a plane section of the geometry before revolving it.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

Start with the solid domain.

4 Click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 4.

4 In the **Height** text field, type 2.

5 Locate the **Position** section. In the **yw** text field, type 25.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 2.

4 In the **Height** text field, type 20.

5 Locate the **Position** section. In the **yw** text field, type 5.

Rectangle 3 (r3)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 6.

4 In the **Height** text field, type 10.

5 Locate the **Position** section. In the **xw** text field, type 2.

6 In the **yw** text field, type 15.

Rectangle 4 (r4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 2.

4 In the **Height** text field, type 8.

5 Locate the **Position** section. In the **xw** text field, type 4.

6 In the **yw** text field, type 7.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	2

Layer name	Thickness (mm)
Layer 2	2
Layer 3	2

Polygon 1 (pol1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the **xw** text field, type `0 0 2 2 20 a`.

4 In the **yw** text field, type `0 5 5 7 7 0`.

Then build the water domain.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Rwater+Rpml`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	Rpml

7 On the **Work Plane** toolbar, click **Build All**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

9 In the **Model Builder** window, click **Geometry 1**.

Revolve 1 (rev1)

1 On the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `90`.

5 Right-click **Revolve 1 (rev1)** and choose **Build Selected**.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **rev1** only.

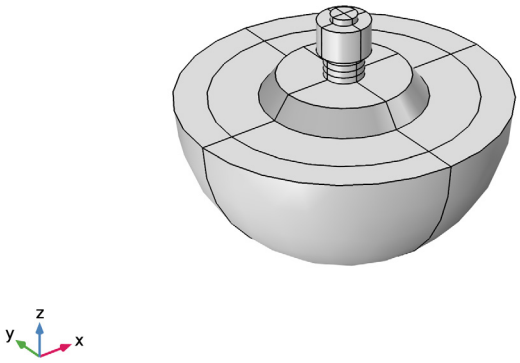
3 In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.

4 In the **Rotation** text field, type `range(90,90,270)`.

5 Locate the **Input** section. Select the **Keep input objects** check box.

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.



8 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

DEFINITIONS

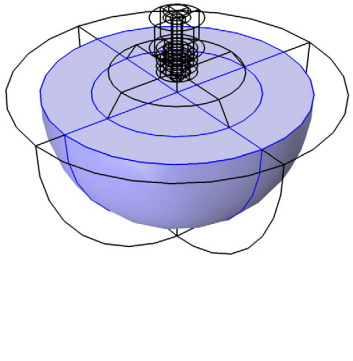
Create domain selections that will be used in the modeling.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Water domain - Inner` in the **Label** text field.

3 Select Domains 3, 4, 22, and 32 only.

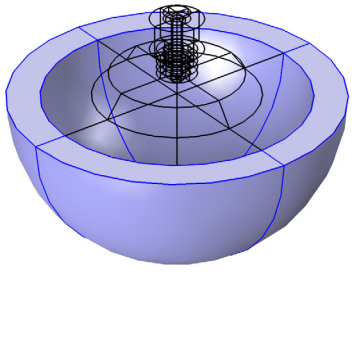


Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Water domain - PML` in the **Label** text field.

3 Select Domains 1, 2, 21, and 31 only.

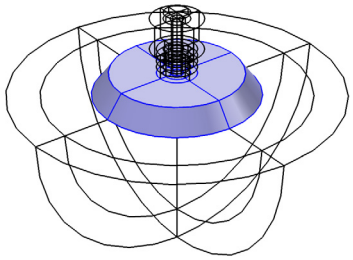


Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Aluminum` in the **Label** text field.

3 Select Domains 5, 6, 23, and 33 only.

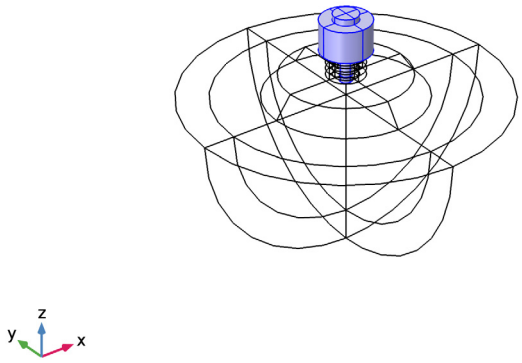


Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Steel` in the **Label** text field.

3 Select Domains 7, 8, 17–20, 24, 29, 30, and 34–36 only.

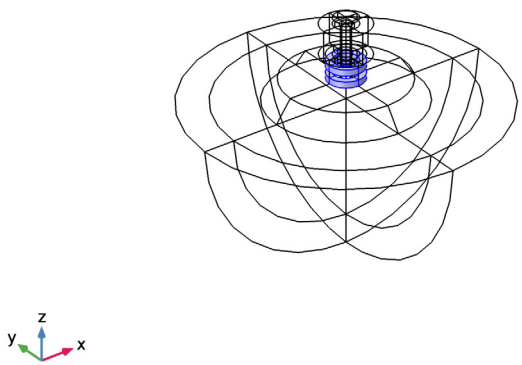


Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `+Z poled Piezo` in the **Label** text field.

3 Select Domains 9, 10, 13, 14, 25, 27, 37, and 39 only.

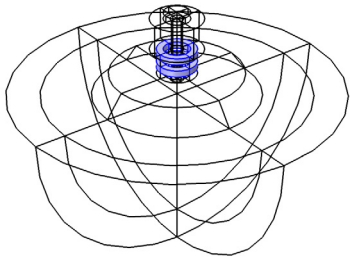


Explicit 6

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `-Z poled Piezo` in the **Label** text field.

3 Select Domains 11, 12, 15, 16, 26, 28, 38, and 40 only.



Union 1

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Water domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Water domain - Inner** and **Water domain - PML**.

5 Click **OK**.

Union 2

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Piezo domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **+Z poled Piezo** and **-Z poled Piezo**.

5 Click **OK**.

Complement 1

1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Solid domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Water domains** in the **Selections to invert** list.

5 Click **OK**.

Complement 2

1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Non-PML domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Water domain - PML** in the **Selections to invert** list.

5 Click **OK**.

Create boundary selections that will be used further in the modeling.

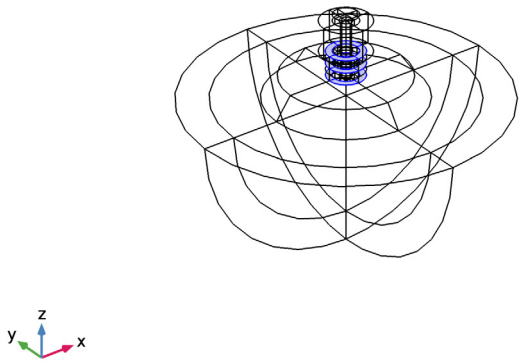
Explicit 7

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Ground boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 28, 29, 38, 39, 45, 46, 93, 99, 103, 145, 151, and 155 only.



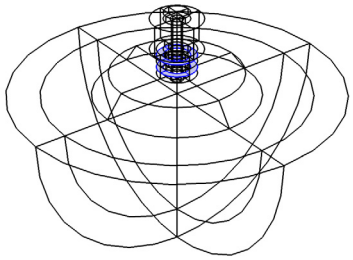
Explicit 8

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Voltage boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 33, 34, 43, 44, 96, 102, 148, and 154 only.



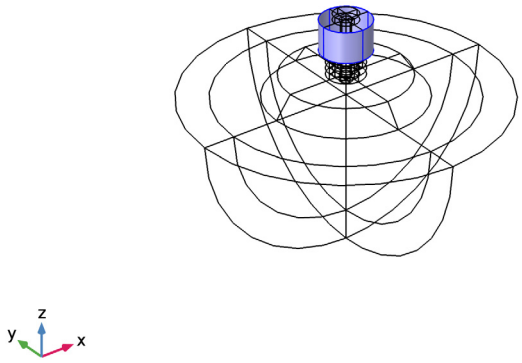
Explicit 9

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Fixed boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 19, 20, 88, and 164 only.



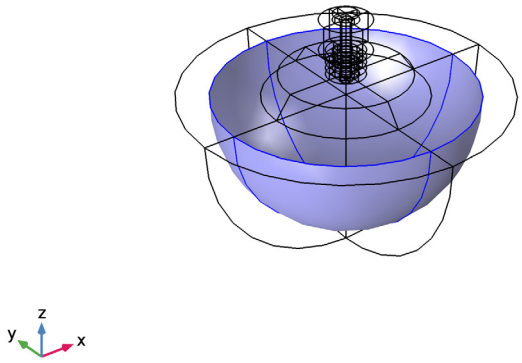
Explicit 10

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Far Field boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 9, 10, 82, and 125 only.



Define an integration operator on the far-field boundary.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Far Field boundaries**.

Define an integration operator on the acoustic-structure interface.

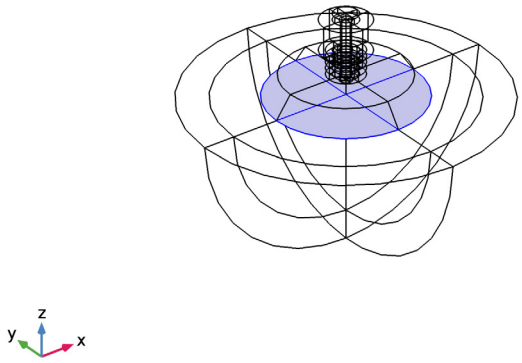
Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 14, 15, 85, and 128 only.



Define an integration operator to retrieve values at the summit point of the water domain.

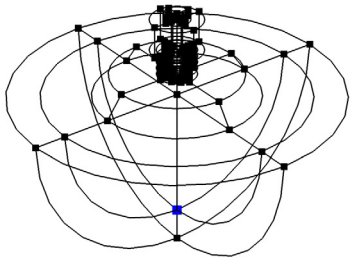
Integration 3 (intop3)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 46 only.



Define a rotated system that will be used for the poling of the -Z poled piezoelectric disks.

Rotated System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for **Rotated System**, locate the **Settings** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the β text field, type `pi`.

Insert a Perfectly Matched Layer to model the absorption of acoustic wave far away from the source.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Water domain - PML**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

5 Locate the **Scaling** section. From the **Coordinate stretching type** list, choose **Rational**.

6 In the **PML scaling factor** text field, type `0.5`.

7 In the **PML scaling curvature parameter** text field, type `5`.

MATERIALS

Add water, aluminum, steel, and peizoelectric materials from the Material Library.

ADD MATERIAL

1 On 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>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Water, liquid (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Water domains**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Aluminum**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Aluminum (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Aluminum**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Steel AISI 4340**.

3 Click **Add to Component** in the window toolbar.

MATERIALS

Steel AISI 4340 (mat3)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Steel**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-4)**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Lead Zirconate Titanate (PZT-4) (mat4)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Piezo domains**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Define physics settings and boundary conditions.

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 **Water domains**.

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

5 Locate the **Typical Wave Speed for Perfectly Matched Layers** section. In the c_{ref} text field, type `1500[m/s]`.

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 In the **Settings** window for **Far-Field Calculation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Far Field boundaries**.

4 Locate the **Far-Field Calculation** section. Select the **Symmetry in the z=0 plane** check box.

5 From the **Type of integral** list, choose **Full integral**.

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 **Solid 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 **+Z poled Piezo**.

4 In the **Model Builder** window, click **Solid Mechanics (solid)**.

Piezoelectric Material 2

1 On the **Physics** toolbar, click **Domains** and choose **Piezoelectric Material**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **-Z poled Piezo**.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Fixed boundaries**.

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 **Piezo domains**.

Ground 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Ground boundaries**.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for **Electric Potential**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Voltage boundaries**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `V0`.

Mesh the geometry; create a tetrahedral mesh in the solid and the water-inner domains and create a swept mesh in the PML.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

3 From the **Sequence type** list, choose **User-controlled mesh**.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Non-PML domains**.

Define a mesh size in the water domain to ensure that the smallest wavelength is resolved by at least 5 elements (here we select `lambda0/6`).

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Water domains**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `1500[m/s]/f0max/6`.

Size 2

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Solid domains**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Resolution of narrow regions** check box.

6 In the associated text field, type `1`.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

Create a boundary layer at the external boundaries of the water domain. This will ensure numerically well defined normal gradients used in the far-field calculation feature. Turn off the smoothing option.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Water domain - Inner**.

5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Far Field boundaries**.

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type `1`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `1500[m/s]/f0max/6/20`.

7 In the **Model Builder** window, click **Mesh 1**.

8 In the **Settings** window for **Mesh**, click **Build All**.

9 Click the **Go to Default View** button on the **Graphics** toolbar.

DEFINITIONS

Import the file containing the variable definitions. These variables will mainly be used for post-processing calculations.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `tonpilz_transducer_variables.txt`.

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 `range(f0min,f0step,f0max)`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (acpr)

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 1**.

Delete the default surface plot and replace it by a multislice plot, which shows the value of the acoustic pressure inside the water domain.

Surface 1

In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node.

Multislice 1

1 Right-click **Surface 1** and choose **Delete**.

2 On the **Acoustic Pressure (acpr)** toolbar, click **More Plots** and choose **Multislice**.

3 Click **Plot**.

4 Click the **Zoom Extents** button on the **Graphics** toolbar.

This plot should look like Figure 5.

Sound Pressure Level (acpr)

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level (acpr)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 1**.

Do the same as for the acoustic pressure plot.

Surface 1

In the **Model Builder** window, expand the **Sound Pressure Level (acpr)** node.

Multislice 1

1 Right-click **Surface 1** and choose **Delete**.

2 On the **Sound Pressure Level (acpr)** toolbar, click **More Plots** and choose **Multislice**.

3 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

4 On the **Sound Pressure Level (acpr)** toolbar, click **Plot**.

This plot should look like Figure 6.

Far-Field Sound Pressure Level (acpr)

Plot the sound pressure level at 10 meters relative to the value on the Z-axis

Far Field 1

1 In the **Model Builder** window, under **Results>Far-Field Sound Pressure Level (acpr)** click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Expression** section.

3 In the **Expression** text field, type `SPL_rel`.

4 Locate the **Evaluation** section. Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.

5 In the θ **start** text field, type `90`.

6 In the θ **range** text field, type `90`.

7 Find the **Sphere** subsection. From the **Sphere** list, choose **Manual**.

8 In the **Radius** text field, type `abs(Zeval)`.

9 Locate the **Expression** section. Select the **Use as color expression** check box.

10 On the **Far-Field Sound Pressure Level (acpr)** toolbar, click **Plot**.

Change the view in the graphics window to obtain a plot similar to Figure 7.

Plot the Far-Field Sound Pressure Level in XZ plane for several frequencies.

Far-Field Sound Pressure Level xy-Plane (acpr)

1 In the **Model Builder** window, expand the **Far-Field Pressure (acpr)** node, then click **Results>Far-Field Sound Pressure Level xy-Plane (acpr)**.

2 In the **Settings** window for **Polar Plot Group**, type `Relative polar beam sensitivity` 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 (Hz))** list, choose **1000**, **10000**, and **40000**.

Far Field 1

1 In the **Model Builder** window, expand the **Results>Relative polar beam sensitivity** node, then click **Far Field 1**.

2 In the **Settings** window for **Far Field**, locate the **Expression** section.

3 In the **Expression** text field, type `SPL_rel`.

4 Locate the **Evaluation** section. Find the **Angles** subsection. From the **Restriction** list, choose **Manual**.

5 In the ϕ **start** text field, type `-90`.

6 In the ϕ **range** text field, type `180`.

7 Find the **Reference direction** subsection. In the **x** text field, type `0`.

8 In the **y** text field, type `0`.

9 In the **z** text field, type `-1`.

The reference direction defines that 0 deg. in the polar plot corresponds to the negative z-axis direction.

10 Find the **Normal** subsection. In the **x** text field, type `0`.

11 In the **y** text field, type `1`.

12 In the **z** text field, type `0`.

13 Find the **Evaluation distance** subsection. In the **Radius** text field, type `abs(Zeval)`.

14 Click to expand the **Legends** section. Select the **Show legends** check box.

Relative polar beam sensitivity

1 In the **Model Builder** window, under **Results** click **Relative polar beam sensitivity**.

2 On the **Relative polar beam sensitivity** toolbar, click **Plot**.

This plot should look like Figure 8.

Replace the stress plot by displacement in solid.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **View 1**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Displacement** 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>Solid Mechanics> Displacement>solid.disp - Total displacement**.

Displacement

1 In the **Model Builder** window, under **Results** click **Displacement**.

2 On the **Displacement** toolbar, click **Plot**.

This plot should look like Figure 9.

DEFINITIONS

Create a new view to plot the electric potential in the piezoelectric disks.

View 5

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

RESULTS

Electric Potential (es)

1 In the **Model Builder** window, expand the **View 5** node, then click **Results> Electric Potential (es)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 5**.

4 On the **Electric Potential (es)** toolbar, click **Plot**.

Change the view in the graphics window to obtain a plot similar to Figure 10.

Insert a 1D plot group to obtain the on-axis pressure graph shown in Figure 11.

First, create a parametrized curve data set to enable plotting of the far-field variable outside the computational mesh: from the edge of the far-field boundary to 500 mm in front of the transducer.

Parameterized Curve 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Parameterized Curve 3D**.

2 In the **Settings** window for **Parameterized Curve 3D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 (1) (sol1)**.

4 Locate the **Expressions** section. In the **z** text field, type `(1-s)*(-1.02*Rwater)-s*500[mm]`.

5 Select the **Only evaluate globally defined expressions** check box.

1D Plot Group 9

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `On-axis Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **Last**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `z (mm)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Pressure (Pa)`.

Line Graph 1

1 Right-click **On-axis Pressure** and choose **Line Graph**.

2 Select Edge 136 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>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>p - Pressure**.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `z`.

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **On-axis Pressure** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Parameterized Curve 3D 1**.

4 From the **Parameter selection (freq)** list, choose **Last**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `pfar(x,y,z)`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `z`.

8 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

9 On the **On-axis Pressure** toolbar, click **Plot**.

Insert a 1D plot to obtain the calculated specific acoustic impedance shown in Figure 12.

1D Plot Group 10

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Specific Acoustic Impedance` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Specific Acoustic Impedance (Zaco)`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (kHz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Z/(\rho c)`.

9 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Specific Acoustic Impedance** 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
`abs(Zaco)`	1	
`real(Zaco)`	1	
`imag(Zaco)`	1	

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `freq`.

6 From the **Unit** list, choose **kHz**.

7 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Absolute value`
`Real part`
`Imaginary part`

9 On the **Specific Acoustic Impedance** toolbar, click **Plot**.

Specific Acoustic Impedance

Duplicate the last plot group and replace it by the transmitting voltage response to obtain the graph shown in Figure 13.

Specific Acoustic Impedance 1

1 In the **Model Builder** window, under **Results** right-click **Specific Acoustic Impedance** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Transmitting Voltage Response` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Transmitting Voltage Response`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `TVR (dB rel. 1\mu Pa/V)`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 In the **Model Builder** window, expand the **Results>Transmitting Voltage Response** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 Click to select row number 3 in the table.

4 Click **Delete**.

5 Click to select row number 2 in the table.

6 Click **Delete**.

7 In the table, enter the following settings:

Expression	Unit	Description
TVR		Transmitting Voltage Response (TVR)

8 Locate the **Legends** section. From the **Legends** list, choose **Automatic**.

9 On the **Transmitting Voltage Response** toolbar, click **Plot**.

An alternative approach to create the above plot is to use the Octave Band Plot. Set the geometric entity level to global, set the expression to pfar(0,0,-1), reference to 1[uPa/V]/Vrms, and the style to continous.

Transmitting Voltage Response

Duplicate the last plot group and modify it to plot the directivity index like in Figure 14.

Transmitting Voltage Response 1

1 In the **Model Builder** window, under **Results** right-click **Transmitting Voltage Response** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Directivity Index (DI)` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Directivity Index (DI)`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `DI (dB)`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 In the **Model Builder** window, expand the **Results>Directivity Index (DI)** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
DI		Directivity index of Tonpilz transducer
DI_fl_pist		Directivity index of flanged piston

4 On the **Directivity Index (DI)** toolbar, click **Plot**.

Directivity Index (DI)

Duplicate the last plot group and modify it to total radiated power like in Figure 15.

Directivity Index (DI) 1

1 In the **Model Builder** window, under **Results** right-click **Directivity Index (DI)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Total Radiated Power` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Total Radiated Power`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Ptot (mW)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 In the **Model Builder** window, expand the **Results>Total Radiated Power** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 Click to select row number 2 in the table.

4 Click **Delete**.

5 In the table, enter the following settings:

Expression	Unit	Description
Ptot	mW	Total radiated power

6 On the **Total Radiated Power** toolbar, click **Plot**.

Create a grid data set in order to plot the far-field pressure `pfar(x,y,z)` outside of the computational mesh. This will reproduce Figure 16.

Grid 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Grid>Grid 3D**.

2 In the **Settings** window for **Grid 3D**, locate the **Parameter Bounds** section.

3 Find the **First parameter** subsection. In the **Maximum** text field, type `0`.

4 Find the **Second parameter** subsection. In the **Minimum** text field, type `-200`.

5 In the **Maximum** text field, type `200`.

6 Find the **Third parameter** subsection. In the **Maximum** text field, type `-200`.

7 Click to expand the **Resolution** section. In the **x resolution** text field, type `2`.

8 In the **y resolution** text field, type `150`.

9 In the **z resolution** text field, type `150`.

3D Plot Group 14

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Exterior Pressure Field` in the **Label** text field.

Surface 1

1 Right-click **Exterior Pressure Field** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Data set** list, choose **Grid 3D 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `if(sqrt(x^2+y^2+z^2)>Rwater,pfar(x,y,z),NaN)`.

The `if()` statement ensures that `pfar(x,y,z)` is only plotted outside of the far-field calculation boundary where it makes mathematical sense to do so. The value NaN will plot nothing.

5 On the **Exterior Pressure Field** toolbar, click **Plot**.

6 Click the **Go to YZ View** button on the **Graphics** toolbar.

Now, create a second plot that represents the sound pressure level outside of the transducer and reproduce Figure 17. The SPL is also plotted inside the transducer.

3D Plot Group 15

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Exterior SPL` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **20000**.

Surface 1

1 Right-click **Exterior SPL** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Data set** list, choose **Grid 3D 1**.

4 From the **Parameter value (freq (Hz))** list, choose **20000**.

5 Locate the **Expression** section. In the **Expression** text field, type `if(sqrt(x^2+y^2+z^2)>Rwater,acpr.ffc1.Lp_pfar,NaN)`.

Duplicate the main data set and add a selection such that variables are not plotted in the PML domain. Use it to also plot the SPL inside the computational mesh.

Selection

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Study 1/Solution 1 (1) (sol1)** and choose **Duplicate**.

2 On the **Results** toolbar, click **Selection**.

3 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 From the **Selection** list, choose **Non-PML domains**.

Slice 1

1 In the **Model Builder** window, under **Results** right-click **Exterior SPL** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 (2) (sol1)**.

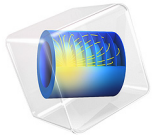
4 From the **Parameter value (freq (Hz))** list, choose **20000**.

5 Locate the **Expression** section. In the **Expression** text field, type `acpr.Lp`.

6 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

7 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

8 On the **Exterior SPL** toolbar, click **Plot**.



Piezoelectric Tonpilz Transducer with a Prestressed Bolt

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

A tonpilz transducer, such as the one shown in Figure 1, is used for relatively low frequency, high power sound emission. It is one of the popular transducer configurations for SONAR applications. The transducer consists of piezoceramic rings stacked between a head mass and a tail mass which are connected by a central bolt. This example shows how to incorporate the effect of a pre-tension in the bolt. The bolt geometry is imported from the Part Libraries. The frequency response of the transducer is studied to determine structural and acoustic response of the device such as deformation, stresses, radiated power, sound pressure level, the transmitting voltage response (TVR) curve, and the directivity index (DI) of the sound beam.

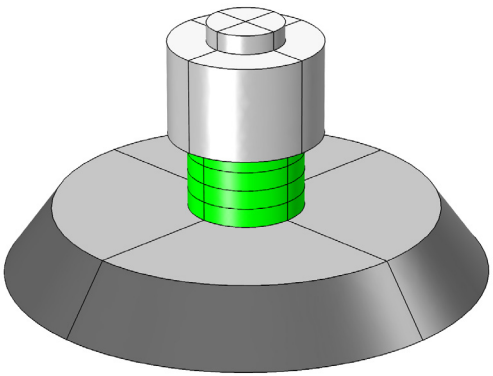


Figure 1: A tonpilz transducer. The aluminum head mass is shown in dark gray, the central steel bolt and steel tail mass are shown in light gray and the piezostack actuator with four disks of PZT-4 is shown in green.

Note: This application requires both the Acoustics Module and the Structural Mechanics Module.

A different version of this tutorial entitled *Piezoelectric Tonpilz Transducer*, is available in the Application Libraries under Acoustics Module only. That version of the tutorial does not implement the pre-tension in the bolt and hence does not require the Structural Mechanics Module. For additional details related to different customized settings and

user-defined variables used in both the tonpilz transducer models, you are encouraged to read the documentation of the *Piezoelectric Tonpilz Transducer* tutorial.

Model Definition

The basic working principle involved in the operation of this transducer is that an AC electrical signal applied to the piezostack actuator produces vibration in the entire transducer which in turn produces sound waves in the surrounding fluid. Thus modeling the operation of the transducer requires coupling electrical, structural and acoustic phenomena.

In this tutorial we will particularly emphasize on the implementation of pre-tension in the central bolt of the transducer and associated solver settings that allow us to model the effect of prestress on the frequency response characteristics of the transducer.

The parameters used in this model are shown in Table 1.

TABLE 1: LIST OF MODELING PARAMETERS

NAME	EXPRESSION	DESCRIPTION
Rwater	40[mm]	Water domain radius
Rpml	10[mm]	PML layer thickness
a	25[mm]	Piston head radius
Zeval	-10[m]	Directivity evaluation distance
Vrms	1[V]	RMS drive voltage
V0	sqrt(2)*Vrms	Zero-to-peak drive voltage
f0min	1[kHz]	Minimum operating frequency
f0max	40[kHz]	Maximum operating frequency
f0step	1[kHz]	Frequency step
F_pre	1[kN]	Bolt pre-stress force

PHYSICS IMPLEMENTATION

The Acoustic-Piezoelectric Interaction, Frequency Domain multiphysics interface available in the Acoustics Module is used for simulating the multiphysics interactions. This predefined interface includes the necessary fundamental physics which are Pressure Acoustics, Solid Mechanics and Electrostatics. The Pressure Acoustics interface is used to solve for the wave equation in the water domain. The Solid Mechanics physics is solved on all structural materials including the PZT-4 disks. The Electrostatics physics is solved on the PZT-4 disks. The multiphysics couplings necessary to model this system are available as predefined nodes under the Multiphysics branch. These couplings are:

Acoustic-Structure Boundary: This node is active on the boundaries that are at the interface of the water domain and transducer head mass. On these boundaries a bidirectional coupling is automatically set up. The fluid pressure evaluated by the Pressure Acoustics physics is applied as a mechanical load in the Solid Mechanics physics. Furthermore, the normal component of the structural acceleration is used as a sound source.

Piezoelectric Effect: This node is active on the PZT-4 domains only and couple the Solid Mechanics and Electrostatics equations solved in these domains via the linear constitutive equations that model the piezoelectric effect by coupling stresses and strains with electric field and electric displacement.

BOUNDARY SETTINGS

The outer curved surface of the steel tail mass is assumed to be fixed. Each of the piezo disks are excited with a 1 V RMS electrical signal.

The head mass is exposed to an unbounded region of water. A Perfectly Matched Layer (PML) is used to model the absorption of sound waves as they propagate far away from the sound source. Although PML is strictly not a boundary condition, it is used to imitate the effect of an open boundary.

A far-field calculation is set up on the interface boundaries between the inner water domain and the PML domains. The far-field integral type is set to *Full integral* which allows computation of both amplitude and phase of the acoustic pressure and sound pressure level (SPL) at any point in space outside the computational domain. The far field pressure variable is used to compute the TVR and DI.

The user-defined variables used to compute the transducer characteristics are shown in Table 2.

TABLE 2: LIST OF VARIABLES

NAME	EXPRESSION	DESCRIPTION
rho0	intop3(acpr.rho)	Density of water at room temperature
c0	intop3(acpr.c)	Speed of sound in water at room temperature
Zaco	intop2(p)/intop2(acpr.iomega*(w+eps))/(rho0*c0)	Specific acoustic impedance
pfar_1	pfar(0,0,-1)	Far field pressure at 1 m
prms	sqrt(0.5*pfar_1*conj(pfar_1))[Pa]	RMS pressure at 1 m

TABLE 2: LIST OF VARIABLES

NAME	EXPRESSION	DESCRIPTION
TVR	20*log10(prms/Vrms/1[uPa/V])	Transmitting Voltage Response (TVR)
pfar_Zeval	pfar(0,0,Zeval[1/m])	Far field pressure at Zeval
Ifront	0.5*pfar_Zeval*conj(pfar_Zeval)[Pa^2]/(rho0*c0)	On-axis intensity at Zeval
Ptot	intop1(down(acpr.lx)*acpr.nx+down(acpr.ly)*acpr.ny+down(acpr.lz)*acpr.nz)	Total radiated power
Iave	Ptot/(4*pi*Zeval^2)	Average intensity of monopole source at Zeval
Di	Ifront/Iave	Intensity directivity
DI	10*log10(Di)	Directivity index of Tonpilz transducer
k0	2*pi*freq/c0	Wave number
DI_fl_pist	10*log10((k0*a)^2/(1-2*besselj(1,2*k0*a)/(2*k0*a)))	Directivity index of flanged piston

MODELING A BOLT WITH PRE-TENSION

When a bolt is mounted on a device to clamp the components, it is tightened by twisting the bolt head. As a reaction to the tightening process, a pre-tension force is experienced by the bolt. This force produces a prestress that helps to hold the bolt in place during regular operation of the device. This also ensures that when additional stresses develop in the bolt during operation of the device, it should not become loose. Note that the tightening of the bolt also produces stresses in materials that are touching the bolt. This is why accounting for the pre-tension force in the bolt would give us an accurate picture of the pre-stress distribution not only in the bolt but also in the entire device.

COMSOL's Structural Mechanics Module provides a Bolt Pre-Tension feature that can be used to implement a desired pre-tension or prestress in bolted joints. You can import the bolt geometry from the Part Libraries. These bolt geometries are created in a certain way so that we can directly use the Bolt Pre-Tension feature on them. In order to use this feature, there should be a cross section surface passing through the shank of the bolt. This surface needs to be associated with the Bolt Selection subnode under the Bolt Pre-Tension node. COMSOL sets up an additional equation for each bolt that computes the pre-deformation of the bolt, the pre-tension force as well as the shear force in the bolt. For example, in this model, on application of 1 kN pre-tension, we get a pre-deformation of 12 μm.

Note that the deformation along the longitudinal axis of the bolt is discontinuous at the surface assigned to the Bolt Selection subnode but the stresses and strains are continuous. For more details on implementation of the Bolt Pre-Tension feature, you can refer to the section on *Using Pre-tensioned Bolts* in the Structural Mechanics Module User's Guide.

As a result of the prestress in the device, if we want to solve a vibrations problem in frequency domain, we need to account for the fact that the harmonic variation of stress and other physical quantities during vibration takes place on top of the static bias stress. Hence we need to solve this model using a two-step approach where the first step involves solving for the static stress distribution using a Stationary study step. The solution from this step is then used as a linearization point for solving the vibration problem in Frequency Domain Perturbation study step.

Note that this workflow is valid only for small perturbations about the static solution. Hence we should only use this technique if the magnitude of the stress and other physical quantities from the frequency domain problem is significantly smaller than the magnitude of the same quantities obtained from the static problem.

The AC voltage signal applied to the piezostack actuator is specified using the `linper()` operator. This operator ensures that the numerical input is used only in the Frequency Domain, Perturbation step and not in the Stationary step when solving the model. Furthermore, when solving the vibrations problem in frequency domain, you only want to account for the stress generated in the device as a result of its operation and hence you do not want to solve the pre-deformation variable in the bolt. This is ensured by changing the solver settings for the frequency domain part and instructing the solver to not solve for the pre-deformation variable.

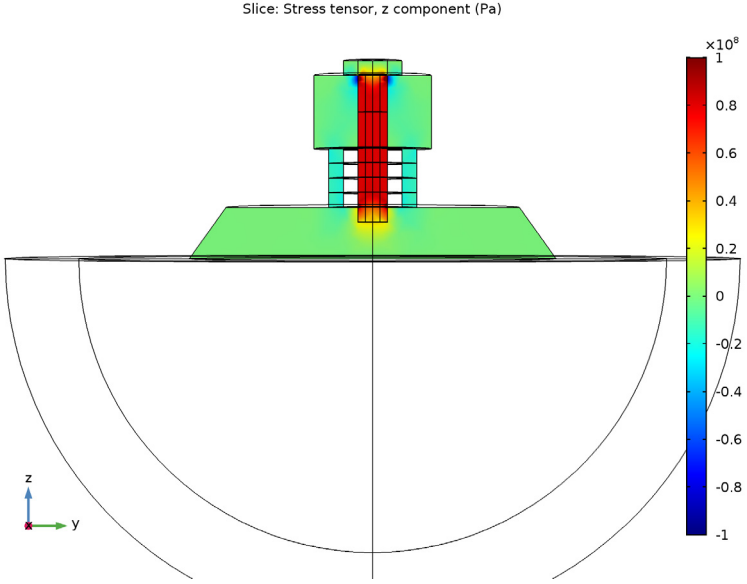


Figure 2: A slice plot showing the z-component of the prestress in the central bolt and other parts of the transducer. This stress arises due to the pre-tension in the bolt.

Figure 2 shows the z-component of the prestress in the transducer. The prestress is fairly uniform in the bolt. Note that the prestress exists mainly in the shank of the bolt and in the central part of the Aluminum head mass which is located directly below the bolt.

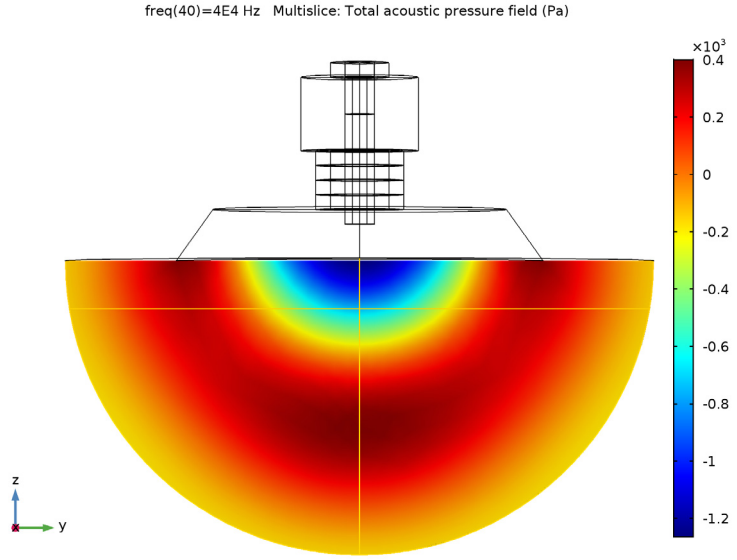


Figure 3: A slice plot showing the total acoustic pressure variation in the water domain at 40 kHz.

Figure 3 shows the total acoustic pressure in the water domain for 40 kHz excitation. The inner water domain in the model captures roughly half a wavelength at this frequency.

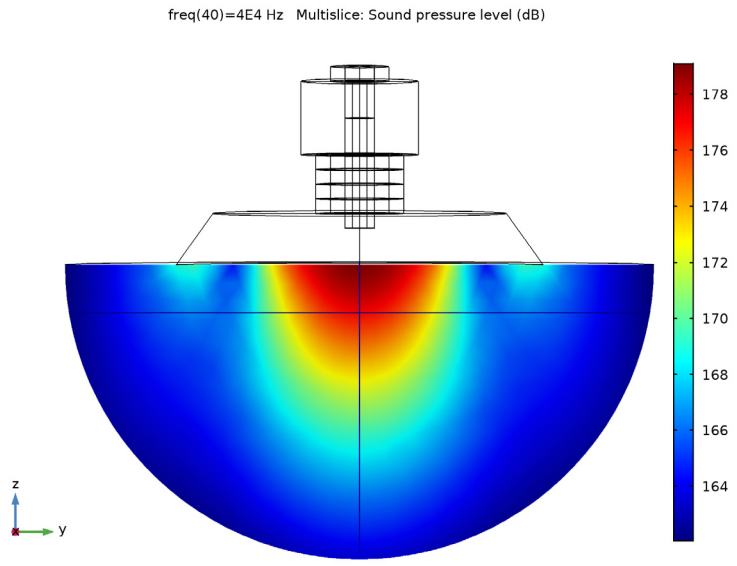


Figure 4: A slice plot showing the variation in sound pressure level (in dB) in the water domain at 40 kHz.

Figure 4 shows the sound pressure level (SPL) in the water domain for 40 kHz excitation. The SPL is highest near the transducer head mass. The variation in the SPL around the transducer would depend on the operating frequency and the dominant mode in which the transducer vibrates.

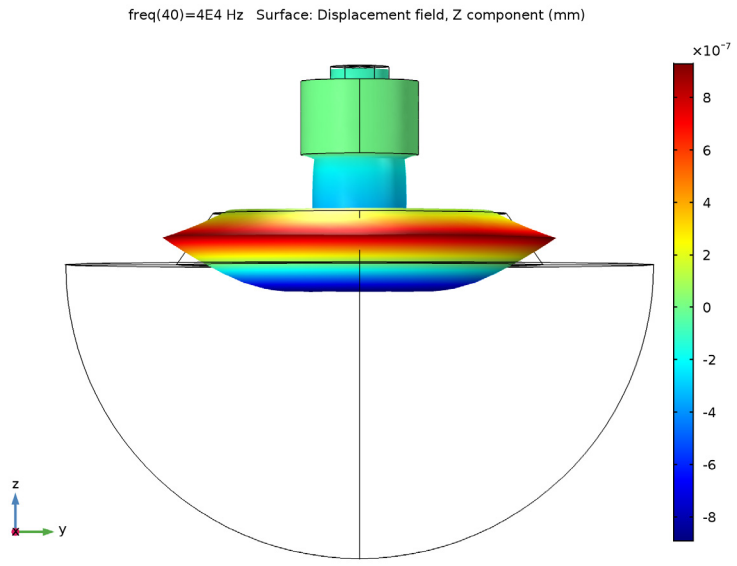


Figure 5: Total structural displacement of the transducer at 40 kHz. An exaggerated deformation has been used for better visualization.

Figure 5 shows the total structural displacement of the tonpilz transducer at 40 kHz excitation. At this frequency, the head mass vibrates in a mode whose shape is somewhat toroidal. If you observe the vibration mode for relatively lower frequencies, say 1 kHz, you will observe that the head mass vibrates mainly along its axis similar to a flanged piston.

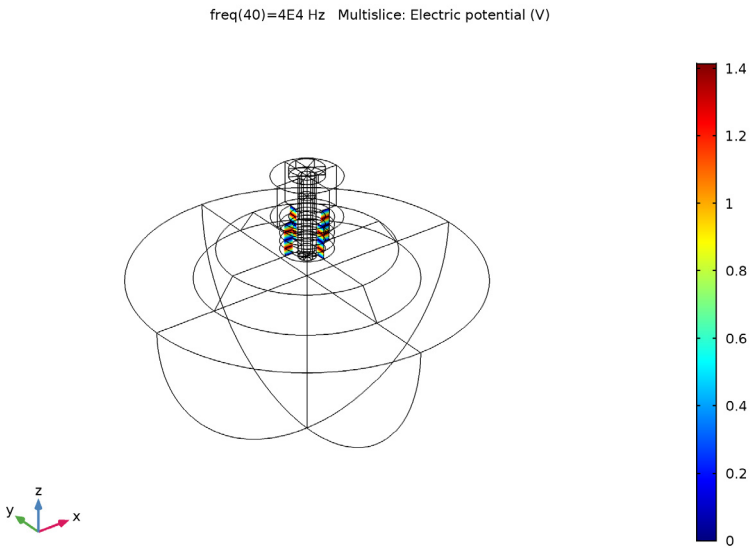


Figure 6: A multislice plot of the electric potential distribution within the four PZT-4 disks.

Figure 6 shows the electric potential distribution through the thickness of the PZT-4 disks. The piezoelectric disks are stacked in a way such that alternate disks are poled along opposite directions. This allows us to use a single electrical terminal at the interface of each pair of disks and obtain the piezoelectric actuation effect in each of the disks along the same direction. Having the piezoelectric strain in-phase in all the disks maximizes the actuation.

In this model, the PZT-4 disks actuate in the d_{33}-mode. Hence two of the disks are poled along the +Z-direction while the other two are poled along the -Z-direction. The default definition of the piezoelectric material properties in COMSOL's Global Coordinate System automatically creates a +Z polarization. In order to create a -Z polarization, a user-defined Rotated Coordinate System is used. In this coordinate system, the Euler angles are set to $\alpha = 0$, $\beta = \pi$ and $\gamma = 0$.

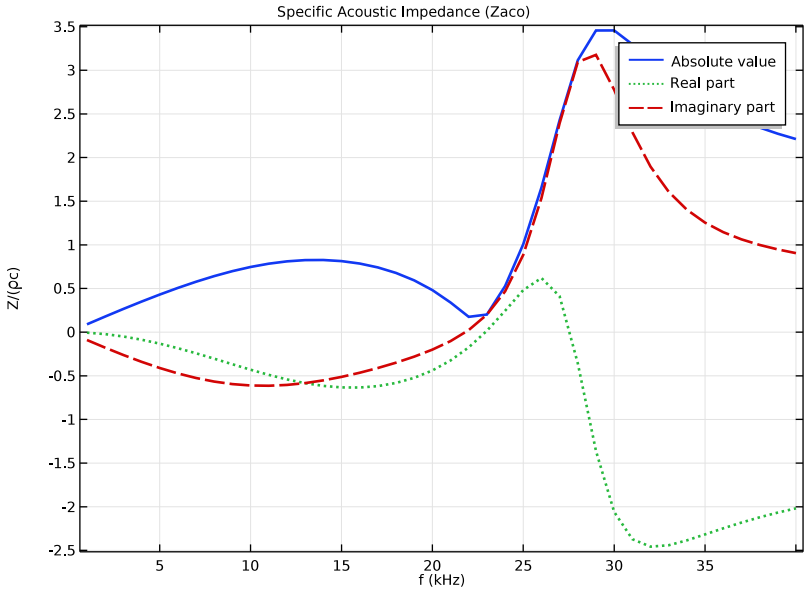


Figure 7: Frequency response plot of the absolute value, real and imaginary components of the specific acoustic impedance at the interface between the head mass and water.

Figure 7 shows the frequency response of the specific acoustic impedance of the head mass surface that is exposed to water. A resonance and anti-resonance is observed within the frequency range of 20 kHz and 35 kHz.

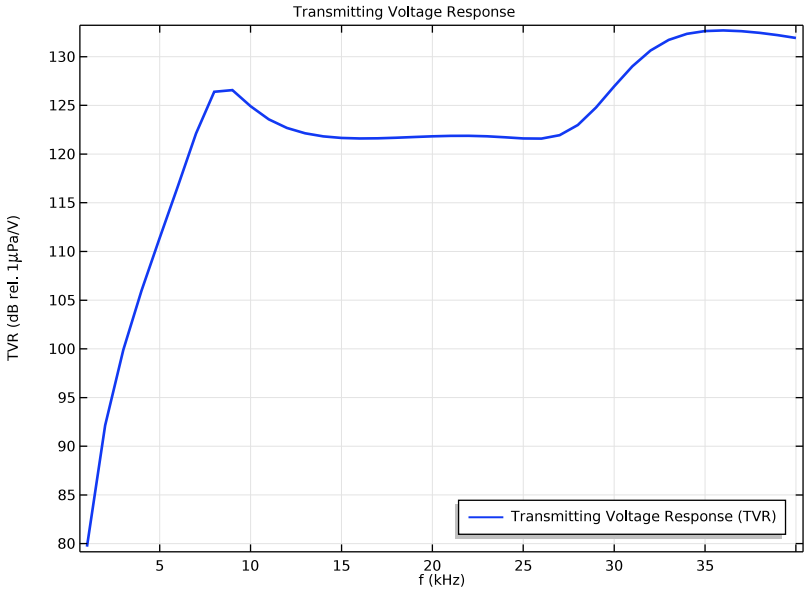


Figure 8: Transmitting Voltage Response (TVR) as a function of frequency obtained at an on-axis distance of 1 m ahead of the head mass and computed relative to 1 μPa/V.

Figure 8 shows the variation in the TVR of the transducer as a function of operating frequency. The fairly flat region between 15 kHz and 25 kHz can be particularly useful for sensing applications.

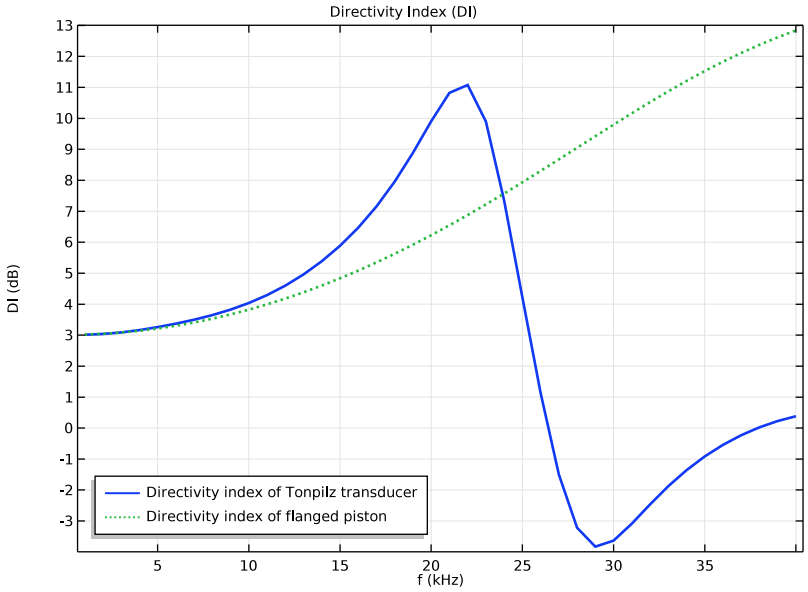


Figure 9: Frequency response of the Directivity Index (DI) computed at an on-axis distance of 10 m from the head mass. The DI of the tonpilz transducer is compared to that of a flanged piston.

Figure 9 shows the Directivity Index (DI) of the tonpilz transducer (blue curve) and compares it with the DI of a flanged piston (green curve). The latter can be computed from analytical expression as shown in Table 2. It is defined by the variable `DI_fl_pist`. Note that when the tonpilz transducer operates like a piston at lower frequencies, its DI becomes very similar to that of a flanged piston. Another feature worth noting is that within the range of 15 kHz and 25 kHz, the DI of the tonpilz transducer changes from nearly 4 dB to 11 dB while its TVR remains nearly constant. This can make the transducer quite versatile within this operating range.

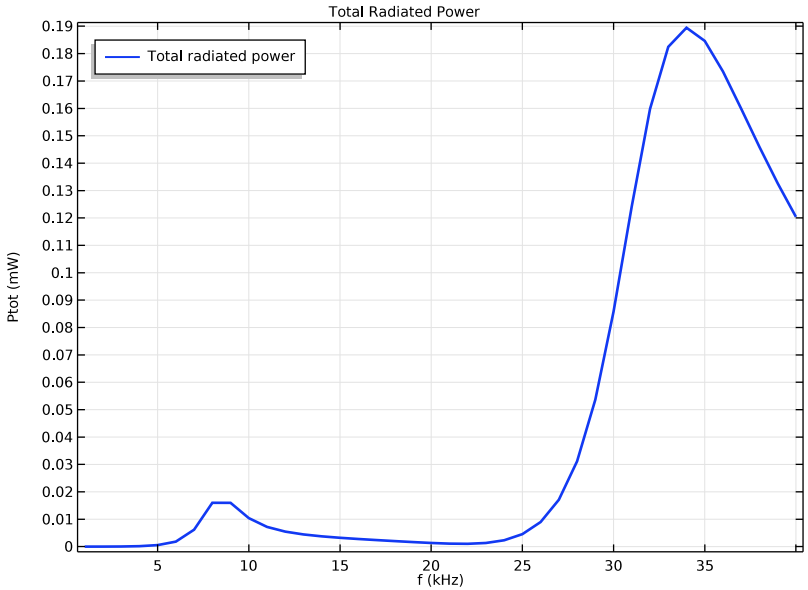


Figure 10: Total radiated power from the tonpilz transducer within the operating frequency range of 1 kHz to 40 kHz.

Figure 10 shows the total radiated power as a function of the operating frequency of the tonpilz transducer. Since the radiated power is proportional to the sound intensity, the peak in power at around 8 kHz and 34 kHz also indicate that peak sound intensity is obtained at these frequencies.

Application Library path: `Acoustics_Module/Piezoelectric_Devices/tonpilz_transducer_prestressed`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Piezoelectric Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Bolt Pre-Tension**.

6 Click **Done**.

GEOMETRY 1

1 In the **Model Builder** window, click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Import the file containing the model parameters.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `tonpilz_transducer_prestressed_parameters.txt`.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

The modeling geometry is created by first drawing the cross section on a workplane and then revolving this cross section to get the 3-dimensional geometry. The central bolt in the transducer is later added from the Part Libraries.

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

2 From the **Plane** list, choose **xz-plane**.

3 Click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `6`.

4 In the **Height** text field, type `10`.

5 Locate the **Position** section. In the **xw** text field, type `2`.

6 In the **yw** text field, type `15`.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `2`.

4 In the **Height** text field, type `8`.

5 Locate the **Position** section. In the **xw** text field, type `4`.

6 In the **yw** text field, type `7`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	2
`Layer 2`	2
`Layer 3`	2

Polygon 1 (pol1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the **xw** text field, type `0 0 2 2 20 a`.

4 In the **yw** text field, type `0 5 5 7 7 0`.

Circle 1 (c1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Rwater+Rpml`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	Rpml

7 On the **Work Plane** toolbar, click **Build All**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.

9 Click **Close**.

Work Plane 1 (wp1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Work Plane 1 (wp1)**.

Revolve 1 (rev1)

1 On the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type 90.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.

Explicit Selection 1 (sel1)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

Create domain and boundary selections that will be used in the model.

2 In the **Settings** window for **Explicit Selection**, type Water domain - Inner in the **Label** text field.

3 On the object **rev1**, select Domain 2 only.

Explicit Selection 2 (sel2)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type Water domain - PML in the **Label** text field.

3 On the object **rev1**, select Domain 1 only.

Explicit Selection 3 (sel3)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type Aluminum in the **Label** text field.

3 On the object **rev1**, select Domain 3 only.

Explicit Selection 4 (sel4)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Steel Part` in the **Label** text field.

3 On the object **rev1**, select Domain 4 only.

4 Locate the **Resulting Selection** section. Click **New**.

5 Clear the **Keep selection** check box.

6 In the **New Cumulative Selection** dialog box, type `Steel` in the **Name** text field.

7 Click **OK**.

Explicit Selection 5 (sel5)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `+Z poled Piezo` in the **Label** text field.

3 On the object **rev1**, select Domains 5 and 7 only.

Explicit Selection 6 (sel6)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `-Z poled Piezo` in the **Label** text field.

3 On the object **rev1**, select Domains 6 and 8 only.

Explicit Selection 7 (sel7)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Ground boundaries` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

5 On the object **rev1**, select Boundaries 19, 25, and 29 only.

Explicit Selection 8 (sel8)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Voltage boundaries` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **rev1**, select Boundaries 22 and 28 only.

Explicit Selection 9 (sel9)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Fixed boundaries` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **rev1**, select Boundary 36 only.

Explicit Selection 10 (sel10)

1 On the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Far Field boundaries` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **rev1**, select Boundary 6 only.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, locate the **Input** section.

3 From the **Input objects** list, choose **Revolve 1**.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type `range(0,90,270)`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build All Objects**.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

The following Union operation combines all the geometry objects created so far. This step ensures that when later on an Assembly option is used for finalizing the geometry sequence, the process does not create additional and unnecessary boundaries at the interface of all the geometry objects.

2 In the **Settings** window for **Union**, locate the **Union** section.

3 From the **Input objects** list, choose **Rotate 1**.

The CAD geometry of a simple bolt with no thread is imported from the Part Libraries. The design parameters of the bolt are adjusted to position the bolt in the tonpilz transducer.

PART LIBRARIES

1 On the **Geometry** toolbar, click **Parts** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **Structural Mechanics Module>Bolts> simple_bolt_no_thread** in the tree.

3 Click **Add to Geometry**.

GEOMETRY 1

Simple Bolt, No Thread 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Simple Bolt, No Thread 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
hdia	8[mm]	8 mm	Head diameter
hthic	2[mm]	2 mm	Head thickness
ndia	4[mm]	4 mm	Nominal diameter
blen	20[mm]	20 mm	Bolt length

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Head inner plane (wp1)**.

5 Find the **Displacement** subsection. In the **zw** text field, type 25.

6 Click to expand the **Domain selections** section. Locate the **Domain Selections** section. Click to select row number 1 in the table.

7 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
All	Steel		√

8 Click to expand the **Boundary selections** section. Locate the **Boundary Selections** section. Click to select row number 5 in the table.

9 In the table, enter the following settings:

Name	Contribute to	Keep	Physics
Exterior	None		√
Shank	None		√
Head, free surface	None		√
Head, contact surface	None		√
Pre-tension cut	None	√	√

10 Click **Build Selected**.

The geometry finalization method is changed to Form an Assembly to ensure that the bolt is not glued or rigidly attached to the adjacent parts.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Select the **Create imprints** check box.

5 On the **Geometry** toolbar, click **Build All**.

DEFINITIONS

An Identity Pair is used to get continuity in solution between the external surfaces of the bolt and the surfaces of other materials touching them. This continuity in solution is applicable for the lower end of the shank that is bolted into the aluminum head mass and the lower surface of the bolt head which is resting on the steel tail mass. The outer surface of the shank should be allowed to slip through the hole in the tail mass. Hence those boundaries need to be removed from the Identity Pair by modifying the default Identity Pair that has been created.

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Identity Boundary Pair 1 (ap1)**.

2 In the **Settings** window for **Pair**, locate the **Pair Type** section.

3 Select the **Manual control of selections** check box.

4 Locate the **Source Boundaries** section. Select the **Active** toggle button.

5 Click **Clear Selection**.

6 Click **Paste Selection**.

7 In the **Paste Selection** dialog box, type `59-66 71 72 75 76 112-114 126-129 135 160 163 164 167` in the **Selection** text field.

8 Click **OK**.

9 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

10 Select the **Active** toggle button.

11 Click **Clear Selection**.

12 Click **Paste Selection**.

13 In the **Paste Selection** dialog box, type `174 175 178 183 185-188 196 198 206 207 216 219 220 230-232 245 249 251 255 256 259` in the **Selection** text field.

14 Click **OK**.

15 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

16 Click **Create Selection**.

17 In the **Create Selection** dialog box, type `Destination boundaries` in the **Selection name** text field.

18 Click **OK**.

Union 1

1 On the **Definitions** toolbar, click **Union**.

Create additional domain selections that will be used in the model.

2 In the **Settings** window for **Union**, type `Water domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Water domain - Inner** and **Water domain - PML**.

5 Click **OK**.

Union 2

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Piezo domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **+Z poled Piezo** and **-Z poled Piezo**.

5 Click **OK**.

Complement 1

1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Solid domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Water domains** in the **Selections to invert** list.

5 Click **OK**.

Complement 2

1 On the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Non-PML domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Water domain - PML** in the **Selections to invert** list.

5 Click **OK**.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

Define an integration coupling operator on the far-field boundary.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Far Field boundaries**.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

Define an integration coupling operator on the acoustic-structure interface.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `14 15 87 125` in the **Selection** text field.

6 Click **OK**.

Integration 3 (intop3)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

Define an integration coupling operator to retrieve values at the summit point of the water domain.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `58` in the **Selection** text field.

6 Click **OK**.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

Import the file containing the variable definitions. These variables will mainly be used for post-processing calculations.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `tonpilz_transducer_prestressed_variables.txt`.

ADD MATERIAL

1 On 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>Water, liquid**.

4 Click **Add to Component 1**.

MATERIALS

Water, liquid (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Water, liquid (mat1)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Water domains**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Aluminum**.

3 Click **Add to Component 1**.

MATERIALS

Aluminum (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat2)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Aluminum**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Steel AISI 4340**.

3 Click **Add to Component 1**.

MATERIALS

Steel AISI 4340 (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat3)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Steel**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Piezoelectric>Lead Zirconate Titanate (PZT-4)**.

3 Click **Add to Component 1**.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Lead Zirconate Titanate (PZT-4) (mat4)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Piezo domains**.

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 **Water domains**.

4 Locate the **Sound Pressure Level Settings** section. From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

5 Locate the **Typical Wave Speed for Perfectly Matched Layers** section. In the c_{ref} text field, type `1500[m/s]`.

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 In the **Settings** window for **Far-Field Calculation**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Far Field boundaries**.

4 Locate the **Far-Field Calculation** section. Select the **Symmetry in the z=0 plane** check box.

5 From the **Type of integral** list, choose **Full integral**.

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 **Solid 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 **+Z poled Piezo**.

4 In the **Model Builder** window, click **Solid Mechanics (solid)**.

Piezoelectric Material 2

1 On the **Physics** toolbar, click **Domains** and choose **Piezoelectric Material**.

2 In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **-Z poled Piezo**.

Define a rotated system that will be used for the poling of the -Z poled piezoelectric disks.

DEFINITIONS

Rotated System 2 (sys2)

1 On the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for **Rotated System**, locate the **Settings** section.

3 Find the **Euler angles (Z-X-Z)** subsection. In the β text field, type `pi`.

SOLID MECHANICS (SOLID)

Piezoelectric Material 2

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Piezoelectric Material 2**.
2. In the **Settings** window for **Piezoelectric Material**, locate the **Coordinate System Selection** section.
3. From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

Fixed Constraint 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2. In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Fixed boundaries**.

Bolt Pre-Tension 1

1. On the **Physics** toolbar, click **Global** and choose **Bolt Pre-Tension**.
2. In the **Settings** window for **Bolt Pre-Tension**, locate the **Bolt Pre-Tension** section.
3. In the F_{p} text field, type `F_pre`.

Bolt Selection 1

1. In the **Model Builder** window, expand the **Bolt Pre-Tension 1** node, then click **Bolt Selection 1**.
2. In the **Settings** window for **Bolt Selection**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Pre-tension cut (Simple Bolt, No Thread 1)**.

Continuity 1

1. On the **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Continuity**.
2. In the **Settings** window for **Continuity**, locate the **Pair Selection** section.
3. In the **Pairs** list, select **Identity Boundary Pair 1 (ap1)**.

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 **Piezo domains**.

Ground 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Ground**.
2. In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Ground boundaries**.

Electric Potential 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for **Electric Potential**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Voltage boundaries**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `linper(V0)`.

The `linper()` operator ensures that the voltage V0 is only applied in the frequency domain perturbation study step and not during the stationary analysis. If you have the AC/DC Module you can right click and add the Harmonic Perturbation sub-feature, which will do the same.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

Insert a Perfectly Matched Layer to model the absorption of acoustic wave far away from the source.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Water domain - PML**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

5 Locate the **Scaling** section. From the **Coordinate stretching type** list, choose **Rational**.

6 In the **PML scaling factor** text field, type `0.5`.

7 In the **PML scaling curvature parameter** text field, type `5`.

Mesh the geometry; create a tetrahedral mesh in the solid and the water-inner domains and create a swept mesh in the PML.

MESH 1

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Non-PML domains**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** and choose **Size**.

Define a mesh size in the water domain to ensure that the smallest wavelength is resolved by at least 5 elements.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Water domains**.

4 Locate the **Element Size** section. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `1500[m/s]/f0max/5`.

Size 2

1 Right-click **Free Tetrahedral 1** and choose **Size**.

Apply a mesh setting on the Destination boundaries of the Identity Pair such that the mesh on these surfaces is somewhat finer than the mesh on the Source boundaries of the Identity Pair.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Destination boundaries**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, click **Build All**.

Create a boundary layer mesh at the external boundaries of the water domain. This will ensure numerically well defined normal gradients used in the far-field calculation feature.

Boundary Layers 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Water domain - Inner**.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Far Field boundaries**.

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type `1`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `1500[m/s]/f0max/5/20`.

7 Click **Build All**.

STUDY 1

Frequency Domain Perturbation

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain> Frequency Domain Perturbation**.

The Bolt Pre-Tension step solves for the effect of pre-tension in the bolt. This is a stationary step and does not solve the Acoustics problem - this can be seen from the small orange exclamation marks next to the acoustics in the **Physics and Variables Selection** section. The Frequency-Domain Perturbation step solves all the physics including the induced effect of the pre-tension of the bolt.

Step 2: Frequency Domain Perturbation

1 In the **Model Builder** window, under **Study 1** click **Step 2: Frequency Domain Perturbation**.

2 In the **Settings** window for **Frequency Domain Perturbation**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `range(f0min,f0step,f0max)`.

Make sure to select the **Include geometric nonlinearity** checkbox - if it is not selected the effect of the pre-tensioning will not be included as the linearization point for the frequency domain study.

4 Select the **Include geometric nonlinearity** check box.

5 In the **Model Builder** window, click **Study 1**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 On the **Study** toolbar, click **Compute**.

The following instructions describe how to create the plots shown in the **Results** section.

RESULTS

Data Sets

In the **Model Builder** window, expand the **Results** node.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/Solution 1 (sol1)**.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Non-PML domains**.

5 Select the **Propagate to lower dimensions** check box.

3D Plot Group 1

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Static stress from pre-tension` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution Store 1 (sol2)**.

4 Click the **Go to YZ View** button on the **Graphics** toolbar.

Slice 1

1 Right-click **Static stress from pre-tension** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 In the **Planes** text field, type `1`.

4 Locate the **Expression** section. In the **Expression** text field, type `solid.sz`.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Minimum** text field, type `-1e8`.

7 In the **Maximum** text field, type `1e8`.

8 On the **Static stress from pre-tension** toolbar, click **Plot**.

3D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure` in the **Label** text field.

Multislice 1

1 On the **Acoustic Pressure** toolbar, click **More Plots** and choose **Multislice**.

2 Click **Plot**.

Acoustic Pressure 1

1 In the **Model Builder** window, under **Results** right-click **Acoustic Pressure** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Results>Sound Pressure Level** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.Lp`.

4 On the **Sound Pressure Level** toolbar, click **Plot**.

3D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D 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**, locate the **Expression** section.

3 In the **Expression** text field, type `w`.

4 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

5 On the **Displacement** toolbar, click **Plot**.

Sound Pressure Level 1

1 In the **Model Builder** window, under **Results** right-click **Sound Pressure Level** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Potential` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Results>Electric Potential** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `V`.

4 On the **Electric Potential** toolbar, click **Plot**.

5 Click the **Go to Default View** button on the **Graphics** toolbar.

1D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Specific Acoustic Impedance` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Specific Acoustic Impedance (Zaco)`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `f (kHz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Z/(\rho c)`.

Global 1

1 Right-click **Specific Acoustic Impedance** 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
`abs(Zaco)`	1	
`real(Zaco)`	1	
`imag(Zaco)`	1	

4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Absolute value`
`Real part`
`Imaginary part`

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `freq`.

8 From the **Unit** list, choose **kHz**.

9 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

10 Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

11 On the **Specific Acoustic Impedance** toolbar, click **Plot**.

Specific Acoustic Impedance 1

1 In the **Model Builder** window, under **Results** right-click **Specific Acoustic Impedance** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Transmitting Voltage Response` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Transmitting Voltage Response`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `TVR (dB rel. 1\mu Pa/V)`.

5 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 In the **Model Builder** window, expand the **Results>Transmitting Voltage Response** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables> TVR - Transmitting Voltage Response (TVR)**.

3 Locate the **Legends** section. From the **Legends** list, choose **Automatic**.

4 On the **Transmitting Voltage Response** toolbar, click **Plot**.

Transmitting Voltage Response 1

1 In the **Model Builder** window, under **Results** right-click **Transmitting Voltage Response** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Directivity Index (DI)` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Directivity Index (DI)`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `DI (dB)`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

Global 1

1 In the **Model Builder** window, expand the **Results>Directivity Index (DI)** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>DI - Directivity index of Tonpilz transducer**.

3 Click **Add Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables>DI_fl_pist - Directivity index of flanged piston**.

4 On the **Directivity Index (DI)** toolbar, click **Plot**.

Directivity Index (DI) 1

1 In the **Model Builder** window, under **Results** right-click **Directivity Index (DI)** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Total Radiated Power` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Total Radiated Power`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Ptot (mW)`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

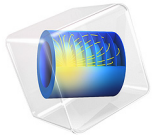
1 In the **Model Builder** window, expand the **Results>Total Radiated Power** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Definitions>Variables> Ptot - Total radiated power**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`Ptot`	`mW`	`Total radiated power`

4 On the **Total Radiated Power** toolbar, click **Plot**.



Transfer Impedance of a Perforate

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Perforates are plates with a distribution of small perforations or holes. They are used in muffler systems, sound absorbing panels, and in many other places as liners, where it is important to control attenuation precisely. As the perforations become smaller and smaller the viscous and thermal losses become more important. The attenuation behavior, which is also frequency dependent, can be controlled by selecting the perforate size and distribution in a plate. Non-linear loss mechanisms occur at higher sound levels or in the presence of a flow (through or over the perforate). Only the linear effects due to viscosity and thermal conduction are studied in this tutorial model. These effects are modeled in detail using the Thermoviscous Acoustics, Frequency Domain interface.

Perforates have been theoretically studied for many years. Typically, analytical or semi-analytical models only apply for simple geometries. A numerical approach is necessary for systems where the holes have various cross sections, if the perforations are tapered, or if the distribution of holes is uneven.

In this model a simple perforate with circular holes is studied. The transfer impedance, surface normal impedance, and attenuation coefficient of the system is determined. The transfer impedance determined in this detailed model can, for example, be used in a larger system simulation using the interior impedance condition that exists in the Pressure Acoustics, Frequency Domain interface.

Model Definition

A sketch of the perforate system simulated is depicted in Figure 1. The model uses the symmetries that exist in the geometry along the x- and y-directions.

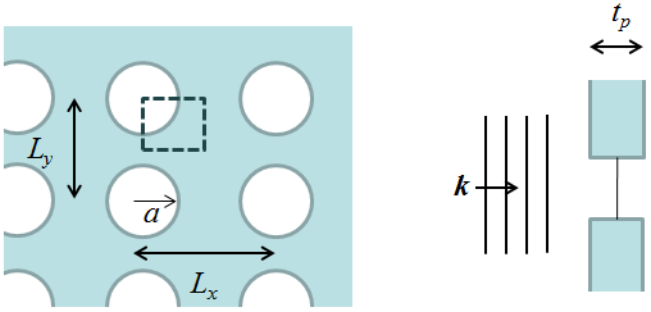


Figure 1: Schematic of the system, front and side views. The modeled region is marked by a box.

In Figure 1 the plate thickness is t_p, the hole radius is named a, the hole-hole x and y distances are named L_x and L_y, respectively. In this model the default values are t_p = 1.0 mm, a = 0.5 mm, L_x = 1.4 mm, and L_y = 2.0 mm. A plane wave is incident from one side, modeled using the Background Acoustic Fields feature in the Thermoviscous Acoustics interface.

The transfer impedance Z_{trans}, surface normal impedance Z_n, and absorption coefficient α are defined as

$$Z_{trans} = \frac{\Delta p}{v_{hole}} \qquad Z_n = \frac{p}{\mathbf{v}\cdot\mathbf{n}} \qquad \alpha = 1 - |R|^2 \tag{1}$$

where Δp is the pressure drop across the plate, v is the mean velocity in the perforation hole, $\mathbf{n}$ is the surface normal of the perforate, and R is the reflection coefficient. These expressions are defined as variables in the model (imported from a file), under the **Definitions** node.

The transfer impedance of a perforate (perforated plate) is a well studied problem in acoustics. Several models exist some are pure analytical and some are semi-analytical. In this tutorial model the results obtained in the simulation are compared to a semi-analytical expression for the transfer impedance Z_{trans} presented in Ref. 3 and Ref. 4. The expression combines an expression for the linear losses including viscosity (given in Ref. 1) and an expression for non-linear effects at high levels (given in Ref. 2). The model also includes hole-hole interaction effects (semi-empirical). In this tutorial the COMSOL model only consider linear acoustics. We include thermal and viscous losses as well as hole-hole interaction fully. In Equation 2, retaining the non-linear term is out of interest and shows how it can be added to an analytical expression in COMSOL. The magnitude of that term is very small with the chosen levels. The semi-analytical model is given by the expression

$$\frac{Z_{trans}}{\rho_0 c_0} = \left(t_p + \frac{16a\Psi(\sqrt{\sigma})}{3\pi}\right)\frac{i\omega}{c_0\sigma\left[1-\frac{2}{k_s a}\frac{J_1(k_s a)}{J_0(k_s a)}\right]} + \frac{1.2(1-\sigma^2)}{2c_0(\sigma C_d)^2}U_{rms} \tag{2}$$

$$k_s^2 = -i\rho_0\omega/\mu$$

where ρ_0 is the fluid density, c_0 the speed of sound, μ is the dynamic viscosity, J_n is the Bessel function of the first kind, k_s is the shear (viscous) wave number, U_{rms} the rms acoustic velocity in the hole, σ the porosity, and C_d is the discharge coefficient (a typical value is 0.76). Ψ is the so-called Fok function is defined by

$$\Psi(\sqrt{\sigma}) = 1 - 1.41(\sqrt{\sigma}) + 0.34(\sqrt{\sigma})^3 + 0.07(\sqrt{\sigma})^5 \quad (3)$$

The transfer impedance expression in Equation 2 does not include thermal effects. These can be added by modifying the linear part of the transfer impedance expression. The thermal effects are small for thin perforated plate (in the small $k \cdot t_p$ limit) where shear and viscous losses dominate, see Ref. 5.

Results and Discussion

The total acoustic pressure in the system is depicted in Figure 2 evaluated at 2000 Hz. The acoustic particle velocity is depicted in Figure 3 at two different frequencies (400 and 2000 Hz). From the figure the viscous boundary layer is easily seen as well as its frequency dependence (decreases with increasing frequency). Moreover, the end correction extent is also visualized here, as the part of the fluid moving axially, that extends out of the orifice. In Figure 4 the acoustic temperature variations can be seen. Here, as for the viscous boundary layer, the thermal boundary layer can be visualized.

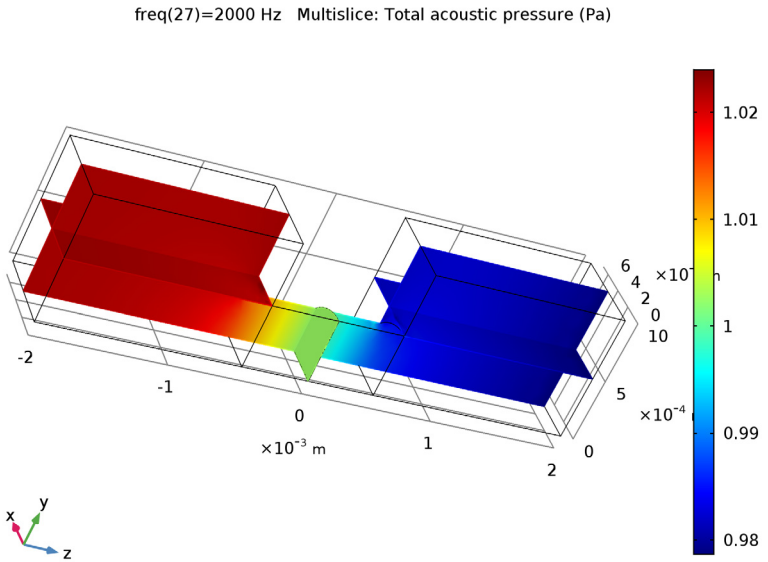


Figure 2: Pressure distribution in the system.

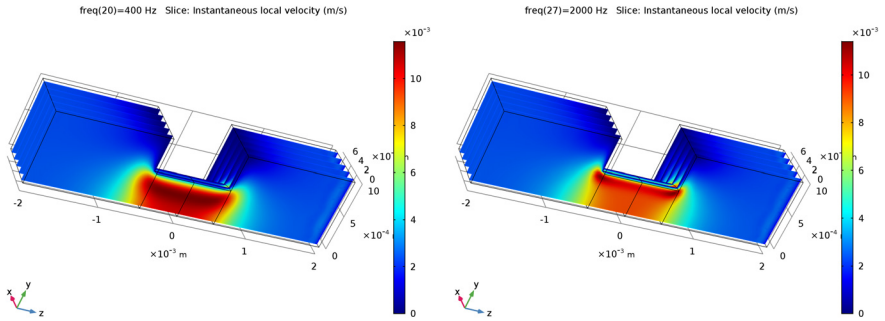


Figure 3: Velocity distribution at 400 Hz and 2000 Hz.

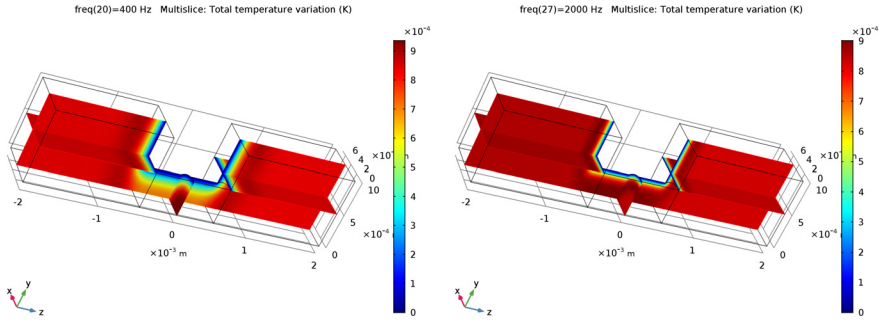


Figure 4: Temperature distribution at 400 Hz and 2000 Hz.

The simulated value of the transfer impedance is depicted in Figure 5 and compared to the semi-analytical model given by Equation 2. The results are seen to coincide well. On the logarithmic y-axis scale there is a constant difference between the curves. This corresponds to a factor multiplied on the linear scale. This could be due to the end correction and the hole-hole interaction modeled with the Fok function. In the present case the holes are not equidistant in all directions. This can also make the simulation results deviate from the analytical assumptions.

The surface normal impedance experienced by a plane wave is depicted in Equation 6 and the absorption coefficient is depicted in Equation 7.

Finally, the instantaneous acoustic particle velocity is depicted using mirror data sets in the last Figure 8.

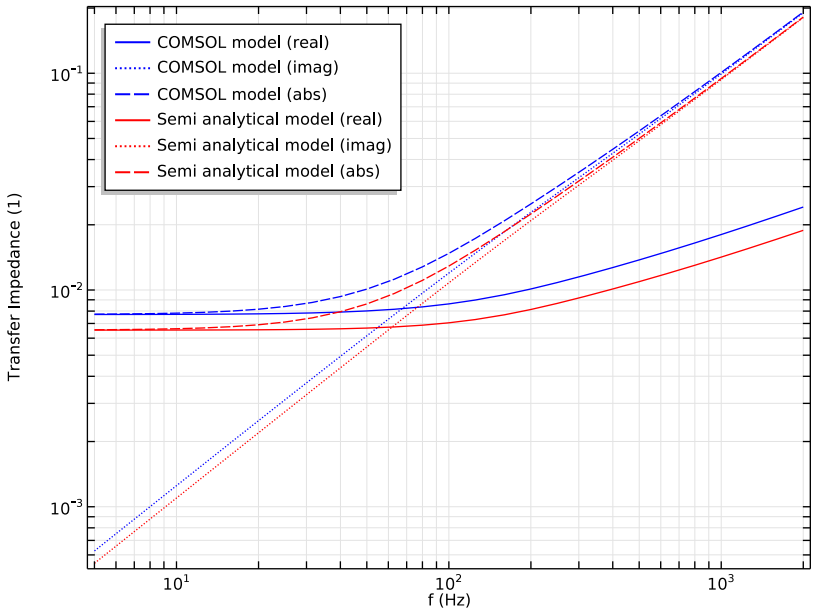


Figure 5: Transfer impedance as function of frequency. Comparing the COMSOL model results and the semi analytical expression.

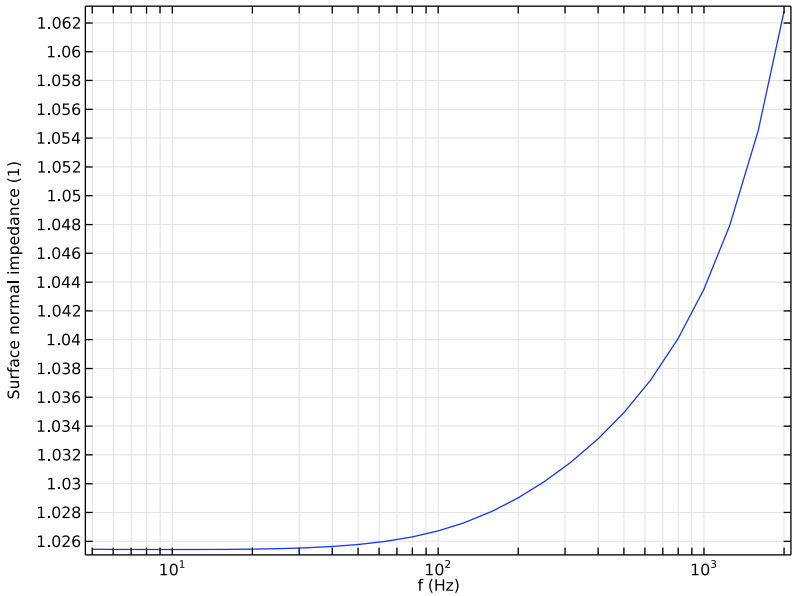


Figure 6: Surface normal impedance of the perforated plate.

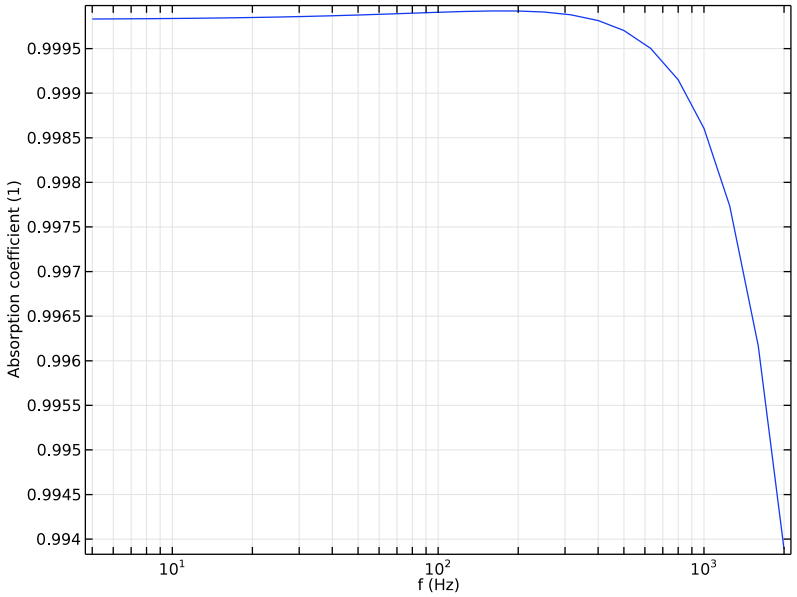


Figure 7: Absorption coefficient of the perforated plate.

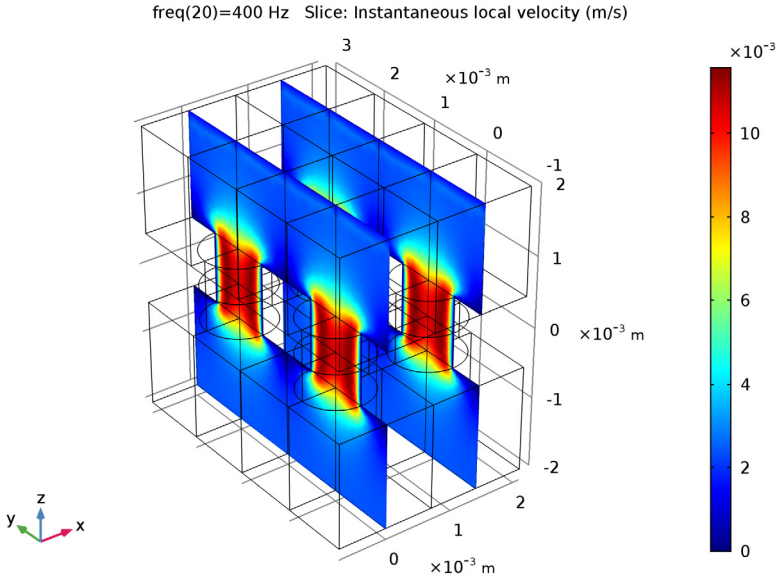


Figure 8: Instantaneous velocity plotted using the symmetry data sets.

Notes About the COMSOL Implementation

In this model the default direct solver has been modified to use an iterative approach with a so-called direct preconditioner. This is one of the predefined iterative solver suggestions generated by COMSOL (it is then simply to enable it). This solver strategy is described in the *Acoustics Module User's Guide* in the *Modeling with Thermoviscous Acoustics* section. The approach is ideal for medium sized pure thermoviscous acoustics models, the iterative solver is both faster and more memory efficient than the default direct solver.

When setting up the mesh, the parameter `dvisc` has been used as a measure of the viscous boundary layer thickness at the maximal study frequency. The viscous boundary layer thickness is given by

$$\delta_{\text{visc}} = \sqrt{\frac{2\mu}{\omega\rho_0}}$$

which at 100 Hz is equal to 0.22 mm for air. Thus the expression `220[um]*sqrt(100[Hz]/fmax)` used in the parameter list for `dvisc`.

In the expression for the reflection coefficient R given under **Definitions > Variables 1** the `down()` operator has been used in order to access the values of the background and scattered field variables on the side of the boundary where the **Background Acoustic Fields** feature is defined. The orientation up/down is relative to the surface normal which can be visualized with an arrow surface plot of the normal vector components `nx`, `ny`, and `nz`.

References

1. I. B. Crandall, "Theory of Vibrating Systems and Sound," D. Van Nostrand Company, New York (1926).

2. T. H. Melling, "The acoustic impedance of perforates at medium and high sound pressure levels," J. Sound Vibration **29**, 1-65 (1973).

3. T. Schultz, F. Liu, L. Cattafesta, M. Sheplak, and M. Jones, "A Comparison Study of Normal-Incidence Acoustic Impedance Measurements of a Perforated Liner", NASA Technical Reports Server LF99-801 (2009).

4. T. Schultz, F. Liu, L. Cattafesta, M. Sheplak, and M. Jones, "A Comparison Study of Normal-Incidence Acoustic Impedance Measurements of a Perforated Liner", 15th AIAA/CEAS Aeroacoustics conference, AIAA 2009-3301.

5. A. W. Nolle, "Small-Signal Impedance of Short Tubes," J. Acoust. Soc. Am. **25**, 32-39 (1952).

Application Library path: `Acoustics_Module/Tutorials/transfer_impedance_perforate`

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 **Acoustics>Thermoviscous Acoustics> Thermoviscous Acoustics, Frequency Domain (ta)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Start by loading the global parameters that define the geometry and mesh parameters.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `transfer_impedance_perforate_parameters.txt`.

GEOMETRY 1

The next task is to set up the geometry of the perforate. The following instructions walk you through the steps of creating a fully parametrized geometry of one quarter of a hole as depicted in Figure 1. The instructions are quite lengthy.

If you wish to skip these steps the geometry sequence can be imported by right clicking the **Geometry** node and select **Insert Sequence**. Browse to your COMSOL installation folder under `Multiphysics>applications>Acoustics_Module>Tutorials` and select `transfer_impedance_perforate.mph`. After this just continue setting up the model from the **Definitions** below.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `a`.

4 In the **Height** text field, type `tp`.

5 Locate the **Position** section. In the **z** text field, type `-tp/2`.

6 Click to expand the **Layers** section. Clear the **Layers on side** check box.

7 Select the **Layers on bottom** check box.

8 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	tp/2

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Lx/2`.

4 In the **Depth** text field, type `Ly/2`.

5 In the **Height** text field, type `Lz+Lpml`.

6 Locate the **Position** section. In the **z** text field, type `tp/2`.

7 Click to expand the **Layers** section. Find the **Layer position** subsection. Clear the **Bottom** check box.

8 Select the **Top** check box.

9 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Lpml

Block 2 (blk2)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `Lx/2`.

4 In the **Depth** text field, type `Ly/2`.

5 In the **Height** text field, type `Lz+Lpml`.

6 Locate the **Position** section. In the **z** text field, type `-(Lz+Lpml+tp/2)`.

7 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Lpml

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

Work Plane 2 (wp2)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Partition Objects 1 (par1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **cyl1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 From the **Work plane** list, choose **Work Plane 1 (wp1)**.

6 Click **Build Selected**.

Partition Objects 2 (par2)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **par1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 Click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

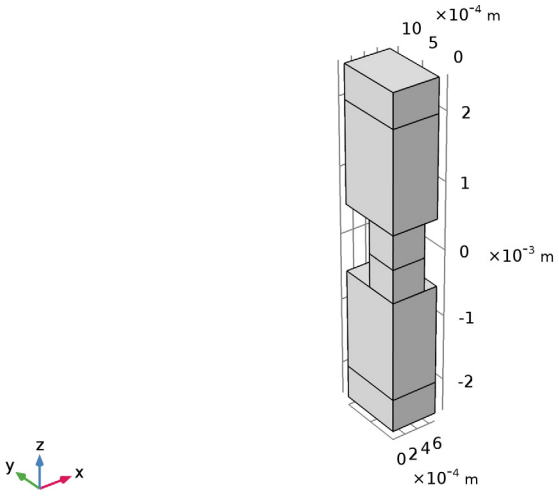
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **par2**, select Domains 1–6 only.

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.



DEFINITIONS

Load the variables that define the transfer impedance, normal surface impedance, and absorption coefficient. The varisbles also define the semi analytical transfer impedance model defined in Equation 2.

1 In the **Model Builder** window, expand the **Definitions** node.

Variables 1

1 Right-click **Component 1 (comp1)>Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `transfer_impedance_perforate_variables.txt`.

Now, first set up integration operators in the two sides of the plate (in and out) as well as in the center of the tube (mid). Create selections for the symmetry planes and the wall. Finally, add the perfectly matched layers (PMLs) that truncate the computational domain.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

4 In the **Settings** window for **Integration**, locate the **Source Selection** section.

5 From the **Geometric entity level** list, choose **Boundary**.

6 Select Boundaries 9 and 21 only.

7 In the **Operator name** text field, type `intop_in`.

Integration 2 (intop2)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 15 and 23 only.

5 In the **Operator name** text field, type `intop_out`.

Integration 3 (intop3)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 12 only.

5 In the **Operator name** text field, type `intop_mid`.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Symmetry` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** check box, to simplify selection and click on one face on each symmetry planes. Alternatively select the following faces.

5 Select Boundaries 1, 2, 4, 5, 7, 8, 10, 11, 13, 14, 16, 17, and 24–31 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Wall` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 20–23 only.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 6 only.

MATERIALS

Add air as the material and set the bulk viscosity to 0.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component 1**.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	0	Pa·s	Basic

Proceed to set up the physics.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

Finally, add an incident plane wave using the **Bakcground Acoustic Fields** feature with the **Plane Wave** option. Since the domain is backed by a PML this configuration will allow reflections from the surface to leave the computational domain.

Background Acoustic Fields 1

1 On the **Physics** toolbar, click **Domains** and choose **Background Acoustic Fields**.

2 Select Domain 2 only.

3 In the **Settings** window for **Background Acoustic Fields**, locate the **Background Acoustic Fields** section.

4 From the **Acoustic field type** list, choose **Plane wave**.

5 In the $|p_b|$ text field, type 1.

6 Specify the $\mathbf{e}_k$ vector as

0	x
0	y
1	z

MESH 1

Now create a mesh that will resolve the acoustic boundary layers. The thickness of the viscous layer at 2000 Hz is defined as a parameter `dvisc`. The mesh should also be structured in the PML regions with at least 8 layers (used for the polynomial stretching type of the PML).

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.

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 `a/3`.

5 In the **Minimum element size** text field, type `dvisc/2`.

6 In the **Resolution of narrow regions** text field, type 4.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2–5 only.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Tetrahedral 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 **Boundary**.

4 From the **Selection** list, choose **Wall**.

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 `dvisc`.

Distribution 1

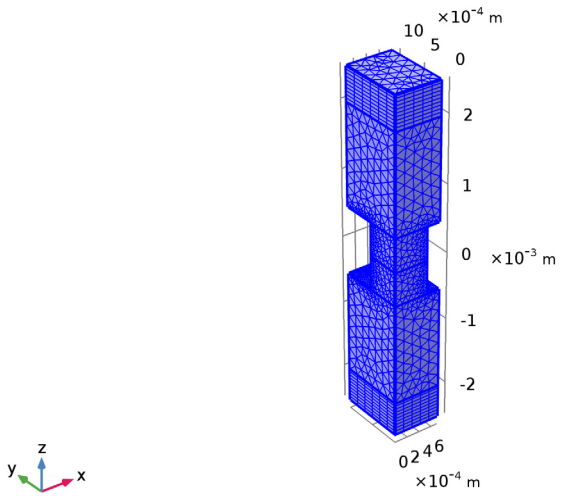
1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `10`.

5 Click **Build All**.



6 Click **Build All**.

STUDY 1

Step 1: Frequency Domain

Proceed to select the frequencies for which to solve the model. Use the built in **ISO preferred frequencies**. In order to see these, first activate the advanced study option view.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, choose **ISO preferred frequencies** from the **Entry method** list.

5 In the **Start frequency** text field, type 5.

6 In the **Stop frequency** text field, type 2000.

7 From the **Interval** list, choose **1/3 octave**.

8 Click **Replace**.

This gives a list with the ISO preferred frequencies between 5 and 2000 Hz with 1/3 octave spacing.

In the next steps set up an iterative solver to solve this thermoviscous acoustics problem. Because of the problem size (not too large) an iterative solver with a direct preconditioner is the best choice over the default direct solver. A discussion of different solver strategies is given in the modeling section under the Thermoviscous Acoustics chapter in the Acoustics Module User's Guide.

Start by generating the default solver, exapnd the nodes, and then enable the interative solver suggestion that uses the direct preconditioner.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (GMRES with Direct Precon.) (ta)** and choose **Enable**.

Step 1: Frequency Domain

On the **Study** toolbar, click **Compute**.

RESULTS

Acoustic Pressure (ta)

Add a selection to the data set to remove plotting in the PML region. Then rotate the figure to produce the pressure, velocity, and temperature plots depicted in Figure 2, Figure 3, and Figure 4.

Note that to analyze the damping inside the PML domains it can be a good idea not to add the selection and then plot the sound pressure level `ta.Lp`.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 2–5 only.

Acoustic Velocity (ta)

1. In the **Model Builder** window, under **Results** click **Acoustic Velocity (ta)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Parameter value (freq (Hz))** list, choose **400**.
4. On the **Acoustic Velocity (ta)** toolbar, click **Plot**.

Temperature Variation (ta)

1. In the **Model Builder** window, under **Results** click **Temperature Variation (ta)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Parameter value (freq (Hz))** list, choose **400**.
4. On the **Temperature Variation (ta)** toolbar, click **Plot**.

Now, create the transfer impedance plot depicted in Figure 5.

1D Plot Group 4

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Transfer Impedance` in the **Label** text field.
3. Click to expand the **Title** section. From the **Title type** list, choose **None**.
4. Locate the **Plot Settings** section. Select the **x-axis label** check box.
5. In the associated text field, type `f (Hz)`.
6. Select the **y-axis label** check box.
7. In the associated text field, type `Transfer Impedance (1)`.
8. Locate the **Axis** section. Select the **x-axis log scale** check box.

9 Select the **y-axis log scale** check box.

10 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Transfer Impedance** 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
real(Ztrans)	1	COMSOL model (real)
imag(Ztrans)	1	COMSOL model (imag)
abs(Ztrans)	1	COMSOL model (abs)

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

5 From the **Color** list, choose **Blue**.

Point Graph 1

1 In the **Model Builder** window, under **Results** right-click **Transfer Impedance** and choose **Point Graph**.

2 Select Point 5 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `real(Ztrans_ana)`.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
Semi analytical model (real)

Point Graph 2

1 Right-click **Transfer Impedance** and choose **Point Graph**.

2 Select Point 5 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `imag(Ztrans_ana)`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Semi analytical model (imag)

Point Graph 3

1 Right-click **Transfer Impedance** and choose **Point Graph**.

2 Select Point 5 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `abs(Ztrans_ana)`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Red**.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Semi analytical model (abs)

10 On the **Transfer Impedance** toolbar, click **Plot**.

Create the surface impedance plot depicted in Figure 6.

1D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Surface Normal Impedance` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `f (Hz)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Surface normal impedance (1)`.

8 Locate the **Axis** section. Select the **x-axis log scale** check box.

9 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Surface Normal Impedance** 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
`abs(Zn)`	1	

4 On the **Surface Normal Impedance** toolbar, click **Plot**.

Next, create the absorption coefficient plot depicted in Figure 7.

1D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Absorption coefficient` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `f (Hz)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Absorption coefficient (1)`.

8 Locate the **Axis** section. Select the **x-axis log scale** check box.

9 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Absorption coefficient** 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
`alpha`	1	

4 On the **Absorption coefficient** toolbar, click **Plot**.

Finally, create a series of mirror data stes to plot the solution in a larger domain (using the symmetries of the model). This will reproduce the plot in Figure 8.

Mirror 3D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

2 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

3 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

4 From the **Data set** list, choose **Mirror 3D 1**.

5 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

Mirror 3D 3

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Data set** list, choose **Mirror 3D 2**.

4 Locate the **Plane Data** section. From the **Plane type** list, choose **General**.

5 From the **Plane entry method** list, choose **Point and normal**.

6 Find the **Point** subsection. In the **x** text field, type `Lx/2`.

7 Find the **Normal** subsection. In the **z** text field, type `0`.

8 In the **x** text field, type `1`.

Mirror 3D 4

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Data set** list, choose **Mirror 3D 3**.

4 Locate the **Plane Data** section. From the **Plane type** list, choose **General**.

5 From the **Plane entry method** list, choose **Point and normal**.

6 Find the **Point** subsection. In the **y** text field, type `Ly/2`.

7 Find the **Normal** subsection. In the **z** text field, type `0`.

8 In the **y** text field, type `1`.

3D Plot Group 7

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Mirror Plot: Velocity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Mirror 3D 4**.

4 From the **Parameter value (freq (Hz))** list, choose **400**.

Slice 1

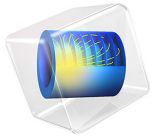
1 Right-click **Mirror Plot: Velocity** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `ta.v_inst`.

4 Locate the **Plane Data** section. In the **Planes** text field, type `2`.

5 On the **Mirror Plot: Velocity** toolbar, click **Plot**.



Ultrasound Flow Meter with Generic Time-of-Flight Configuration

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Knowing the speed of a moving fluid is important in all cases where the fluid is used to transport material or energy. It just so happens that this process is fundamental to such a great part of the tasks we do or interact with in our everyday lives: from industrial applications in production of for instance foods and beverages, drugs, and other chemicals, to personal tasks such as measuring water consumption at your house or determining how much gas was pumped into your car, to medical applications like determining the flow of blood in the veins or the inhalation of air into the lungs. Thus, it should come as no surprise that a large variety of methods have been developed to determine the flow speed.

In the time-of-flight or transit-time method for determining flow speed an ultrasonic signal is transmitted across the main flow in a pipe to non-invasively determine its speed. By transmitting the signal at an angle relative to the main flow, the ultrasound signal will travel faster than the speed of sound if it moves in the direction of the main flow, or slower than the speed of sound if it moves against it. The difference in travel times in the two directions increase linearly with the speed of the main flow. One can therefore infer the speed of the main flow without putting any physical objects into it. Flow meters of this type find many uses particularly in industrial settings where their non-invasiveness is advantageous and examples include abrasive, corrosive and sanitary flows, and in fact all flows in which immersed objects would degrade or significantly obstruct the flow.

The only major limitation to the use of time-of-flight ultrasound flow meters is for fluids containing significant amounts of bubbles or suspended particles. The reason is that the ultrasound will be attenuated significantly by the objects resulting in only a very weak signal arriving at the sensor across the main pipe. To measure the flow speed in these cases, a single combined transmitter and receiver emits the ultrasound, which is reflected off the suspended objects, and the flow speed can be calculated from the Doppler shift between the emitted and received signal.

Here we illustrate how to model a generic wetted transient-time ultrasound flow meter in COMSOL Multiphysics. Wetted ultrasound flow meters have a dedicated signal tube for the ultrasound signal, and the whole device is therefore mounted into the industrial plant. Conversely, portable clamp-on ultrasound flow meters also exist, they are mounted on the outside of the main pipe during operation; this type of flow meter is less accurate due to ultrasound interference resulting from transmitting the signal through the pipe wall.

The model set up here will solve the transient problem of a signal traversing the flow downstream. The signal moving upstream is pre-calculated and imported as data. The difference in arrival times is used to estimate the speed of the main flow. Due to the high frequency of the employed signal (~ 2.5 MHz), we will use the Convected Wave Equation,

Time Explicit physics interface, found under the Ultrasound branch, which is tailored to transient high-frequency situations. The method is very memory efficient and is able to solve the ~7.5 million degrees of freedom (~10^7 DOFs) of this model using around 6 GB of RAM.

Note: This application requires both the Acoustics Module and the CFD Module.

Model Definition

A schematic illustration of the modeled ultrasound flow meter (using one symmetry plane) with annotations and geometry is shown in Figure 1.

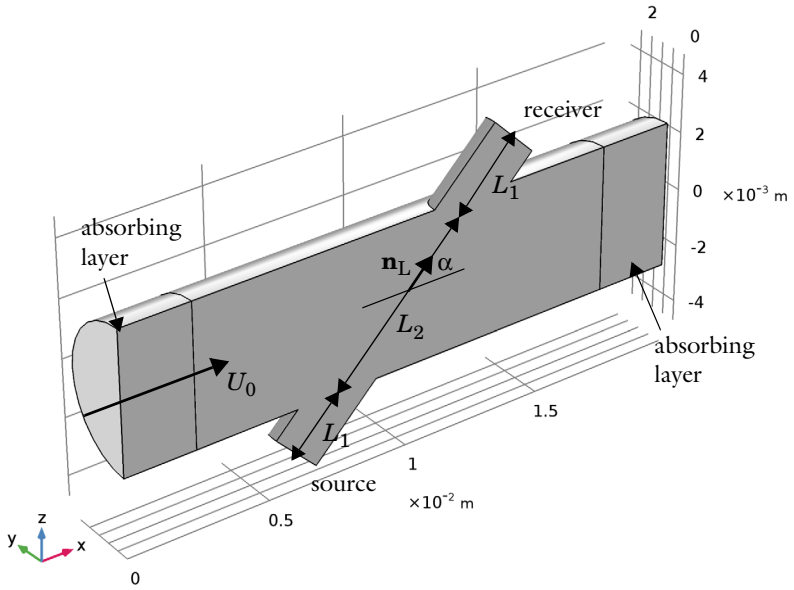


Figure 1: Schematic illustration of ultrasound flow meter. The smaller signal tube of length L_1 is angled relative to the main pipe which contains the flow under investigation.

The flow meter consists of a main pipe and a smaller signal tube (transducer duct) which is placed at some angle α relative to the main pipe. The diameter D of the main pipe is 5 mm while the signal tube has a diameter of 2 mm and is angled at $\alpha = 45^o$ relative to the main pipe. The length of the signal tube is $L = 2L_1 + L_2$. The fluid is taken to be water at

normal atmospheric conditions, and the mean speed of the background flow is taken to be $U_0 = 10$ m/s.

This model solves the full three-dimensional problem. The axial symmetry is exploited for computational efficiency, so only half the channel is modeled.

BACKGROUND FLOW: CFD

The first step in the modeling procedure is to establish a realistic background flow (p_0, $\mathbf{u}_0$). This flow is taken to be stationary, and is obtained using a CFD simulation. The background flow is expected to be turbulent with a Reynolds number (Re) of about $5 \cdot 10^4$ under the operating conditions. The flow is modeled using the Turbulent Flow, k-ε interface of the CFD Module.

The setup of the CFD problem is similar to the one described in the model Flow Through a Pipe Elbow from the CFD Module.

ACOUSTICS: CONVECTED WAVE EQUATION, TIME EXPLICIT

In the acoustics problem, an ultrasound interrogation signal is prescribed at the transducer (signal) end of the signal tube. The time-of-flight method requires that a signal is sent from both ends so in an actual devices both of these ends act as transmitter as well as receiver. In the current model we only model the signal moving downstream while the signal moving upstream is imported from a data file. The data can easily be generated by simply switching the source and receiver boundaries.

The interrogation signal is a sine wave modulated with a Gaussian pulse. It is modeled by prescribing the normal velocity v_n at the source boundary

$$v_n(t) = Ae^{-(f_0(t-3T_0)^2)}\sin(\omega_0 t) \tag{1}$$

where $A = 0.1$ mm is the signal amplitude, $\omega_0 = 2\pi f_0$, $f_0 = 2.5$ MHz, and $T_0 = 1/f_0$. The signal in Equation 1 has a bandwidth of roughly f_0 which is the same as the carrying signal frequency. Using the Convected Wave Equation, Time Explicit interface the mesh needs to have a size roughly half the size of the smallest wavelength λ_0 (largest frequency) that should be resolved. This is the case since the interfaces, per default, uses fourth order shape functions. This combination of mesh size and shape functions gives the optimal performance for the discontinuous Galerkin method, on which the interface is based.

ESTIMATING THE FLOW SPEED

One can construct an estimate for the time it takes the two signals to reach their receivers on the opposite side. This estimate can be used to derive an approximate expression for

the mean speed of the background flow U_0; in fact, this is the working principle behind ultrasound flow meters. That is, measure the time difference ΔT between the two emitted signals and use this time to derive the mean flow speed, possibly corrected by a correction factor as described below.

The background flow is predominantly in the main tube, with very low speeds in the signal tube, see Figure 1. The distance of the signal tube across the main pipe is denoted by L_2, while the part extending on either side has axial length L_1 (the side branches). Thus, the total signal tube length satisfies $L = 2L_1 + L_2$. The direction of the signal tube is denoted by $\mathbf{n}_L$ with the angle between the signal tube and the main pipe being the aforementioned α. Assuming that the background flow in the side branches can be neglected, the time T_1 it takes to traverse is therefore $T_1 = L_1/c_0$. Independently of whether the ultrasound signal is moving upstream or downstream. Now, assuming that the background flow is constant across the main pipe (this can be a reasonable approximation for very turbulent flows) the velocity field can we written $\mathbf{u}_0 = U_0\mathbf{e}_x$. The time T_2 it takes the signal to propagate the distance L_2 depends on whether it is moving upstream or downstream. The time T_2 is, under the assumptions made,

$$T_{2,\,\text{upstream}} = \frac{L_2}{c_0 + \mathbf{n}_L \cdot \mathbf{u}_0} = \frac{L_2}{c_0 + U_0 \cos\alpha}$$
$$T_{2,\,\text{downstream}} = \frac{L_2}{c_0 - \mathbf{n}_L \cdot \mathbf{u}_0} = \frac{L_2}{c_0 - U_0 \cos\alpha}$$

where c_0 is the speed of sound. Therefore, the expected time difference ΔT between the two signals is given by

$$\Delta T = T_{2,\,\text{downstream}} - T_{2,\,\text{upstream}} = \frac{2L_2 U_0 \cos\alpha}{c_0^2 + U_0^2(\cos\alpha)^2}$$

This above expression can be used to estimate the average background mean flow based on a measurement of the time difference ΔT. The expression is

$$U_0 = \frac{L_2}{\Delta T \cos\alpha}\left(1 - \sqrt{1 - \frac{\Delta T^2 c_0^2}{L_2^2}}\right) \tag{2}$$

Note that a critical simplifying assumption for Equation 2 is that the background flow is constant across the tube. In reality, most background flows are not constant across the main pipe. The extreme case is laminar flow where the flow profile will be a parabola, but typical turbulent flow will also exhibit a varying velocity profile.

Results and Discussion

The background mean flow is first simulated using the Turbulent Flow interface from the CFD Module. The velocity amplitude and flow profile are depicted in Figure 2 and Figure 3, respectively.

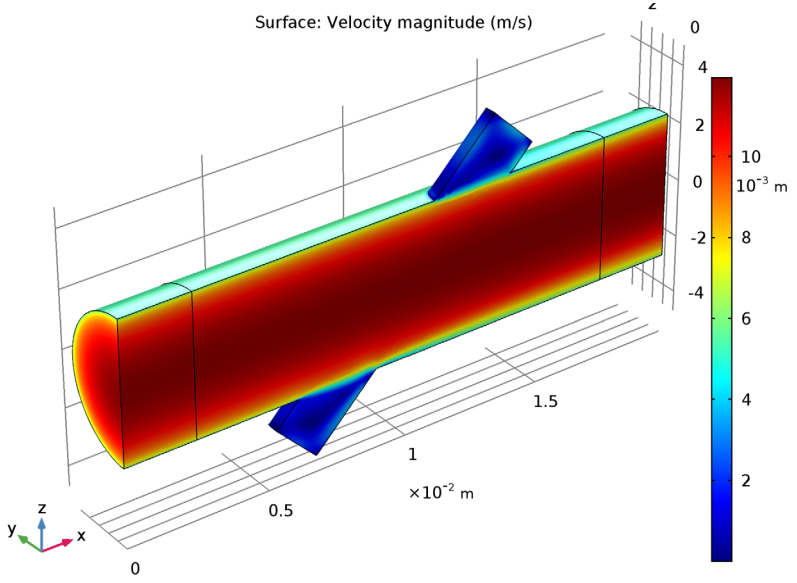


Figure 2: Background mean flow magnitude in the flow meter.

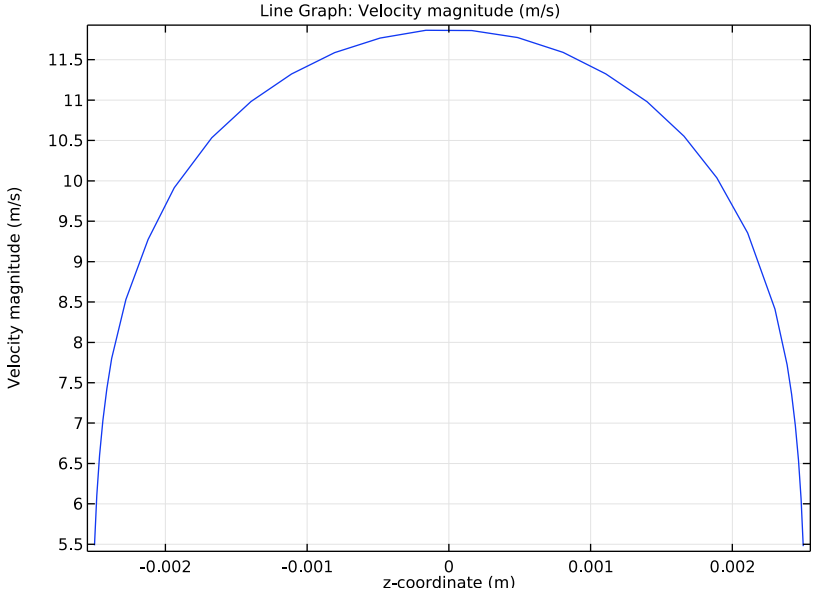


Figure 3: Background mean flow profile, plotted through the cross-section of the main duct.

The acoustic pressure in the symmetry plane of the flow meter is depicted for four different times in Figure 4. This sequence (including all stored times) can be animated within COMSOL under the **Export** node selecting **Animation > Player**. This gives a very intuitive way to visualize the propagating pulse.

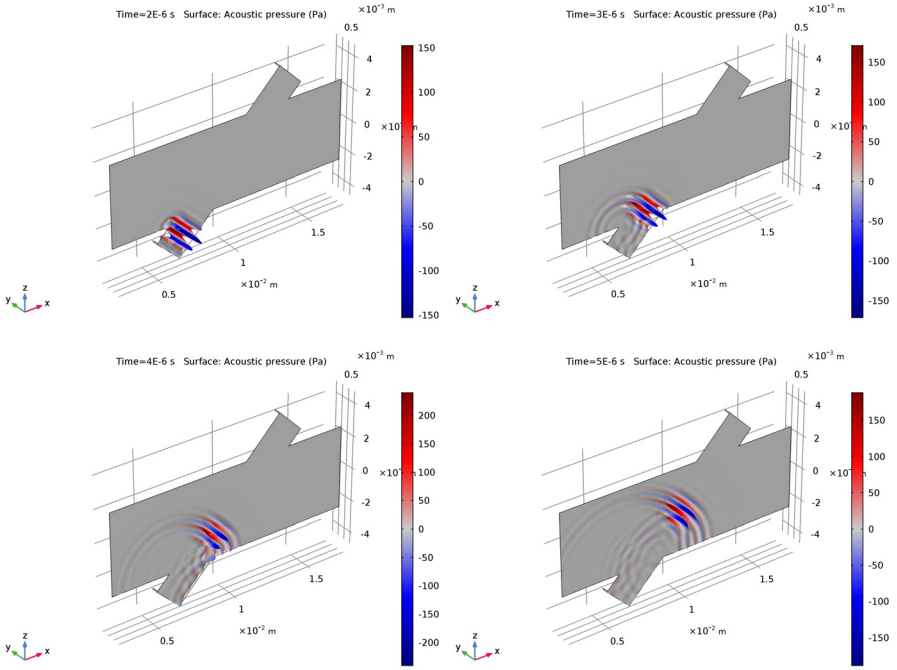


Figure 4: Pressure signal in the symmetry plane of the ultrasound flow meter at 4 times.

The acoustic particle velocity at a time $t = 8 \cdot 10^{-6}$ s just before the pulse reaches the receiver is depicted in Figure 5. The instantaneous intensity amplitude and the intensity vectors (normalized) is depicted in Figure 6 at the same instance as the acoustic velocity in Figure 5. The intensity plot enables a better visualization of the energy flow of the acoustic signal in the flow meter system.

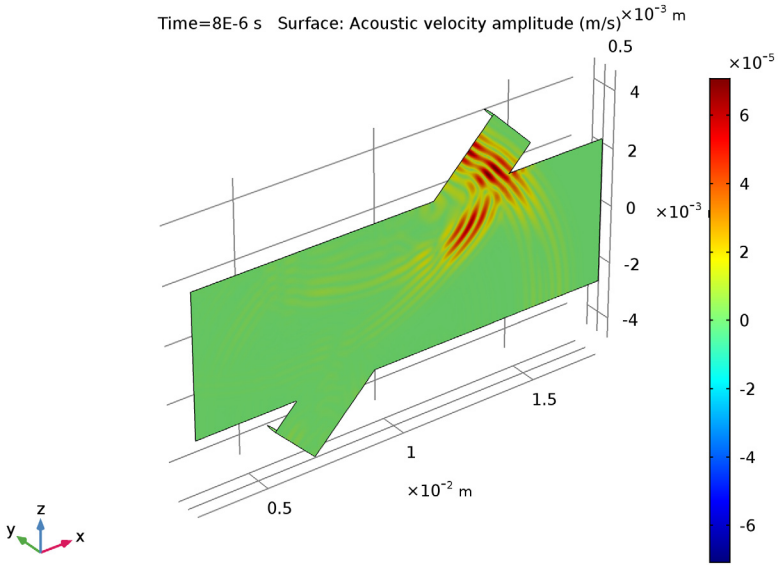


Figure 5: Acoustic particle velocity in the symmetry plane of the flow meter.

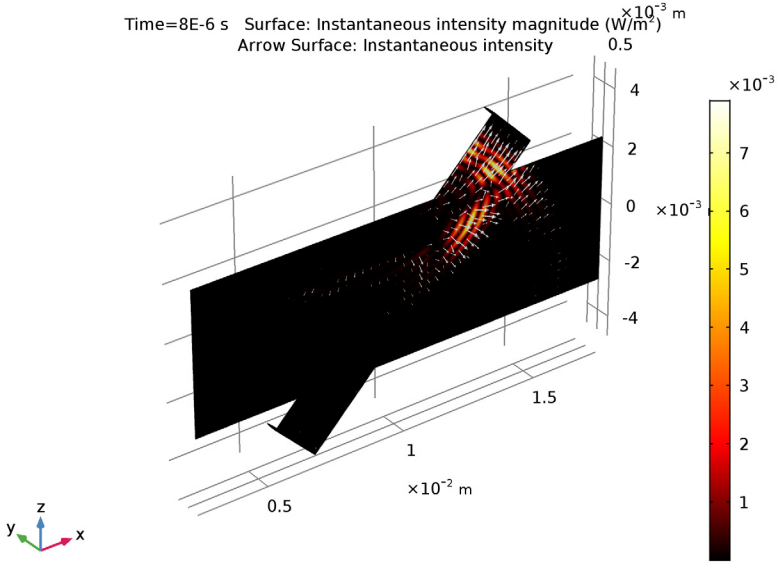


Figure 6: Acoustic instantaneous intensity amplitude (colors) and normalized intensity vectors as white arrows depicted just before the signal reaches the receiver.

In Figure 7 the pressure signal is depicted at time t = $4 \cdot 10^{-6}$ s along the main axis of the transducer tube (see Figure 1). Finally, in Figure 8, one of the important results of this model. The average pressure as recorded on the receiver for the signal moving downstream and the signal moving upstream. The latter data is imported but can be easily generated by switching the source and receiver selections. Using this figure, it is possible to measure the time difference between the two signals. By zooming in on the figure, by visual inspection, it can be found to be $\Delta T_{\text{simulated}} = 4.9 \cdot 10^{-8}$ s (this is stored in the parameter `DT_simulated`).

This simulated time difference can be compared to the expected time difference that can be calculated for the known flow profile $\mathbf{u}_0(\mathbf{x})$ (the flow profile is, of course, not normally known in measurements). The calculated time ΔT_{calc} is given by

$$\Delta T_{\text{calc}} = \int_0^L \left[\frac{1}{c_0 - \mathbf{n}_L \cdot \mathbf{u}_0(\mathbf{x})} - \frac{1}{c_0 + \mathbf{n}_L \cdot \mathbf{u}_0(\mathbf{x})} \right] dl \tag{3}$$

This integral is evaluated under **Results > Derived Values** node and it gives $\Delta T_{\text{calc}} = 4.947 \cdot 10^{-8}$ s which is very close the simulated (visually read) value mentioned above (ΔT_{calc} is stored in the parameter `DT_calc`).

Now, using the simulated time difference $\Delta T_{\text{simulated}}$ in the expression for the estimated mean flow velocity given in Equation 2 you get $U_{0,\text{est}} = 10.75$ m/s. This value is evaluated (both for the simulated and the calculated time differences) under the **Results > Derived Values** node.

FLOW PROFILE CORRECTION FACTOR

The error in estimating U_0 from the simplified model above is one of the central problems with data analysis for ultrasound flow meters, Ref. 1. This is typically described by the flow profile correction factor (FPCF)

$$\text{FPCF} = \frac{\text{Measured average flow speed along tube}}{\text{Actual average mean flow speed}}$$

The issue in determining FPCF theoretically is that the background flow is in general not known; this is especially true for turbulent flows. Moreover, the flow profile across the signal tube likely depends on both the acoustic medium, the flow speed, the pitch angle α as well as design parameters such as the radii of the main pipe and the signal tube.

In this model we do not make any explicit assumptions about the exact velocity profile in the signal tube. Instead, this is determined by the CFD simulation and therefore explicitly include contributions from all the unknown factors listed above. Therefore, COMSOL

Multiphysics can be used to calculate the instrument-specific FPCF, resulting in more accurate instruments across varying flow speeds and geometries. We find for the specific case above

$$\text{FPCF} = \frac{10.75\ \text{m/s}}{10.0\ \text{m/s}} = 1.075$$

and the model could easily be extended to sweep across for instance flow speed and thus obtain FPCF for each situation. Tabulating these values would allow users to obtain the correct result across different working conditions; such tables can easily be included into the data analysis software shipped with the instrument.

Other factors affecting the performance of the flow meter can also be investigated and simulated. For example, the geometry of the signal tubes and how they taper into the main duct could be of interest. Maybe avoiding unnecessary reflections and diffractions. The modification of the emitted acoustic signal by the flow profile (due to gradients in the flow or even density gradients) can also be studied and used to enhance the detection technique, and allowing better prediction of the flow profile.

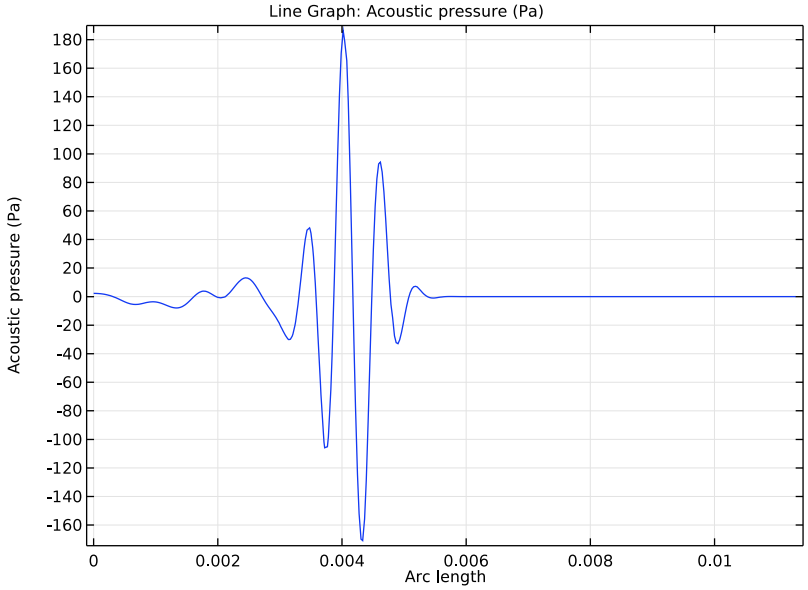


Figure 7: Acoustic pressure signal depicted along the central transducer axis.

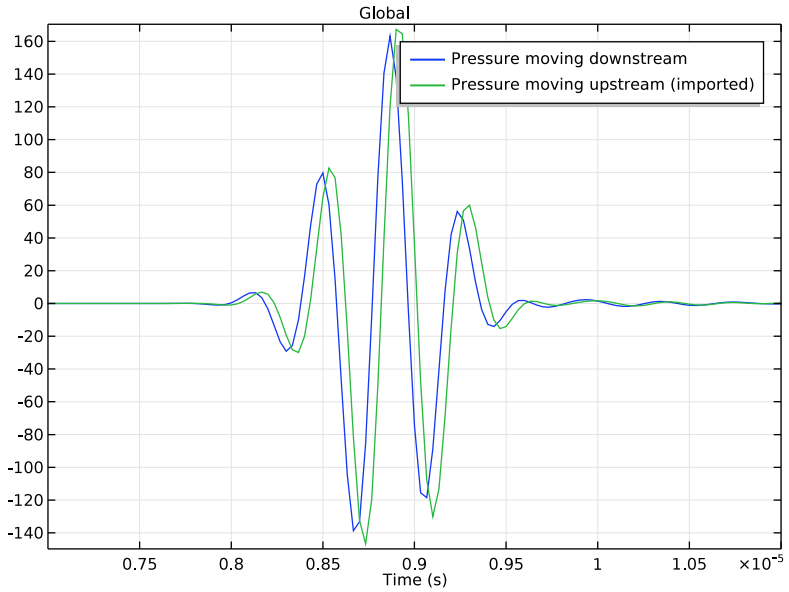


Figure 8: The average acoustic pressure on the receiver for the signal moving downstream (modeled in this tutorial) and the signal moving upstream (data imported).

Reference

1. J. C. Jung and P. H. Seong, "Estimation of the Flow Profile Correction Factor of a Transit-Time Ultrasonic Flow Meter for the Feedwater Flow Measurement in a Nuclear Power Plant," IEEE Transactions on Nuclear Science, vol. 52, 2005.

Application Library path: `Acoustics_Module/Ultrasound/ultrasound_flow_meter_generic`

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 **Fluid Flow>Single-Phase Flow>Turbulent Flow> Turbulent Flow, k-ε (spf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Empty Study**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `ultrasound_flow_meter_generic_parameters.txt`.

GEOMETRY 1

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `D/2`.

4 In the **Height** text field, type `100*D`.

5 Click **Build All Objects**.

MATERIALS

On the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Water, liquid**.

3 Click **Add to Component 1**.

MATERIALS

Water, liquid (mat1)

On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

TURBULENT FLOW, K-ε (SPF)

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `U0`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 3 only.

MESH 1

Distribution 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Mapped**.

2 Right-click **Mapped 1** and choose **Distribution**.

3 Select Boundary 3 only.

4 In the **Settings** window for **Distribution**, locate the **Distribution** section.

5 From the **Distribution properties** list, choose **Predefined distribution type**.

6 In the **Number of elements** text field, type `15`.

7 In the **Element ratio** text field, type `10`.

8 Select the **Reverse direction** check box.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `50`.

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Mapped 1**.

2 In the **Settings** window for **Mapped**, click to expand the **Advanced settings** section.

3 Locate the **Advanced Settings** section. Select the **Adjust evenly distributed edge mesh** check box.

4 Click **Build All**.

DEFINITIONS

General Extrusion 1 (genext1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.

2 In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

5 Locate the **Destination Map** section. In the **r-expression** text field, type `sqrt(y^2+z^2)`.

6 Clear the **z-expression** text field.

7 Locate the **Source** section. Select the **Use source map** check box.

8 Clear the **z?-expression** text field.

ROOT

On the **Home** toolbar, click **Component** and choose **Add Component>3D**.

GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

Cylinder 1 (cyl1)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `D/2`.

4 In the **Height** text field, type `L`.

5 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.5*D

7 Clear the **Layers on side** check box.

8 Select the **Layers on bottom** check box.

9 Select the **Layers on top** check box.

Cylinder 2 (cyl2)

1 On the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `D_transducer/2`.

4 In the **Height** text field, type `L_transducer`.

5 Locate the **Position** section. In the **x** text field, type `L/2`.

6 In the **z** text field, type `-L_transducer/2`.

Rotate 1 (rot1)

1 On the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **cyl2** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.

4 In the **Rotation** text field, type `alpha`.

5 Locate the **Point on Axis of Rotation** section. In the **x** text field, type `L/2`.

6 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **y-axis**.

7 Click **Build All Objects**.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Union**, click **Build All Objects**.

Work Plane 1 (wp1)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Partition Objects 1 (par1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

5 Click **Build All Objects**.

Delete Entities 1 (del1)

1 Right-click **Partition Objects 1 (par1)** and choose **Delete Entities**.

Switch to wireframe rendering to more easily see the domains inside the model, when selecting.

2 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

3 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 On the object **par1**, select Domains 1, 3, 5, 7, 9, and 11 only.

6 Click **Build All Objects**.

Form Composite Domains 1 (cmd1)

1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 On the object **fin**, select Domains 2–5 only.

3 On the **Geometry** toolbar, click **Build All**.

The geometry should look like Figure 1.

ADD MATERIAL

1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Recent Materials>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 2, 6, and 16 only.

5 In the **Label** text field, type `Symmetry`.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Flow Inlet` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 1 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Flow Outlet` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 19 only.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Source` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 10 only.

Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Receiver` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 14 only.

Explicit 6

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Walls` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 3, 4, 7–9, 11, 12, 17, and 18 only.

ADD PHYSICS

1 On the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Recently Used>Turbulent Flow, k-ε (spf)**.

4 Click **Add to Component** in the window toolbar.

ADD PHYSICS

1 Go to the **Add Physics** window.

2 In the tree, select **Acoustics>Ultrasound>Convected Wave Equation, Time Explicit (cwe)**.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

TURBULENT FLOW, K-ε 2 (SPF2)

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Flow Inlet**.

4 Locate the **Velocity** section. Click the **Velocity field** button.

5 Specify the $\mathbf{u}_0$ vector as

`comp1.genext1(w)`	x
`comp1.genext1(u)*cos(atan2(z,y))`	y
`comp1.genext1(u)*sin(atan2(z,y))`	z

6 Locate the **Turbulence Conditions** section. Click the **Specify turbulence variables** button.

7 In the k_0 text field, type `comp1.genext1(k)`.

8 In the ε_0 text field, type `comp1.genext1(ep)`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Flow Outlet**.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

MESH 2

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, type `Mesh - CFD` in the **Label** text field.

3 Click **Build All**.

STUDY 1

1 In the **Settings** window for **Study**, type `Study 1 - CFD` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Stationary

On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

STUDY 1 - CFD

Step 1: Stationary

Deselect the acoustics model (cwe) and the 3D CFD model (spf2).

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for **Turbulent Flow, k-ε 2 (spf2)** and **Convected Wave Equation, Time Explicit (cwe)**.

3 Click to expand the **Mesh selection** section. Locate the **Mesh Selection** section. In the table, enter the following settings:

Geometry	Mesh
Geometry 2	No mesh

Step 2: Stationary 2

1 On the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

Again, deselect the acoustics model (cwe) and now the 2D axisymmetric CFD model (spf).

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Turbulent Flow, k-ε (spf)** and **Convected Wave Equation, Time Explicit (cwe)**.

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - CFD, Stationary**.

Compute the flow solution. This will take about 1 to 2 h (depending on the hardware) and requires 15 GB of RAM.

7 On the **Study** toolbar, click **Compute**.

DEFINITIONS

General Extrusion 2 (genext2)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.
2. In the **Settings** window for **General Extrusion**, type `genext_in` in the **Operator name** text field.
3. Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 5 only.
5. Locate the **Destination Map** section. In the **x-expression** text field, type `y`.
6. In the **y-expression** text field, type `z`.
7. Clear the **z-expression** text field.
8. Locate the **Source** section. Select the **Use source map** check box.
9. In the **x?-expression** text field, type `y`.
10. In the **y?-expression** text field, type `z`.
11. Clear the **z?-expression** text field.

General Extrusion 3 (genext3)

1. Right-click **General Extrusion 2 (genext2)** and choose **Duplicate**.
2. In the **Settings** window for **General Extrusion**, type `genext_out` in the **Operator name** text field.
3. Locate the **Source Selection** section. Click **Clear Selection**.
4. Select Boundary 15 only.

Create an average operator to be used in postprocessing to calculate the average pressure at the receiver.

Average 1 (aveop1)

1. On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.
2. In the **Settings** window for **Average**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Receiver**.

Now, set up the absorbing layers (sponge layers) used to truncate the computational domain.

Absorbing Layer 1 (ab1)

1. On the **Definitions** toolbar, click **Absorbing Layer**.

2 Select Domains 1 and 3 only.

Now, disable the **Absorbing Layer** in the CFD study if you need to run it again. The CFD physics does not support the scaling associated with an absorbing layer.

Note that if you need to work with the CFD Mesh or the CFD Physics you need to manually disable the **Absorbing Layer** in the **Model Builder** tree.

STUDY 1 - CFD

Step 2: Stationary 2

1 In the **Model Builder** window, under **Study 1 - CFD** click **Step 2: Stationary 2**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 2 (comp2)>Definitions> Absorbing Layer 1 (ab1)**.

5 Click **Disable**.

CONVECTED WAVE EQUATION, TIME EXPLICIT (CWE)

On the **Physics** toolbar, click **Turbulent Flow, k-ε 2 (spf2)** and choose **Convected Wave Equation, Time Explicit (cwe)**.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

2 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

Convected Wave Equation Model 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Convected Wave Equation, Time Explicit (cwe)** click **Convected Wave Equation Model 1**.

2 In the **Settings** window for **Convected Wave Equation Model**, locate the **Model Input** section.

3 From the p_0 list, choose **Absolute pressure (spf2)**.

4 From the $\mathbf{u}_0$ list, choose **Velocity field (spf2)**.

5 Locate the **Fluid Properties** section. From the ρ_0 list, choose **Density (spf2/fp1)**.

6 In the **Model Builder** window, click **Convected Wave Equation, Time Explicit (cwe)**.

Convected Wave Equation Model 2

1 On the **Physics** toolbar, click **Domains** and choose **Convected Wave Equation Model**.

2 Select Domain 1 only.

3 In the **Settings** window for **Convected Wave Equation Model**, locate the **Model Input** section.

4 In the p_0 text field, type `genext_in(spf2.pA)`.

5 Specify the $\mathbf{u}_0$ vector as

`genext_in(u2)`	x
`genext_in(v2)`	y
`genext_in(w2)`	z

6 Locate the **Fluid Properties** section. From the ρ_0 list, choose **User defined**. In the associated text field, type `genext_in(spf2.rho)`.

Convected Wave Equation Model 3

1 On the **Physics** toolbar, click **Domains** and choose **Convected Wave Equation Model**.

2 Select Domain 3 only.

3 In the **Settings** window for **Convected Wave Equation Model**, locate the **Model Input** section.

4 In the p_0 text field, type `genext_out(spf2.pA)`.

5 Specify the $\mathbf{u}_0$ vector as

`genext_out(u2)`	x
`genext_out(v2)`	y
`genext_out(w2)`	z

6 Locate the **Fluid Properties** section. From the ρ_0 list, choose **User defined**. In the associated text field, type `genext_out(spf2.rho)`.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

Acoustic Impedance 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Acoustic Impedance**.

2 Select Boundaries 1 and 19 only.

Define the driving signal as given in Equation 1.

GLOBAL DEFINITIONS

Analytic 1 (an1)

1. On the **Home** toolbar, click **Functions** and choose **Global>Analytic**.
2. In the **Settings** window for **Analytic**, type `vn` in the **Function name** text field.
3. Locate the **Definition** section. In the **Expression** text field, type `A*sin(omega0*t)*exp(-(f0*(t-3*T0))^2)`.
4. In the **Arguments** text field, type `t`.
5. Locate the **Units** section. In the **Arguments** text field, type `s`.
6. In the **Function** text field, type `m/s`.

CONVECTED WAVE EQUATION, TIME EXPLICIT (CWE)

Normal Velocity 1

1. On the **Physics** toolbar, click **Boundaries** and choose **Normal Velocity**.
2. In the **Settings** window for **Normal Velocity**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Source**.
4. Locate the **Normal Velocity** section. In the $v_n(t)$ text field, type `vn(t)`.

COMPONENT 2 (COMP2)

Mesh 3

On the **Mesh** toolbar, click **Add Mesh**.

MESH 3

1. In the **Settings** window for **Mesh**, type `Mesh - Acoustics` in the **Label** text field.
2. Right-click **Component 2 (comp2)>Meshes>Mesh - Acoustics** and choose **Free Tetrahedral**.

MESH - ACOUSTICS

Size

1. In the **Settings** window for **Size**, locate the **Element Size** section.
2. Click the **Custom** button.
3. Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambda0/1.5`.

4 In the **Minimum element size** text field, type `lambda0/2`.

With this setting you get a bit more than 1.5 elements per wavelength at the frequency `f0`. This may need to be increased to, for example, `lambda0/2` in order to well resolve the small frequency content above f0, that exists in the Gaussian pulse.

Free Tetrahedral 1

When modeling using physics interfaces that are based on the DG method, it is important to avoid small mesh elemnts, since they control the time steps taken by the solver. In order to avoid this use the **Element Quality Optimization** functionality available for the tetrahedral mesh. This step is very important.

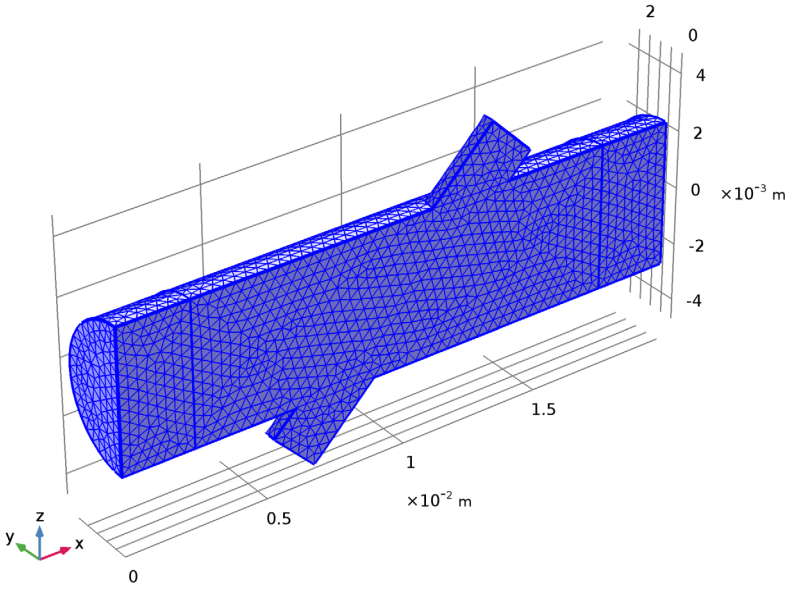
1 In the **Model Builder** window, under **Component 2 (comp2)>Meshes>Mesh - Acoustics** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, click to expand the **Element quality optimization** section.

3 Locate the **Element Quality Optimization** section. Select the **Avoid too small elements** check box.

4 From the **Optimization level** list, choose **High**.

5 Click **Build All**.



Before solving, create an interpolation function and import data for a signal moving the opposite way (upstream) to the signal modeled here (downstream). You will use that data in postprocessing. The data was generated by switching the source and receiver selections and running the model.

GLOBAL DEFINITIONS

Interpolation 1 (int1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `ultrasound_flow_meter_generic_upstream_signal.txt`.

If you want to import the data into the model, and not rely on the .txt file, click **Import**.

6 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
`p_upstream`	1

7 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

8 Locate the **Units** section. In the **Arguments** text field, type `s`.

9 In the **Function** text field, type `Pa`.

10 Right-click **Interpolation 1 (int1)** and choose **Build All**.

ADD STUDY

1 On 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 **Preset Studies**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Turbulent Flow, k-ε (spf)** and **Turbulent Flow, k-ε 2 (spf2)**.

5 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies> Time Dependent**.

6 Click **Add Study** in the window toolbar.

7 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1 In the **Settings** window for **Study**, type `Study 2 - Acoustics` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

STUDY 2 - ACOUSTICS

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2 - Acoustics** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Times** text field, type `range(0,T0/12,30*T0)`.

This setting saves the solution 12 times per period and runs the model for 30 periods. The setting only influences when to store the solution (and thus the file size). The internal time steps taken by the solver are automatically controlled by COMSOL to fulfill the appropriate CFL condition.

4 Click to expand the **Values of dependent variables** section. Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1 - CFD, Stationary 2**.

7 Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

8 Under **Selections**, click **Add**.

9 In the **Add** dialog box, in the **Selections** list, choose **Symmetry**, **Source**, and **Receiver**.

10 Click **OK**.

The amount of data generated in this model is large. A good way to restrict the file size is to only store the data on desired selections. Here the solution is only stored on the symmetry plane, and on the source and receiver faces.

Check that the correct mesh is used for the transient acoustic simulation.

11 In the **Settings** window for **Time Dependent**, click to expand the **Mesh selection** section.

Now solve the convected wave equation (acoustics) model. This will take around 5h (depending on the hardware) and will require about 9 GB of RAM. Remember that the model contains arround 7.5 million degrees of freedom (DOFs).

12 On the **Home** toolbar, click **Compute**.

Before generating any plots and looking at results, clean up the data sets. Three of them can be deleted, for example, the data set of the acoustic solution on the 2D axisymmetric geometry is not of interest (and has no solutions in it).

RESULTS

Data Sets

In the **Model Builder** window, expand the **Results** node.

Study 1 - CFD/Solution 1 (1) (sol1)

In the **Model Builder** window, expand the **Data Sets** node.

Study 2 - Acoustics/Solution 3 (5) (sol3)

1 Right-click **Study 1 - CFD/Solution 1 (1) (sol1)** and choose **Delete**.

2 In the **Model Builder** window, under **Results>Data Sets** right-click **Study 1 - CFD/Solution Store 1 (4) (sol2)** and choose **Delete**.

3 Right-click **Study 2 - Acoustics/Solution 3 (5) (sol3)** and choose **Delete**.

It is good practice to add the same selections to the result data set as the selections chosen in the **Store fields in output**. For postprocessing purposes we will only select the symmetry plane, source, and receiver in the Acoustics data set (not selecting the absorbing layer).

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 6, 10, and 14 only.

Finally, create a data set that is a cut line following the axis of the transducer duct.

Cut Line 3D 1

1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2 - Acoustics/Solution 3 (sol3)**.

4 Locate the **Line Data** section. From the **Line entry method** list, choose **Point and direction**.

5 Find the **Point** subsection. In the **x** text field, type `L/2`.

6 Find the **Direction** subsection. In the **x** text field, type `nLx`.

7 In the **y** text field, type `nLy`.

8 In the **z** text field, type `nLz`.

9 Select the **Snap to closest boundary** check box.

In the following postprocessing the **Resolution** is increased under the **Quality** section in all acoustics plots. This is because the shape functions used for the convected wave equation interface are 4th order per default. So in order to properly represent the solution, the resolution needs to be increased.

3D Plot Group 1

1 On the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Background Flow Velocity` in the **Label** text field.

Surface 1

1 Right-click **Background Flow Velocity** and choose **Surface**.

2 On the **Background Flow Velocity** toolbar, click **Plot**.

The background mean flow velocity amplitude plotted on the surface of the geometry should look like Figure 2.

1D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Background Flow Velocity Profile` in the **Label** text field.

Line Graph 1

1 Right-click **Background Flow Velocity Profile** and choose **Line Graph**.

2 Select Edge 7 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 `z`.

6 On the **Background Flow Velocity Profile** toolbar, click **Plot**.

The background mean flow profile is depicted in Figure 3.

3D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 - Acoustics/ Solution 3 (sol3)**.

Surface 1

1 Right-click **Acoustic Pressure** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `p3`.

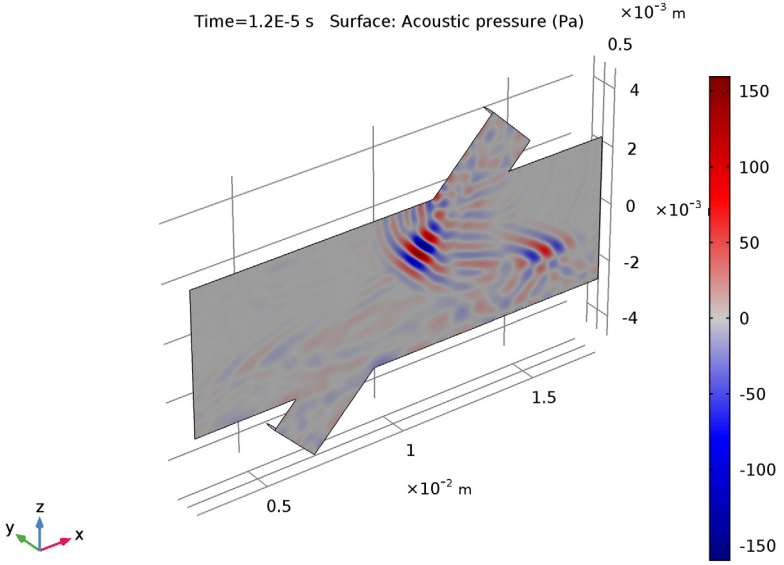
4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

5 Select the **Symmetrize color range** check box.

6 Click to expand the **Quality** section. From the **Resolution** list, choose **Custom**.

7 In the **Element refinement** text field, type `6`.

8 On the **Acoustic Pressure** toolbar, click **Plot**.



Add a deformation to better visualize the nature of the waves propagating in the symmetry plane.

9 In the **Model Builder** window, expand the **Surface 1** node.

Deformation 1

1 Right-click **Results>Acoustic Pressure>Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **x component** text field, type `0`.

4 In the **y component** text field, type `p3`.

5 In the **z component** text field, type `0`.

6 On the **Acoustic Pressure** toolbar, click **Plot**.

The first plot that is shown is for the last stored time, at `30*T0`. Change the **Time** selection to see the solution at other instances. Some are shown in Figure 4.

Acoustic Pressure

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **2E-6**.

4 On the **Acoustic Pressure** toolbar, click **Plot**.

3D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Velocity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 - Acoustics/ Solution 3 (sol3)**.

4 From the **Time (s)** list, choose **8E-6**.

Surface 1

1 Right-click **Acoustic Velocity** 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 2>Convected Wave Equation, Time Explicit>Acceleration and velocity>cwe.v_inst - Acoustic velocity amplitude**.

3 Locate the **Coloring and Style** section. Select the **Symmetrize color range** check box.

4 Locate the **Quality** section. From the **Resolution** list, choose **Custom**.

5 In the **Element refinement** text field, type `6`.

6 On the **Acoustic Velocity** toolbar, click **Plot**.

The acoustic velocity amplitude is depicted in Figure 5. Once again, change the **Time** selection to study the wave propagation.

Proceed to plot the instantaneous intensity as depicted in Figure 6.

3D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Intensity` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 - Acoustics/ Solution 3 (sol3)**.

4 From the **Time (s)** list, choose **8E-6**.

Surface 1

1 Right-click **Acoustic Intensity** 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 2>Convected Wave Equation, Time Explicit>Intensity>cwe.Ii_mag - Instantaneous intensity magnitude**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

4 Locate the **Quality** section. From the **Resolution** list, choose **Custom**.

5 In the **Element refinement** text field, type `6`.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** right-click **Acoustic Intensity** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2> Convected Wave Equation, Time Explicit>Intensity>cwe.Iix,...,cwe.Iiz - Instantaneous intensity**.

3 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

4 In the **Range quotient** text field, type `400`.

5 In the **Number of arrows** text field, type `1000`.

6 From the **Color** list, choose **White**.

7 On the **Acoustic Intensity** toolbar, click **Plot**.

Now, proceed to plot the pressure pulse along the central transducer duct axis in a 1D graph. The figure should look like Figure 7.

1D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure Along Transducer Axis` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 3D 1**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, select **4E-6**.

Line Graph 1

1 Right-click **Pressure Along Transducer Axis** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `p3`.

4 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

5 On the **Pressure Along Transducer Axis** toolbar, click **Plot**.

The final graph to create shows the average pressure signal at the receiver for the pulse moving downstream (the model simulated here). The figure also includes the pressure from a signal moving upstream, as taken from the imported data. After zooming, the graph should look like Figure 8.

1D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure Signal at Receivers` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2 - Acoustics/ Solution 3 (sol3)**.

Global 1

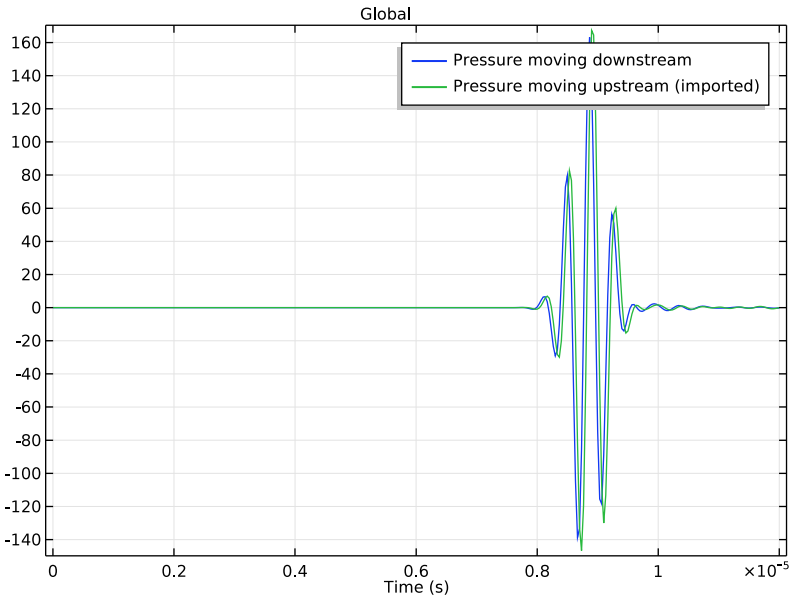
1 Right-click **Pressure Signal at Receivers** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`aveop1(p3)`	`Pa`	`Pressure moving downstream`
`p_upstream(t)`	`Pa`	`Pressure moving upstream (imported)`

4 On the **Pressure Signal at Receivers** toolbar, click **Plot**.



Finally, calculate the integral expression given in Equation 3 under the **Derived Values** node. Here you can also calculate the predicted mean flow speeds from Equation 2. The results are discussed in the main results section.

Line Integration 1

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration> Line Integration**.

2 In the **Settings** window for **Line Integration**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 3D 1**.

4 From the **Time selection** list, choose **First**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`1/(cwe.c0-(nLx*u2+nLy*v2+nLz*w2))-1/(cwe.c0+(nLx*u2+nLy*v2+nLz*w2))`		

6 Click **Evaluate**.

Global Evaluation 1

1 On the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

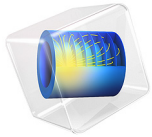
3 From the **Data set** list, choose **Study 2 - Acoustics/Solution 3 (sol3)**.

4 From the **Time selection** list, choose **First**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`L2/(DT_calc*cos(alpha))*(1-sqrt(1-DT_calc^2*c0^2/L2^2))`		
`L2/(DT_simulated*cos(alpha))*(1-sqrt(1-DT_simulated^2*c0^2/L2^2))`		

6 Click **Evaluate**.



Focused Ultrasound Induced Heating in Tissue Phantom

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

When an ultrasound beam passes through a volume of tissue, some of the energy of the primary acoustic field is absorbed locally by the tissue and turned into heat. This results in a temperature increase whose magnitude is a function of the physical properties of the medium (acoustic absorption coefficient, density, and specific heat), the properties of the ultrasound device (beam geometry), and the frequency and time-averaged acoustic intensity of the acoustic field; see Ref. 1 and Ref. 2. The actual temperature rises that can be obtained also depend on the conduction and convection properties of the tissue involved, for example, the blood perfusion rate of the tissue.

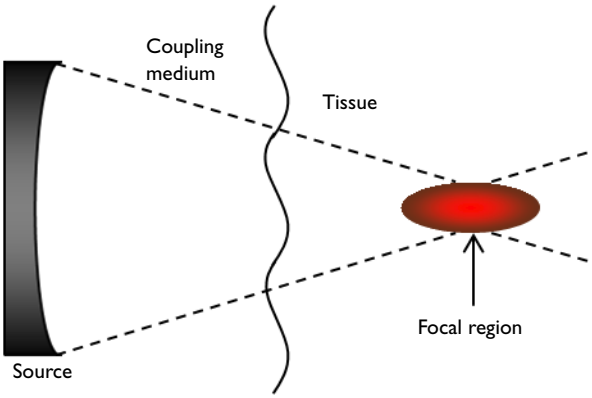


Figure 1: Focused ultrasound makes selective and targeted heating possible: heating of tissues lying within the focal volume can be achieved with minimal damage to nearby healthy tissue and other structures lying elsewhere in the path of the beam.

Note: This application requires the Acoustics Module and the Heat Transfer Module.

Therapeutic applications of ultrasound typically involve focused beams, which allow the ultrasound energy to be directed into a small area within the tissue region that needs the treatment. Moreover, heating of tissues lying within the focal volume can be achieved with minimal damage to nearby healthy tissue and other structures lying elsewhere in the path of the beam, as shown in Figure 1.

Depending on the dosage parameters—that is, field intensity and exposure time—the clinical applications of focused ultrasound (FU) are generally grouped into two categories, namely "ultrasound hyperthermia" and "focused ultrasound surgery" (FUS); see Ref. 3. Generally, during hyperthermia applications, tissues are exposed to ultrasound for long

periods (from 10 to 60 minutes) at lower intensity levels, such that the irradiated tissue temperature is elevated and maintained at 41 °C to 45 °C during the therapy. The biological change thus induced can be reversed. In contrast, focused ultrasound surgery utilizes intense, relatively short bursts (0.1 s to 30 s) to induce irreversible changes in the focal tissue volume. In this type of applications nonlinear acoustic effects and acoustic cavitation usually play essential roles; the tissue temperature in the focal zone can reach 70 °C to 90 °C within a few seconds. This technique is also known as ultrasound ablation.

The thermal effect of focused ultrasound also brings concerns about possible harmful effects of diagnostic ultrasound, especially in obstetrical examinations when the fetus is exposed to ultrasound. The U.S. Food and Drug Administration has set up regulations on the maximum thermal index, a dosage parameter reflecting the combined effect of temperature and exposure time that is an estimate of risk from heat (Ref. 4). Diagnostic ultrasound systems now come with displays with the Thermal Index (TI) and the Mechanical Index (MI), the other estimate of risk from the nonthermal effects of ultrasound in order to meet the U.S. government's regulations. Safety in the use of ultrasound has also been addressed extensively in the academic field; see, for example, Ref. 5, Ref. 6, and Ref. 7.

The current model is inspired by the experiments to measure focused ultrasound induced heating in a tissue phantom from Ref. 8. The model uses the same geometry and material properties as in Ref. 8. The model exemplifies how to use COMSOL Multiphysics to model tissue heating induced by focused ultrasound. The simulation results are compared to the experimental data in the reference.

Model Definition

Figure 2 shows the geometry simulated in this model. Both the tissue phantom and the acoustic transducer are immersed in water. The transducer is bowl shaped with a focal length of 62.64 mm, an aperture of 35 mm in radius, and a hole of 10 mm in radius in the center. The tissue phantom has the shape of a cylinder with 53.6 mm in radius and 80.5 mm in length. The tissue phantom and the transducer are arranged coaxially so the model can be defined as being 2D axisymmetric.

The transducer is driven at the frequency of 1 MHz. It is turned on for 1 second and then turned off to let the tissue phantom cool down completely in water. The model thus solves

for the heating of the tissue phantom for 1 second and then simulates the cooling process after the acoustic source is turned off.

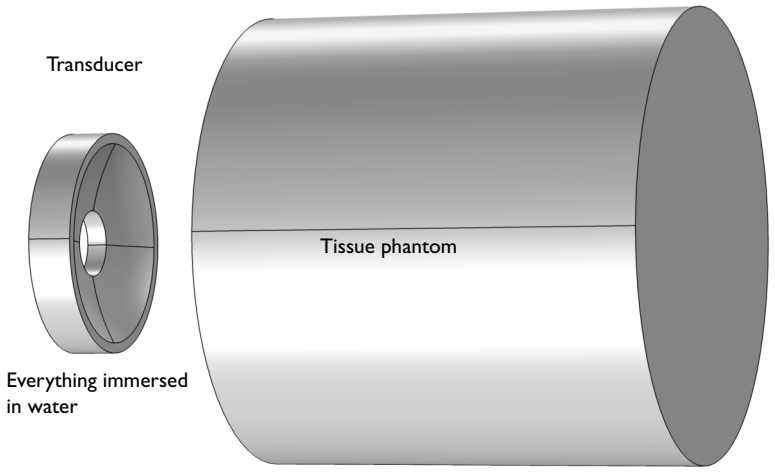


Figure 2: The geometry of the acoustic transducer and tissue phantom. The transducer is bowl shaped with a hole in the center. Both the tissue phantom and the transducer are immersed in water. The axisymmetric geometry allows for a 2D axisymmetric simulation.

The model uses the Pressure Acoustics, Frequency Domain interface to model the stationary acoustic field in the water and the tissue domain to obtain the acoustic intensity distribution in the tissue phantom. The absorbed acoustic energy is calculated and used as the heat source for the Bioheat Transfer interface model. Because the acoustic focal region (like the heated area) is much smaller than the size of the tissue phantom, the thermal simulation is performed only in the tissue domain.

The wave equation solved is the homogeneous Helmholtz equation in 2D axisymmetric cylindrical coordinates:

$$\frac{\partial}{\partial r}\left[-\frac{r}{\rho_c}\left(\frac{\partial p}{\partial r}\right)\right] + r\frac{\partial}{\partial z}\left[-\frac{1}{\rho_c}\left(\frac{\partial p}{\partial z}\right)\right] - \left[\left(\frac{\omega}{c_c}\right)^2\right]\frac{rp}{\rho_c} = 0 \qquad (1)$$

Here r and z are the radial and axial coordinates, p is the acoustic pressure, and ω is the angular frequency. The density, ρ_c, and the speed of sound, c_c, are complex-valued to account for the material's damping properties.

Using Equation 1 involves the assumption that the acoustic wave propagation is linear and also that the amplitude of shear waves in the tissue domain are much smaller than that of the pressure waves. Nonlinear effects and shear waves are therefore neglected.

Given the acoustic pressure field the acoustic intensity field is readily derived. The heat source Q for thermal simulation, given in the plane-wave limit, is then calculated as

$$Q = 2\alpha_{\mathrm{ABS}} I = 2\alpha_{\mathrm{ABS}} \left| \mathrm{Re}\left(\frac{1}{2} p \mathbf{v}\right) \right| \tag{2}$$

where α_{ABS} is the acoustic absorption coefficient, I is the acoustic intensity magnitude, p is the acoustic pressure, and $\mathbf{v}$ is the acoustic particle velocity vector. In COMSOL, the intensity is a derived variable where the magnitude can be accessed as `acpr.I_mag`. Likewise the dissipated power density (for plane waves) is defined as `acpr.Q_pw`. The heat source Q is thus readily calculated once the acoustic field is solved.

Note: For further details about the intensity variables choose **Help>Documentation** and then search for the string *special variables in the Acoustics Module*. Two sections exist here; one describing the intensity variables and one describing power dissipation variables.

Inserting the volumetric acoustic heat source into the Pennes' Bioheat Transfer equation to model heat transfer within biological tissue gives

$$\rho C_{\mathrm{p}} \frac{\partial T}{\partial t} = \nabla \cdot (\mathrm{k} \nabla T) - \rho_{\mathrm{b}} C_{\mathrm{b}} w_{\mathrm{b}} (T - T_{\mathrm{b}}) + Q + Q_{\mathrm{met}}$$

where T is the temperature, ρ is the density, C_{p} is the specific heat, k is the thermal conductivity, ρ_{b} is the density of blood, C_{b} is the specific heat of blood, w_{b} is the blood perfusion rate, T_{b} is the temperature of the blood, Q is the heat source (the absorbed ultrasound energy calculated from Equation 2), and Q_{met} is the metabolic heat source.

In this model, assume that the tissue properties do not change when the temperature rises. Blood perfusion is also neglected this can be added.

Figure 3 shows the model geometry and material domains defined in the model. Four cylindrical perfectly matched layers (PMLs) (`r1`-`r4`) and one spherical PML (`c1`) are used

to absorb the outgoing waves. The pressure acoustics simulation is performed in all domains while the heat transfer model is only applied in the tissue phantom domain.

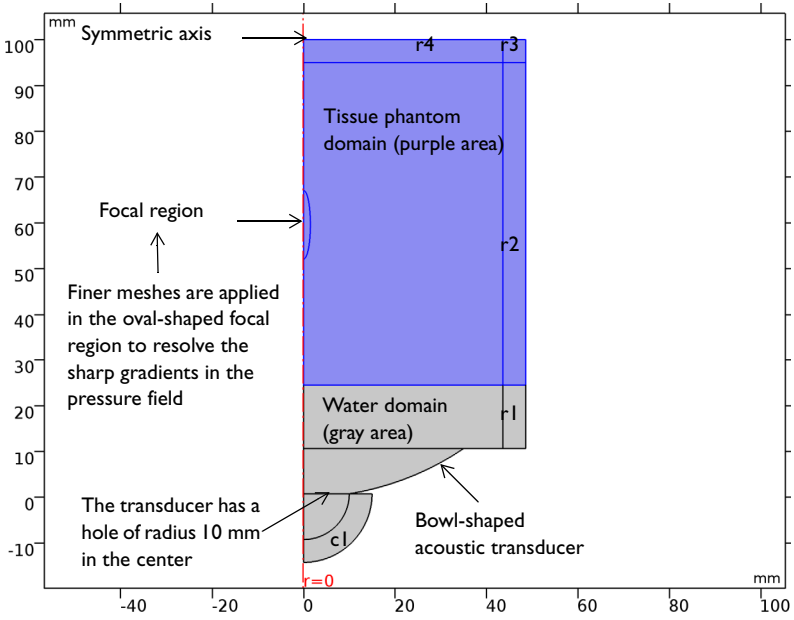


Figure 3: Model geometry. Water domains are shown in blue and tissue phantom domains are shown in purple. Four cylindrical PMLs (`r1-r4`*) and one spherical PML (*`c1`*) are used to absorb the outgoing waves.*

To accurately resolve the sharp pressure gradient in the focal region, the model uses a fine mesh with size λ/6 (where λ is the wavelength) within that region. A coarser mesh with size λ/4 is used for the other domains. Quartic (4th order) elements are used to discretize the acoustic pressure, and quadratic (2nd order) elements are used to discretize temperature.

Table 1 shows the material properties used in the model simulation. The properties used for the tissue phantom are the measured data described in Ref. 8. For comparison, the

table also lists properties for human tissue published in Ref. 9.

TABLE 1: MATERIAL PROPERTIES USED IN THE MODEL.

Property	Density (kg/m^3)	Speed of sound (m/s)	Attenuation (Np/m/MHz)	Specific heat (J/(kg·K))	Thermal conductivity (W/(m·K))
Water (at 293.7 K)	1000	1483	0.025	N/A	N/A
Tissue phantom	1044	1568	8.55	3710	0.59
Human tissue	1000–1100	1450–1640	4.03–17.27	3600–3890	0.45–0.56

Results and Discussion

Figure 4 depicts the acoustic pressure. The ultrasonic beam travels through the water layer and into the tissue phantom. The beam converges into a focal area where the acoustic pressure amplitude reaches as high as **1.11** MPa at the focal point. This result agrees well with the results presented in Ref. 8. Note the diffraction-like pattern near the edges (especially at the top of the computational domain); this is not a physical phenomenon but an effect of the perfectly matched layer.

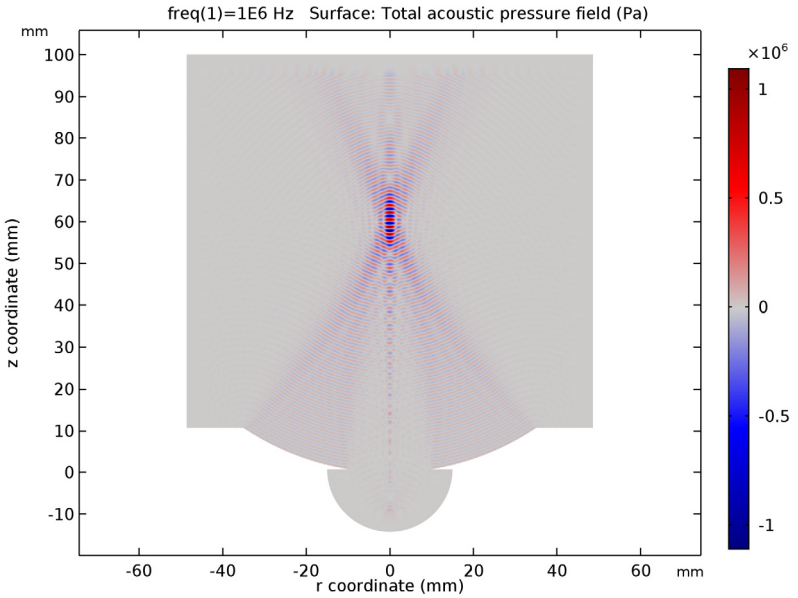


Figure 4: The acoustic pressure field in the water and tissue domains.

A depiction of the acoustic intensity magnitude is given in Figure 5. This plot shows more clearly how the acoustic energy is focused and distributed in the area of interest. Most of the heating happens in the oval-shaped focal area which is about 8 mm long and 1.3 mm wide. Figure 6 shows the acoustic pressure amplitude profile along the z-axis ($r = 0$). By zooming in around the peak pressure amplitude in Figure 6, the exact location of the acoustic focus is seen to be on-axis and 35 mm away from the tissue phantom and water interface (at $z = 59.6$ mm). Figure 7 shows the acoustic pressure amplitude profile along the radial direction in the focal plane ($z = 59.6$ mm).

Figure 8 shows the result of the temperature rise in the tissue phantom for a focal pressure amplitude of 1.11 MPa after 1 second of insonation. At $t = 1$ s, the maximum temperature rise is about one degree. The oval-shaped heated spot is about the same size as that of the acoustic focal area, which is more easily seen in the contour plot of the temperature as shown in Figure 9.

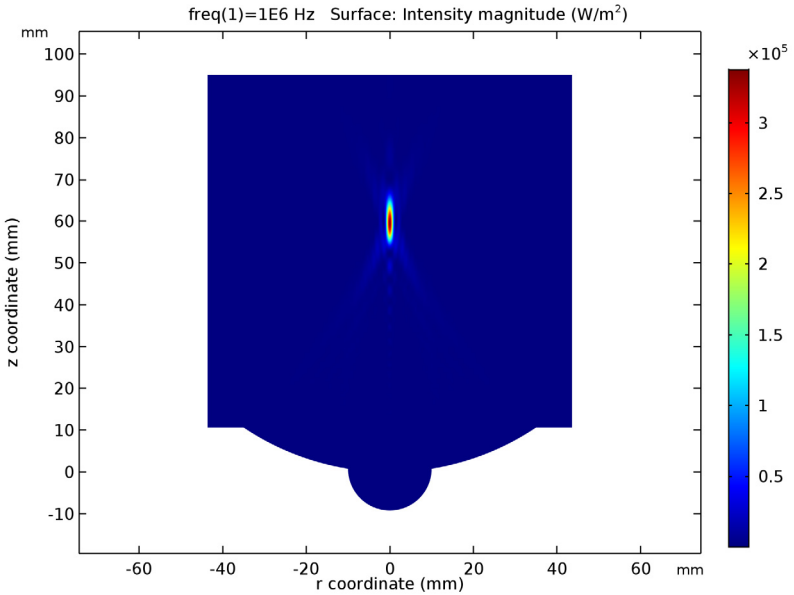


Figure 5: Surface plot of the acoustic intensity field, showing the acoustic energy to be concentrated to the focal region.

Figure 10 plots the heating and cooling curves at acoustic focus and 0.5 mm off acoustic focus in the focal plane. This is again for a focal pressure amplitude of 1.11 MPa. At these locations the tissue heats up when insonated and then cools down through natural conduction. The result at the acoustic focus agrees well with the results presented in Ref. 8.

This example shows how to model tissue heating induced by focused ultrasound when the acoustic pressure at focus is well below acoustic cavitation threshold. Because the model does not take nonlinearity into account it can be solved in the frequency domain. The maximal pressure in the focal region is comparable to the product ρc = 1.6 MPa which is the large signal limit for the pressure p. At this pressure level, acoustic energy is pumped out from the fundamental frequency to higher harmonics (Ref. 10). In tissue, the absorption coefficient is close to $f^{1.1}$ (for water, the power law for absorption is f^2). Higher harmonics result in a narrower focal region and more energy dissipation. Therefore, the current model tends to underestimate the total energy deposition

associated with the absorption of the ultrasonic wave and consequently also the maximum temperature rise at the focal region.

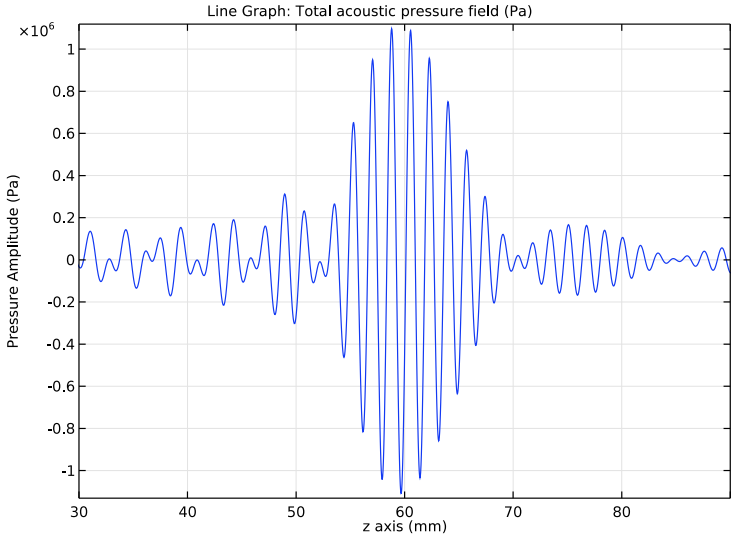


Figure 6: Acoustic pressure amplitude profile along the symmetry axis (r = 0).

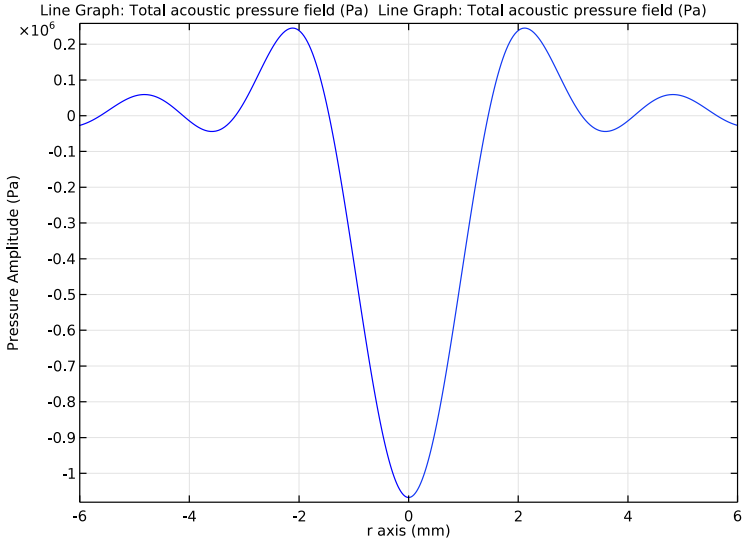


Figure 7: Acoustic pressure amplitude profile along the radial direction in the focal plane.

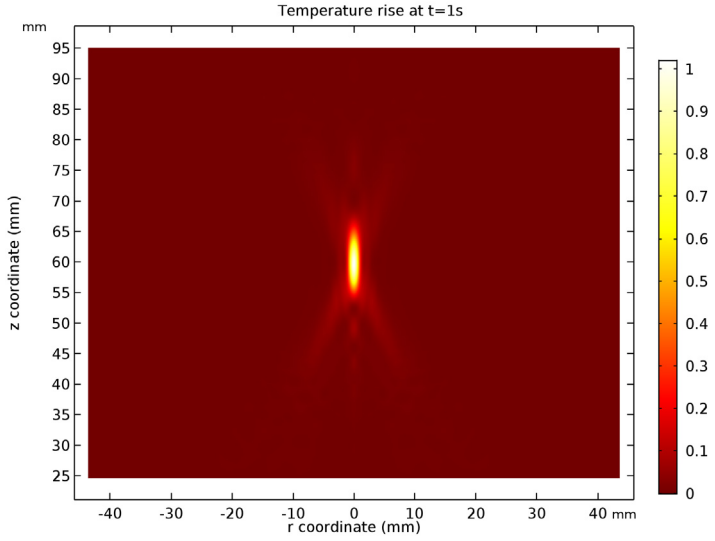


Figure 8: Surface plot of the temperature rise in the tissue phantom after 1 second insonation for a focal pressure amplitude of 1.11 MPa.

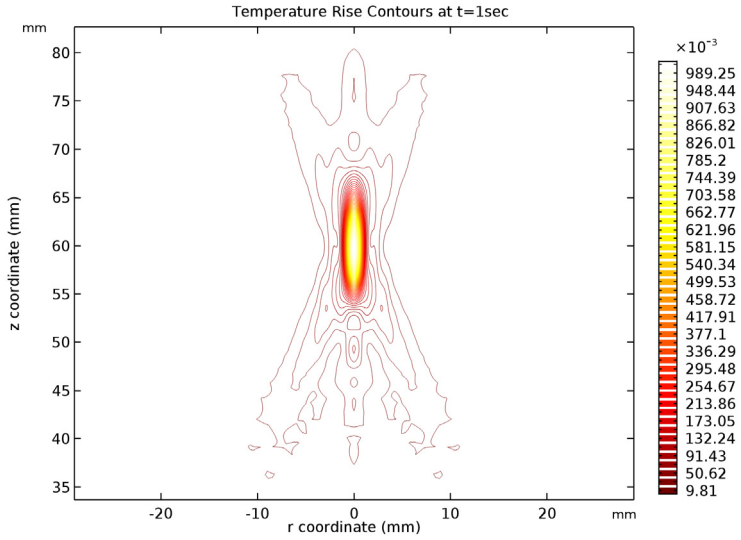


Figure 9: Contour plot of the temperature rise in the tissue phantom after 1 second of insonation for a focal pressure amplitude of 1.11 MPa. The oval-shaped heated spot is about the same size as that of the acoustic focal area.

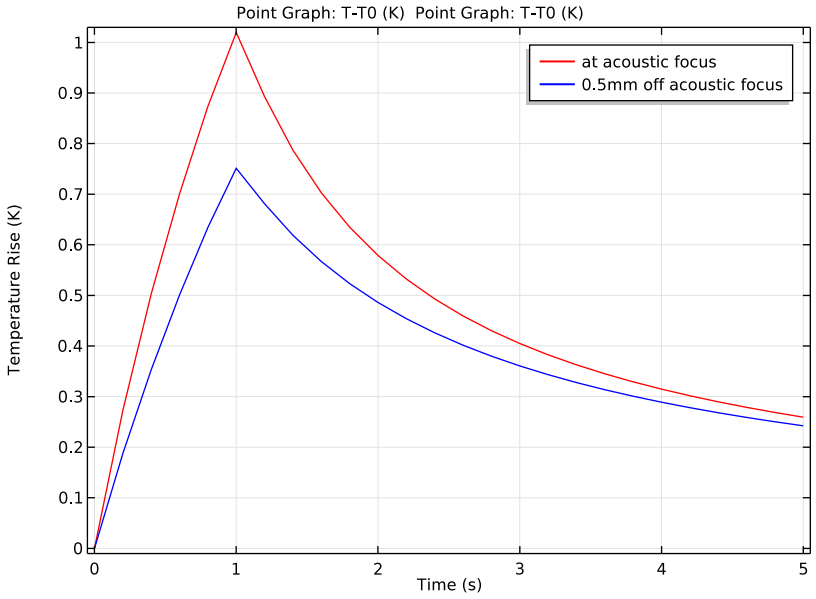


Figure 10: Heating and cooling curves at acoustic focus and 0.5 mm off-axis in the focal plane for 1 second of insonation with a focal pressure amplitude of 1.11 MPa.

The effect of energy dissipation serves to counteract the nonlinear effect and thus mitigate the waveform distortion (Ref. 10). Therefore the assumption of linear progressive wave motion should remain good as long as the nonlinearity is relatively small, especially for hyperthermia applications. The simulation results show that this model provides a good estimate of both acoustic field and temperature rise for focal pressure up to 1.11 MPa. It not only simulates the heating and cooling behavior at the focal region but also gives other useful information, such as the shape of the heated area and the side-lobe heating effect at those surrounding locations with pressure maxima outside the focal region (which is the main lobe). The results also show that the side lobes in the intensity field are mitigated in the temperature response due to the smoothing effect of conduction, as shown in Figure 11. In general, this model suggests that the temperature change is roughly proportional to the acoustic intensity.

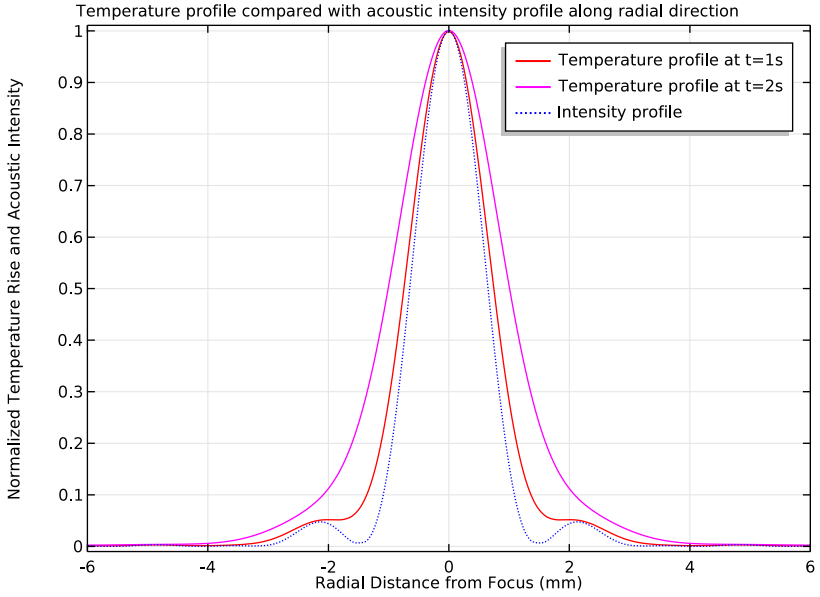


Figure 11: The temperature and intensity profiles show that the side lobes in the intensity field are mitigated in the temperature response due to the smoothing effect of the thermal conduction.

Notes About the COMSOL Implementation

This model uses two studies. Study 1 solves for Pressure Acoustics in the Frequency Domain to obtain the acoustic pressure field. In Study 2, the heat source calculated from the solution of Study 1 is then used in a time-dependent analysis to model heating and cooling of the tissue phantom.

References

1. K.J. Parker, "The Thermal Pulse Decay Technique for Measuring Ultrasonic Absorption Coefficients," *J. Acoust. Soc. Am.*, vol. 74, no. 5, pp. 1356–1361, 1983.

2. R.L. Clarke and G. ter Haar, "Temperature Rise Recorded during Lesion Formation by High Intensity Focused Ultrasound," *Ultrasound Med. Biol.*, vol. 23, no. 2, pp. 299–306, 1997.

3. N.T. Sanghvi, K. Hynynen, and F.L. Lizzi, "New Developments in Therapeutic Ultrasound," *IEEE Eng. Med. Biol. Mag.*, vol. 15, no. 6, pp. 83–92, 1996.

4. "Exposure Criteria for Medical Diagnostic Ultrasound: II. Criteria Based on All Known Mechanisms (Report No 140)," *Diagnostic Ultrasound Safety: A Summary of the Technical Report*, issued by the National Council on Radiation Protection and Measurements (NCRP), 2002.

5. P.N.T. Wells, "Biological Effects," *Biomedical Ultrasonics*, chap. 9, Academic Press, New York, 1977.

6. G. ter Haar, "Biological Effects of Ultrasound in Clinical Applications," *Ultrasound: Its Chemical, Physical, and Biological Effects*, K.S. Suslick, ed., VCH Publishers, New York, 1988.

7. W.L. Nyborg, "Biological Effects of Ultrasound: Development of Safety Guidelines. Part II: General Review," *Ultrasound Med. Biol.*, vol. 27, no. 3, pp. 301–333, 2001.

8. J. Huang, R.G. Holt, R.O. Cleveland, and R.A. Roy, "Experimental Validation of a Tractable Numerical Model for Focused Ultrasound Heating in Flow-through Tissue Phantoms," *J. Acoust. Soc. Am.*, vol. 116, no. 4, pt. 1, pp. 2451–2458, 2004.

9. F.A. Duck, *Physical Properties of Tissue*, Academic Press, 1990.

10. M.F. Hamilton and D.T. Blackstock eds., *Nonlinear Acoustics*, Academic Press, 1998.

Application Library path: `Acoustics_Module/Ultrasound/ultrasound_induced_heating`

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>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer>Bioheat Transfer (ht)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Custom Studies> Preset Studies for Some Physics Interfaces>Frequency Domain**.

8 Click **Done**.

GLOBAL DEFINITIONS

Define the parameters used in the model.

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
d0	3.8[nm]	3.8E-9 m	Displacement amplitude of transducer
z_tissue	24.6[mm]	0.0246 m	Starting position of tissue phantom
T0	293.7[K]	293.7 K	Initial temperature value
alpha_water	0.025[1/m]	0.025 1/m	Absorption coefficient of water
alpha_tissue	8.55[1/m]	8.55 1/m	Absorption coefficient of tissue phantom
f0	1[MHz]	1E6 Hz	Source frequency

Define a step function to turn off the acoustic source after 1 second of insonation. This is used for the transient thermal simulation. To improve convergence, define a smoothing transition zone that gently decrease the amplitude of the source to zero.

Step 1 (step1)

1 On the **Home** toolbar, click **Functions** and choose **Global>Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **From** text field, type 1.

4 In the **To** text field, type `0`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `0.005`.

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**.

Now, proceed to create the geometry. A sketch is depicted in Figure 2 and Figure 3. First, create the bowl-shaped transducer used as the acoustic source.

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `62.64`.

4 Locate the **Position** section. In the **z** text field, type `62.64`.

5 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `35`.

4 In the **Height** text field, type `10.69`.

5 Right-click **Rectangle 1 (r1)** and choose **Build Selected**.

Intersection 1 (int1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 Right-click **Intersection 1 (int1)** and choose **Build Selected**.

Create the tissue phantom domain and add layers for the perfectly matched layer (PML).

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `48.6`.

4 In the **Height** text field, type `75.5`.

5 Locate the **Position** section. In the **z** text field, type `z_tissue`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	5

7 Select the **Layers to the right** check box.

8 Select the **Layers on top** check box.

9 Clear the **Layers on bottom** check box.

10 Right-click **Rectangle 2 (r2)** and choose **Build Selected**.

Create the water domains and, also here, add layers for the PML.

Rectangle 3 (r3)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `48.6`.

4 In the **Height** text field, type `z_tissue-10.69`.

5 Locate the **Position** section. In the **z** text field, type `10.69`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	5

7 Select the **Layers to the right** check box.

8 Clear the **Layers on bottom** check box.

9 Right-click **Rectangle 3 (r3)** and choose **Build Selected**.

Circle 2 (c2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `15`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **z** text field, type `0.80336`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	5

8 Right-click **Circle 2 (c2)** and choose **Build Selected**.

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **c2** and **int1** only.

3 Right-click **Union 1 (uni1)** and choose **Build Selected**.

Delete Entities 1 (del1)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **uni1**, select Boundary 10 only.

3 Right-click **Component 1 (comp1)>Geometry 1>Delete Entities 1 (del1)** and choose **Build Selected**.

Finally, create an oval-shaped focal region where a finer mesh can be applied to resolve high gradient in the pressure field.

Ellipse 1 (e1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Ellipse**.

2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `7.5`.

4 In the **b-semiaxis** text field, type `1.5`.

5 In the **Sector angle** text field, type `180`.

6 Locate the **Position** section. In the **z** text field, type `z_tissue+35`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `270`.

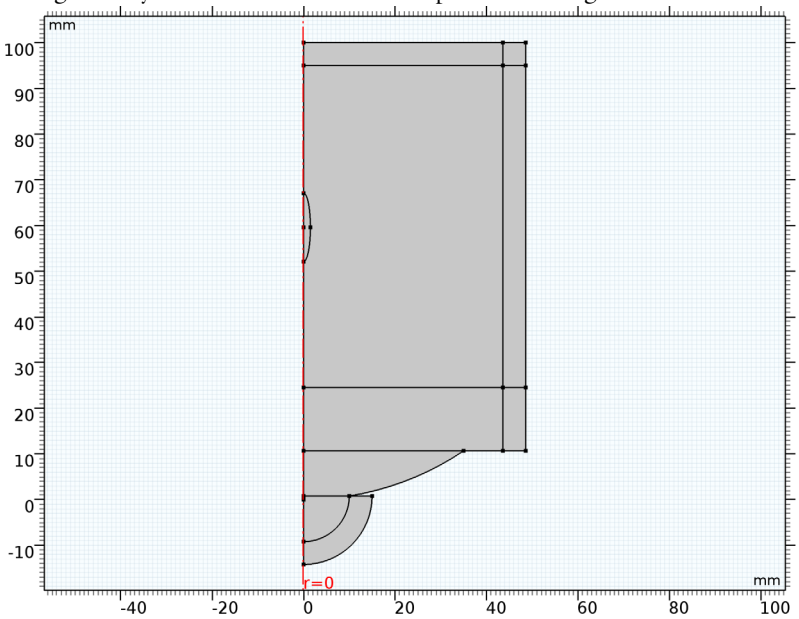
8 Right-click **Ellipse 1 (e1)** and choose **Build Selected**.

Form Union (fin)

1 On the **Geometry** toolbar, click **Build All**.

2 Click the **Zoom Extents** button on the **Graphics** toolbar.

The geometry should look like the one depicted in the figure below.



DEFINITIONS

Define the Perfectly Matched Layer domains for the Pressure Acoustics simulation. The rational stretching type is more efficient than the polynomial in these open problems, whereas, the polynomial is preferred in waveguide-like problems.

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 7–10 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

5 Locate the **Scaling** section. From the **Coordinate stretching type** list, choose **Rational**.

Perfectly Matched Layer 2 (pml2)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domain 1 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 In the **Center coordinate** text field, type `0.8034[mm]`.

5 Locate the **Scaling** section. From the **Coordinate stretching type** list, choose **Rational**.

6 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

This enables you to change the element order used for the discretization of the pressure dependent variable. Use quartic elements to discretize the pressure and the standard quadratic elements to discretize the temperature.

Now, set up the physics for Pressure Acoustics, Frequency Domain.

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 **Sound Pressure Level Settings** section.

3 From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

4 Locate the **Typical Wave Speed for Perfectly Matched Layers** section. In the c_{ref} text field, type `1483[m/s]`.

5 Click to expand the **Discretization** section. From the **Element order** list, choose **Quartic Lagrange**.

Specify the physical properties for the water domains.

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 **Linear elastic with attenuation**.

4 In the α text field, type `alpha_water`.

Add a second Pressure Acoustics Material Model node for the tissue domains. You will define the tissue material below.

Pressure Acoustics 2

1 On the **Physics** toolbar, click **Domains** and choose **Pressure Acoustics**.

2 Select Domains 5–7, 9, and 10 only.

3 In the **Settings** window for **Pressure Acoustics**, locate the **Pressure Acoustics Model** section.

4 From the **Fluid model** list, choose **Linear elastic with attenuation**.

5 In the α text field, type `alpha_tissue`.

Define the normal displacement amplitude d0 at the surface of the ultrasound transducer.

Normal Displacement 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Displacement**.

2 Select Boundary 32 only.

3 In the **Settings** window for **Normal Displacement**, locate the **Normal Displacement** section.

4 In the d_n text field, type `d0`.

Now, specify the physics of the Bioheat Transfer.

BIOHEAT TRANSFER (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Bioheat Transfer (ht)**.

2 In the **Settings** window for **Bioheat Transfer**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domains 5 and 6 only.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, type `T0` in the T text field.

Add the absorbed acoustic energy as the domain heat source. The variable `acpr.Q_pw` represents the dissipated power density for plane waves.

3 In the **Model Builder** window, click **Bioheat Transfer (ht)**.

Heat Source 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domains 5 and 6 only.

3 In the **Settings** window for **Heat Source**, locate the **Heat Source** section.

4 In the Q_0 text field, type `acpr.Q_pw*step1(t[1/s]-1)`.

Apply a constant temperature boundary condition on the computational boundaries of the thermal simulation.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundaries 9, 14, and 20 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `T0`.

The next step is to set up the materials used in the model. Use the default water material and define your own tissue material.

ADD MATERIAL

1 On 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>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Define the material properties of tissue phantom and apply it to the tissue phantom domains.

Material 2 (mat2)

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 `Tissue phantom` in the **Label** text field.

3 Select Domains 5–7, 9, and 10 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1044	kg/m³	Basic
Speed of sound	c	1568	m/s	Basic
Thermal conductivity	k	0.59	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	3710	J/(kg·K)	Basic

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

Pressure Acoustics 1

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)** node, then click **Pressure Acoustics 1**.
2. In the **Settings** window for **Pressure Acoustics**, locate the **Model Inputs** section.
3. In the T text field, type `T0`.

Pressure Acoustics 2

1. In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)** click **Pressure Acoustics 2**.
2. In the **Settings** window for **Pressure Acoustics**, locate the **Model Inputs** section.
3. In the T text field, type `T0`.

MESH 1

In the following steps, create a first mesh for the pressure acoustics simulation and then create a mesh for the thermal problem. Since the two physics are different the meshes need to have different properties. For the acoustic simulation the mesh should resolve the wavelength of the problem. Using fourth order quintic elements, 4 mesh elements per wavelength are adequate.

Free Triangular 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.
2. In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 2–6 only.

   Use a finer mesh in the focal region to resolve the large gradients in the pressure field. Use 6 elements per wavelength here.

Size 1

1. Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. Click **Clear Selection**.
4. Select Domain 6 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 `1568[m/s]/f0/6`.

Size 2

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 Click **Clear Selection**.

4 Select Domain 5 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 `1568[m/s]/f0/4`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `1483[m/s]/f0/4`.

Set up a mapped mesh in the perfectly matched layer region.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1 and 7–10 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

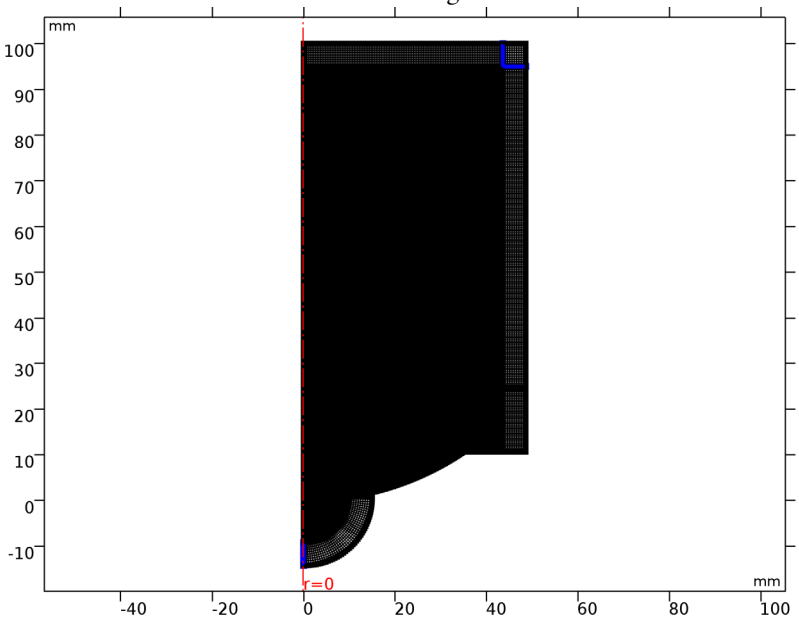
2 Select Boundaries 1, 22, and 23 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `10`.

5 Click **Build All**.

The mesh should look like the one in the figure below.



Create a coarser mesh for thermal simulation.

COMPONENT 1 (COMP1)

Mesh 2

On the **Mesh** toolbar, click **Add Mesh**.

MESH 2

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** right-click **Mesh 2** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 5 and 6 only.

Size 1

1 Right-click **Component 1 (comp1)>Meshes>Mesh 2>Free Triangular 1** and choose **Size**.

2 Select Domain 6 only.

3 In the **Settings** window for **Size**, locate the **Element Size** section.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `1483[m/s]/f0/8`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes>Mesh 2** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

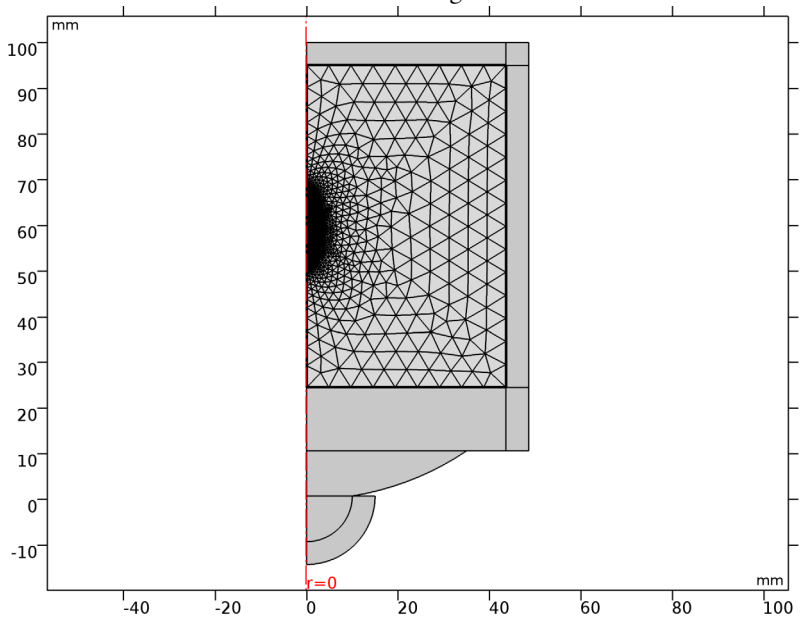
3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `5`.

5 In the **Maximum element growth rate** text field, type `1.2`.

6 Click **Build All**.

The mesh should look like the one in the figure below.



Solve the physics of Pressure Acoustics only in frequency domain using the finer mesh **Mesh 1**.

STUDY 1

Step 1: Frequency Domain

1. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
2. In the **Frequencies** text field, type `f0`.
3. Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for the **Bioheat Transfer** interface.
4. Click to expand the **Mesh selection** sectionto check that **Mesh 1** is selected.
5. On the **Home** toolbar, click **Compute**.

ROOT

Now, add a transient analysis study type and solve the bioheat transfer model in the time domain using the coarser **Mesh 2**. The acoustic model serves as input to calculate the heat source.

ADD STUDY

1. On 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 **Custom Studies> Preset Studies for Some Physics Interfaces>Time Dependent**.
4. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Pressure Acoustics, Frequency Domain (acpr)** interface.
5. Click **Add Study** in the window toolbar.
6. On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
2. Click **Range**.
3. In the **Range** dialog box, type `0.2` in the **Step** text field.
4. In the **Stop** text field, type `5`.
5. Click **Replace**.
6. In the **Settings** window for **Time Dependent**, click to expand the **Values of dependent variables** section.

7 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

8 From the **Method** list, choose **Solution**.

9 From the **Study** list, choose **Study 1, Frequency Domain**.

10 Click to expand the **Mesh selection** sectionto check that **Mesh 2** is selected.

Specify the time steps for the time-dependent solver.

Solution 2 (sol2)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time stepping** section.

4 Locate the **Time Stepping** section. From the **Maximum BDF order** list, choose **5**.

5 Select the **Maximum step** check box.

6 In the associated text field, type `0.02`.

7 On the **Study** toolbar, click **Compute**.

RESULTS

Data Sets

Having solved the model, proceed to the results analysis. Follow the steps below to generate plots of the acoustic pressure and the temperature fields.

First, create a mirror dataset to better visualize the results in a full cut plane through the axisymmetric geometry.

Acoustic Pressure (acpr)

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 2D**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Mirror 2D 1**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `r coordinate (mm)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `z coordinate (mm)`.

8 Clear the **Plot data set edges** check box.

Use the wave colortable to get an enhanced view of the acoustic field.

Surface 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **Wave**.

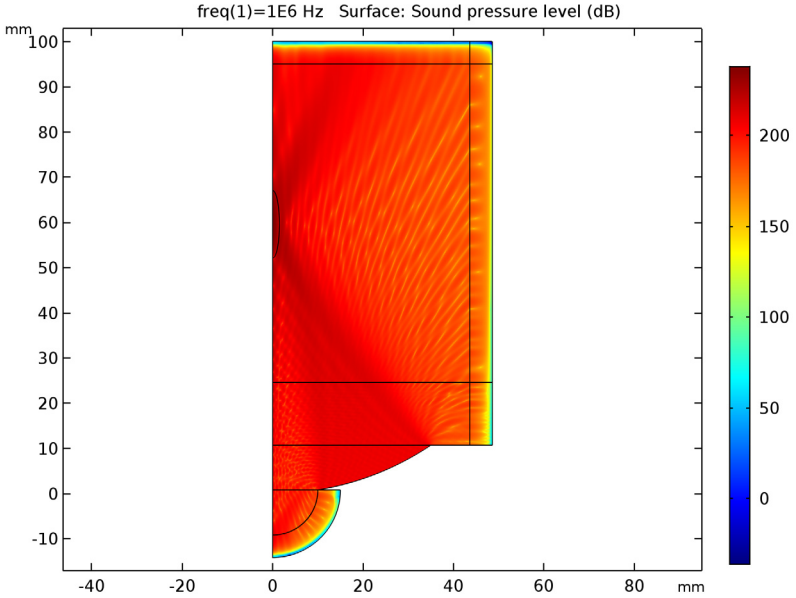
4 On the **Acoustic Pressure (acpr)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The plot should like look that shown in Figure 4.

Sound Pressure Level (acpr)

This second plot shows the sound pressure level (in the acoustics domain as well as in the PML region); it should look like that in the figure below. Note that the acoustic field is damped by about 150 dB in the PML region.



Deactivate plotting in the unphysical PML region for the remaining of the results analysis.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Data Sets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1. On the **Results** toolbar, click **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Click in the **Graphics** window and then press Ctrl+A to select all domains.
5. Select Domains 2–6 only.

Generate the acoustic intensity plot (the magnitude of the intensity vector) as shown in Figure 5.

2D Plot Group 7

1. On the **Results** toolbar, click **2D Plot Group**.
2. In the **Settings** window for **2D Plot Group**, type `Acoustic Intensity field` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Mirror 2D 1**.
4. Locate the **Plot Settings** section. Select the **x-axis label** check box.
5. In the associated text field, type `r coordinate (mm)`.
6. Select the **y-axis label** check box.
7. In the associated text field, type `z coordinate (mm)`.
8. Clear the **Plot data set edges** check box.

Surface 1

1. Right-click **Acoustic Intensity field** 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>Pressure Acoustics, Frequency Domain>Intensity>acpr.I_mag - Intensity magnitude**.
3. On the **Acoustic Intensity field** toolbar, click **Plot**.

Next, generate a line plot of the acoustic pressure amplitude along the axis of symmetry, as shown in Figure 6.

1D Plot Group 8

1. On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure Amplitude along Axial z-Axis` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **x-axis label** check box.

4 In the associated text field, type `z axis (mm)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Pressure Amplitude (Pa)`.

Line Graph 1

1 Right-click **Pressure Amplitude along Axial z-Axis** and choose **Line Graph**.

2 Select Boundaries 3, 4, 6, 8, and 10–12 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 `z`.

6 On the **Pressure Amplitude along Axial z-Axis** toolbar, click **Plot**.

Pressure Amplitude along Axial z-Axis

1 In the **Model Builder** window, under **Results** click **Pressure Amplitude along Axial z-Axis**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `30`.

5 In the **x maximum** text field, type `90`.

6 On the **Pressure Amplitude along Axial z-Axis** toolbar, click **Plot**.

You can use the **Zoom Box** and **Zoom Extents** tools to zoom in around the acoustic focal point. The maximum pressure amplitude is located at z = 59.6 mm.

Define a line data set and generate a plot of the acoustic pressure amplitude along the radial direction in the focal plane, as shown in Figure 7.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **Z** to `59.6`.

4 In row **Point 2**, set **R** to `43.6` and **z** to `59.6`.

5 Click **Plot**.

1D Plot Group 9

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Pressure Amplitude along Radial Axis on the focal plane` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `r axis (mm)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Pressure Amplitude (Pa)`.

Line Graph 1

1 Right-click **Pressure Amplitude along Radial Axis on the focal plane** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `r`.

Because the pressure field is represented by fourth order (quartic) elements you can increase the resolution of the plot. The resolution sets the number of interpolation points used inside each finite element.

5 Click to expand the **Quality** section. From the **Resolution** list, choose **Finer**.

6 On the **Pressure Amplitude along Radial Axis on the focal plane** toolbar, click **Plot**.

Line Graph 2

1 Right-click **Results>Pressure Amplitude along Radial Axis on the focal plane>Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `-r`.

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

Pressure Amplitude along Radial Axis on the focal plane

1 In the **Model Builder** window, under **Results** click **Pressure Amplitude along Radial Axis on the focal plane**.

2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `-6`.

5 In the **x maximum** text field, type `6`.

6 On the **Pressure Amplitude along Radial Axis on the focal plane** toolbar, click **Plot**.

Data Sets

Now, use the following steps to generate the temperature field plots as shown in the results.

First, create a mirror data-set for better visualization of the temperature field.

Mirror 2D 2

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 2D**.

2 In the **Settings** window for **Mirror 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

2D Plot Group 10

1 On the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Temperature Rise at t=1s` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Mirror 2D 2**.

4 From the **Time (s)** list, choose **1**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Temperature rise at t=1s`.

7 Locate the **Plot Settings** section. Select the **x-axis label** check box.

8 In the associated text field, type `r coordinate (mm)`.

9 Select the **y-axis label** check box.

10 In the associated text field, type `z coordinate (mm)`.

11 Clear the **Plot data set edges** check box.

Surface 1

1 Right-click **Temperature Rise at t=1s** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `T-T0`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

5 On the **Temperature Rise at t=1s** toolbar, click **Plot**.

6 Click the **Zoom Extents** button on the **Graphics** toolbar.

The plot should like look that shown in Figure 8.

Next, generate the isothermal contours plot as shown in Figure 9.

2D Plot Group 11

1. On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.
2. In the **Settings** window for **2D Plot Group**, type `Isothermal Contours at t=1s` in the **Label** text field.
3. Locate the **Data** section. From the **Data set** list, choose **Mirror 2D 2**.
4. From the **Time (s)** list, choose **1**.
5. Locate the **Title** section. From the **Title type** list, choose **Manual**.
6. In the **Title** text area, type `Temperature Rise Contours at t=1sec`.
7. Locate the **Plot Settings** section. Select the **x-axis label** check box.
8. In the associated text field, type `r coordinate (mm)`.
9. Select the **y-axis label** check box.
10. In the associated text field, type `z coordinate (mm)`.
11. Clear the **Plot data set edges** check box.

Contour 1

1. Right-click **Isothermal Contours at t=1s** and choose **Contour**.
2. In the **Settings** window for **Contour**, locate the **Expression** section.
3. In the **Expression** text field, type `T-T0`.
4. Locate the **Levels** section. In the **Total levels** text field, type `50`.
5. Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.
6. On the **Isothermal Contours at t=1s** toolbar, click **Plot**.
7. Click the **Zoom Extents** button on the **Graphics** toolbar.

Define two point data-sets, one at the focus and the other at 0.5mm off the acoustic focus. Then generate two point graphs (within a 1D Plot Group) of the temperature rise as function of time, as shown in Figure 10.

Cut Point 2D 1

1. On the **Results** toolbar, click **Cut Point 2D**.
2. In the **Settings** window for **Cut Point 2D**, locate the **Data** section.
3. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.
4. Locate the **Point Data** section. In the **R** text field, type `0`.
5. In the **Z** text field, type `59.6`.

Cut Point 2D 2

1 Right-click **Cut Point 2D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

3 In the **R** text field, type `0.5`.

1D Plot Group 12

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature Rise vs. Time at focus and 0.5mm off focus` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Time (s)`.

6 Select the **y-axis label** check box.

7 In the associated text field, type `Temperature Rise (K)`.

Point Graph 1

1 Right-click **Temperature Rise vs. Time at focus and 0.5mm off focus** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 2D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `T-T0`.

5 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`at acoustic focus`

Point Graph 2

1 Right-click **Results>Temperature Rise vs. Time at focus and 0.5mm off focus> Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Point 2D 2**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
0.5mm off acoustic focus

6 On the **Temperature Rise vs. Time at focus and 0.5mm off focus** toolbar, click **Plot**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Define a line data set and generate a 1D Line Graph of the temperature rise along the radial direction on the focal plane. Plot this after 1 and 2 seconds of insonation, as shown in Figure 11.

Cut Line 2D 1

In the **Model Builder** window, expand the **Data Sets** node.

Cut Line 2D 2

1 Right-click **Cut Line 2D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

1D Plot Group 13

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature profile compared with acoustic intensity profile` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Temperature profile compared with acoustic intensity profile along radial direction`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Radial Distance from Focus (mm)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Normalized Temperature Rise and Acoustic Intensity`.

Line Graph 1

1 Right-click **Temperature profile compared with acoustic intensity profile** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 2D 2**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (s)** list, select **1**.

6 Locate the **y-Axis Data** section. In the **Expression** text field, type `(T-T0)/1.0208`.

7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `r`.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **Red**.

10 Click to expand the **Legends** section. Select the **Show legends** check box.

11 From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
Temperature profile at t=1s

Line Graph 2

1 Right-click **Results>Temperature profile compared with acoustic intensity profile> Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `-r`.

4 Locate the **Legends** section. Clear the **Show legends** check box.

Line Graph 3

1 Right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 In the **Times (s)** list, select **2**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `(T-T0)/0.5785`.

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Magenta**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
Temperature profile at t=2s

Line Graph 4

1 Right-click **Results>Temperature profile compared with acoustic intensity profile> Line Graph 3** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `-r`.

4 Locate the **Legends** section. Clear the **Show legends** check box.

Line Graph 5

1 In the **Model Builder** window, under **Results>**
Temperature profile compared with acoustic intensity profile right-click **Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Intensity>**
acpr.I_mag - Intensity magnitude.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `acpr.I_mag/3.3743e5`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

7 Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

8 Locate the **Legends** section. In the table, enter the following settings:

Legends
Intensity profile

Line Graph 6

1 Right-click **Results>Temperature profile compared with acoustic intensity profile>**
Line Graph 5 and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 In the **Expression** text field, type `-r`.

4 Locate the **Legends** section. Clear the **Show legends** check box.

5 On the **Temperature profile compared with acoustic intensity profile** toolbar, click **Plot**.

Temperature profile compared with acoustic intensity profile

1 In the **Model Builder** window, under **Results** click
Temperature profile compared with acoustic intensity profile.

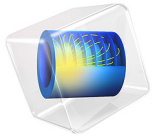
2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `-6`.

5 In the **x maximum** text field, type `6`.

6 On the **Temperature profile compared with acoustic intensity profile** toolbar, click **Plot**.



Uniform Layer Waveguide

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Whenever dimensions in waveguides become small compared to the viscous and thermal boundary layers, it is necessary to model acoustics using thermoviscous acoustics. In the present model the thermoviscous acoustic wave field in a shallow uniform waveguide is modeled and compared to an analytical solution.

Model Definition

An infinitely wide slit of length L and height H is subject to a harmonically varying pressure drop of 1 Pa. In order to reduce the model size, only a section of width L is modeled using symmetry conditions. The waveguide top and bottom are modeled as no slip isothermal walls.

Model parameters are summarized in Table 1.

TABLE 1: LOADED PARAMETERS

PARAMETER	EXPRESSION	DESCRIPTION
f_0	500 Hz	Frequency
T_0	293 K	Ambient temperature
p_0	1 atm	Atmospheric pressure
H	1 mm	Waveguide height
L	5 mm	Waveguide side lengths
p_{in}	1 Pa	Inlet pressure
d_{visc}	$0.22\text{mm}\sqrt{\frac{100\text{Hz}}{f_0}}$	Approximate expression for the viscous boundary layer thickness at 20°C and 1 atm.
μ_{B0}	0 Pa·s	Bulk viscosity

ANALYTICAL THEORY

A thermoviscous acoustics problem can in principle be thought of as a three wave problem, each with its own wave number: the acoustic k_0, the viscous k_v, and the thermal k_h, with

$$k_0 = \frac{\omega}{c_0} \qquad k_v^2 = \frac{-i\omega\rho_0}{\mu} \qquad k_h^2 = \frac{-i\omega\rho_0 C_p}{k}$$

where c_0 is the isentropic speed of sound, ω the angular frequency, ρ_0 is the static density, μ is the dynamic viscosity, C_p is the heat capacity, and k is the thermal conductance.

The thermal and viscous waves are rapidly decaying waves normal to a wall. The three waves interact and in the case of geometries with small dimensions, this interaction becomes evident. In simple geometries, analytical solutions of this interaction exist. In the case of a uniform slit, the cross-sectional variation of the temperature T and velocity $\mathbf{u}$ is:

$$\mathbf{u} = \frac{-\Psi_v \nabla p}{ik_0 Z_0} \qquad T = \frac{\Psi_h p}{\rho_0 C_p}$$

where the function Ψ is a complex-valued scalar field

$$\Psi_\phi = 1 - \frac{\cos(k_\phi z)}{\cos\left(k_\phi \frac{H}{2}\right)}$$

and ϕ is either v or h. The pressure gradient and the pressure are in this simple geometry given by (disregarding the phase)

$$\nabla p = \frac{p_{\text{in}} - p_{\text{out}}}{L} \qquad p = \nabla p \cdot (L - x)$$

Results and Discussion

At 500 Hz, the characteristic size of the viscous boundary layer is about 0.1 mm (for air at 20^{o} C), this length is 1/10 of the waveguide thickness H. This is seen in Figure 1 as the changing colors region near the wall where the velocity is varying to fulfill the no slip condition.

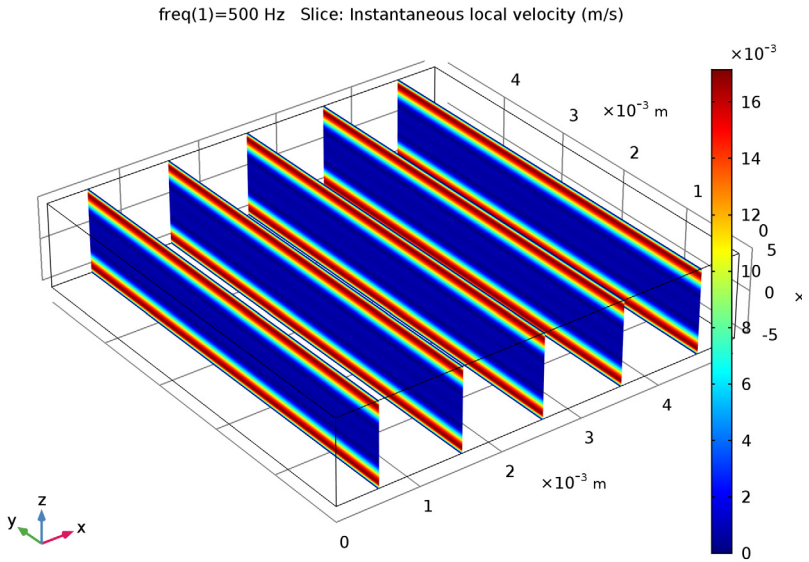


Figure 1: Slice plot of the instantaneous particle velocity U. Outside the boundary layer (red regions) the velocity profile becomes flat as in pressure acoustics.

The velocity and temperature profiles are probed using a 3D cut line and compared to the expressions found in the theory. The velocity profile is shown in Figure 2, while the temperature profile is shown in Figure 3. The results agree very well. When modeling acoustics in small dimensions, it is essential to include the thermal and viscous losses (see Ref. 1).

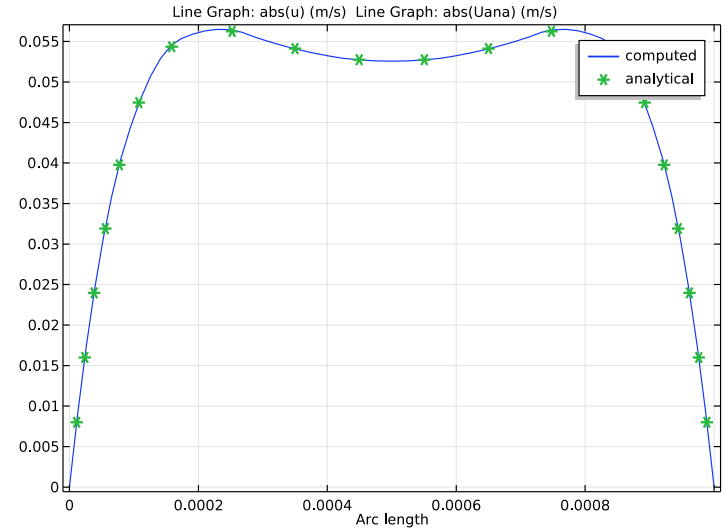


Figure 2: Comparison of the analytical and numerical velocity profiles.

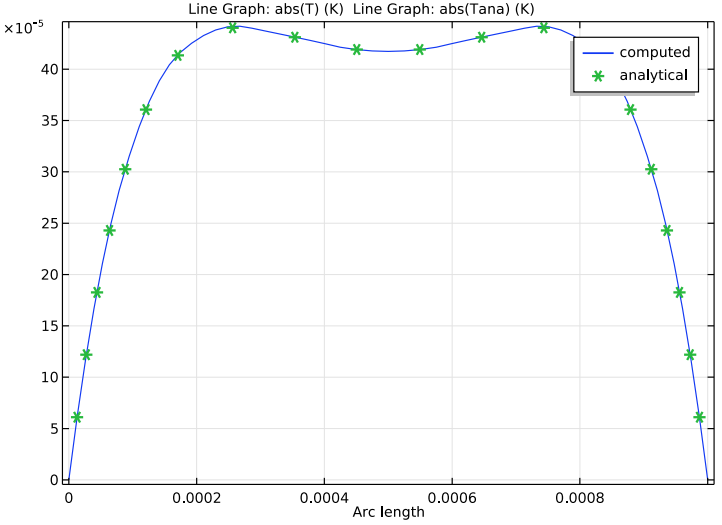


Figure 3: Comparison of the analytical and the COMSOL generated solution for the amplitude of the acoustic temperature variation T.

Reference

1. H. Tijdeman, "On the propagation of sound waves in cylindrical tubes," *J. Sound Vib*, vol. 39, no. 1, pp. 1–33, 1975.

Application Library path: `Acoustics_Module/Verification_Examples/uniform_layer_waveguide`

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 **Acoustics>Thermoviscous Acoustics> Thermoviscous Acoustics, Frequency Domain (ta)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

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 `uniform_layer_waveguide_parameters.txt`.

DEFINITIONS

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `uniform_layer_waveguide_variables.txt`.

GEOMETRY 1

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `L`.

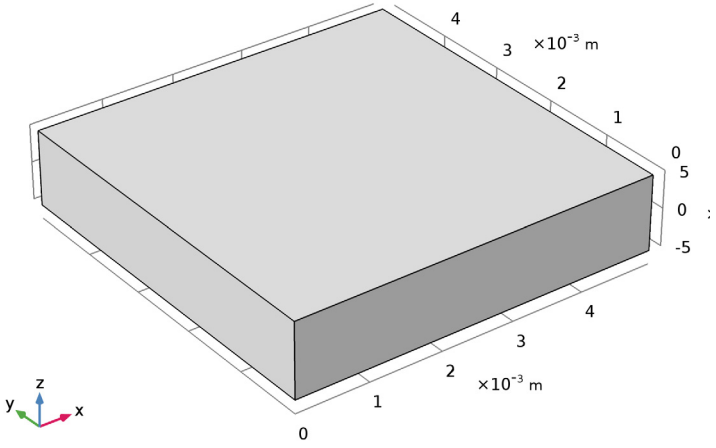
4 In the **Depth** text field, type `L`.

5 In the **Height** text field, type `H`.

6 Locate the **Position** section. In the **z** text field, type `-H/2`.

7 Click **Build Selected**.

8 Click the **Zoom Extents** button on the **Graphics** toolbar.



ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	muB0	Pa·s	Basic

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Thermoviscous Acoustics Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)** click **Thermoviscous Acoustics Model 1**.

2 In the **Settings** window for **Thermoviscous Acoustics Model**, locate the **Thermoviscous Acoustics Model** section.

3 From the $\rho_0(p_{0,T0})$ list, choose **From material**.

4 Locate the **Model Input** section. In the T_0 text field, type `T0`.

5 In the p_0 text field, type `p0`.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 2 and 5 only.

Pressure (Adiabatic) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure (Adiabatic)**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Pressure (Adiabatic)**, locate the **Pressure** section.

4 In the p_{bnd} text field, type `pin`.

Pressure (Adiabatic) 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Pressure (Adiabatic)**.

2 Select Boundary 6 only.

MESH 1

When modeling thermoviscous acoustics, the mesh is important as it needs to resolve the boundary layers properly. In the parameters list the quantity `d_visc` approximates the thickness of the viscous boundary layer, in air, for the frequency `f0`. This parameter is used in the boundary layer mesh properties in order to get a proper mesh.

Mapped 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 1 only.

Distribution 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 4 and 6 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `3`.

5 Click **Build Selected**.

Distribution 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, click **Build Selected**.

Boundary Layer Properties

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Boundary Layers**.

2 Select Boundaries 3 and 4 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.

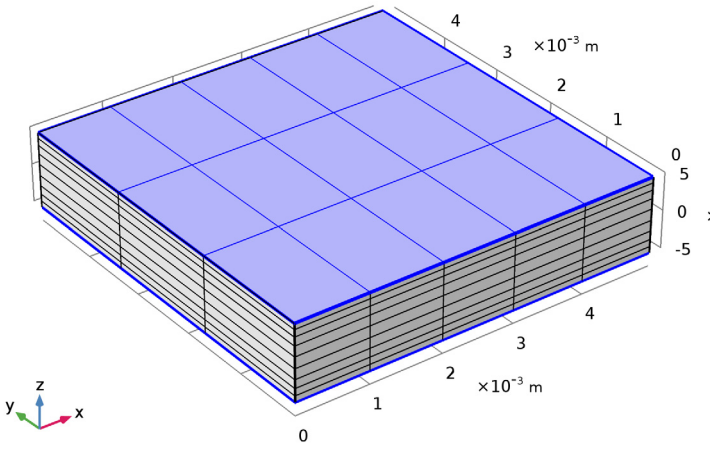
4 In the **Number of boundary layers** text field, type `3`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `0.8*d_visc`.

7 Click **Build All**.

The finished mesh should look like that in the figure below.



Mesh of the uniform layer waveguide including the boundary layer mesh.

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 **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `f0`.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Acoustic Velocity (ta)

The second default plot shows a slice plot of the instantaneous particle velocity Figure 1.

Data Sets

To compare the analytical and numerical velocity and temperature profiles, as done in Figure 2 and Figure 3, follow the steps given below.

Cut Line 3D 1

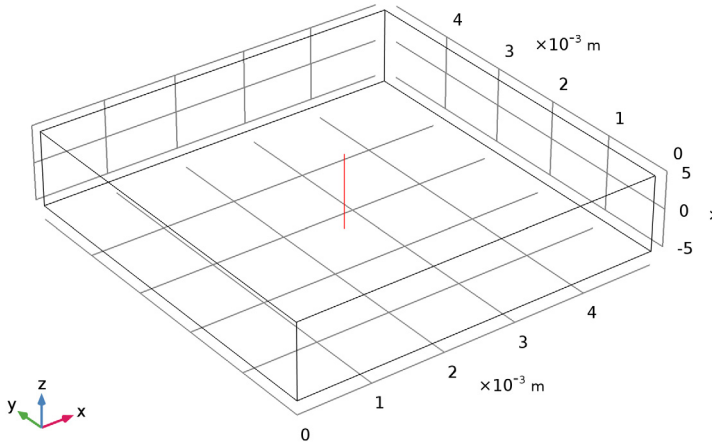
1 On the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **X** to `L/2`, **y** to `L/2`, and **z** to `H/2`.

4 In row **Point 2**, set **X** to `L/2`, **y** to `L/2`, and **z** to `-H/2`.

5 Click **Plot**.



1D Plot Group 4

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Velocity Profile` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 3D 1**.

Line Graph 1

1 Right-click **Velocity Profile** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `abs(u)`.

4 Click to expand the **Legends** section. Select the **Show legends** check box.

5 From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
computed

Line Graph 2

1 In the **Model Builder** window, under **Results** right-click **Velocity Profile** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `abs(Uana)`.

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 In the **Number** text field, type `20`.

7 Locate the **Legends** section. Select the **Show legends** check box.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
analytical

10 On the **Velocity Profile** toolbar, click **Plot**.

Velocity Profile 1

1 Right-click **Velocity Profile** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature Profile` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Results>Temperature Profile** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `abs(T)`.

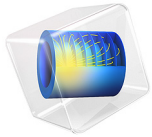
Line Graph 2

1 In the **Model Builder** window, under **Results>Temperature Profile** click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `abs(Tana)`.

4 On the **Temperature Profile** toolbar, click **Plot**.



Loudspeaker Driver in a Vented Enclosure

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example models the acoustic behavior of a loudspeaker driver mounted in a bass reflex enclosure.

Two of the most important design parameters for a loudspeaker driver is its sensitivity and the directivity of the system. The sensitivity is commonly defined as the on-axis sound pressure level, measured at a 1 m distance, as the driver is loaded by an AC voltage of 4 V. The directivity is assessed using the Directivity plot and represents the spatial sensitivity plotted against the frequency in a contour-like plot.

To isolate the driver's performance from that of the environment it usually operates in, the driver is often set directly in an infinite baffle. This approach is used in another example Loudspeaker Driver model in the Acoustics Module Application Library. The model described here borrows the electromagnetic results from that example and shows how the enclosure affects the sensitivity.

The model uses the Acoustic-Shell Interaction, Frequency Domain multiphysics interface. This interface provides automatic coupling between the Shell equation for the moving parts and the Pressure Acoustics equation in the surrounding air.

Note: The model requires the Acoustics Module and the Structural Mechanics Module since it involves the use of the Shell interface.

Model Definition

Figure 1 shows the geometry of the considered driver in an infinite baffle, as modeled in the Loudspeaker Driver example. In the model described here, the driver is set in a frame and placed in a bass reflex enclosure (Figure 2). The defining feature of this enclosure type is the vent, which in a properly designed enclosure acts to boost the sound at low frequencies.

Figure 3 shows the driver mounted in the enclosure. The infinite baffle is now flush with the front wall of the enclosure. The moving parts of the driver are drawn as surfaces rather than thin volumes. This lets you model them as shells, and vastly reduces the number of mesh elements required to resolve the model.

Figure 4 displays the complete model geometry, which includes a spherical domain for the air outside the enclosure. The concentric spherical mantle that envelopes the air domain is

a perfectly matched layer (PML), acting to absorb the outgoing waves with a minimum of reflections.

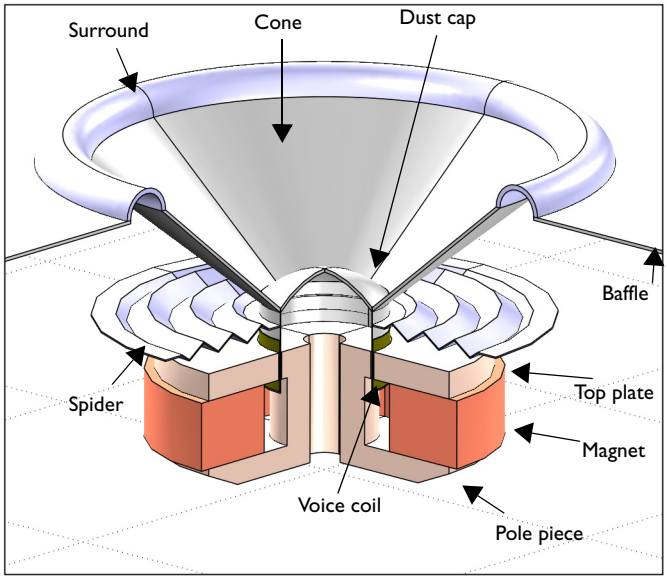


Figure 1: The driver, here set in an infinite baffle as in the Loudspeaker Driver model.

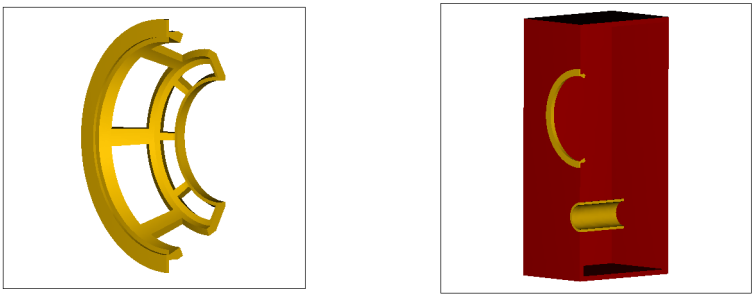


Figure 2: The frame and the vented enclosure.

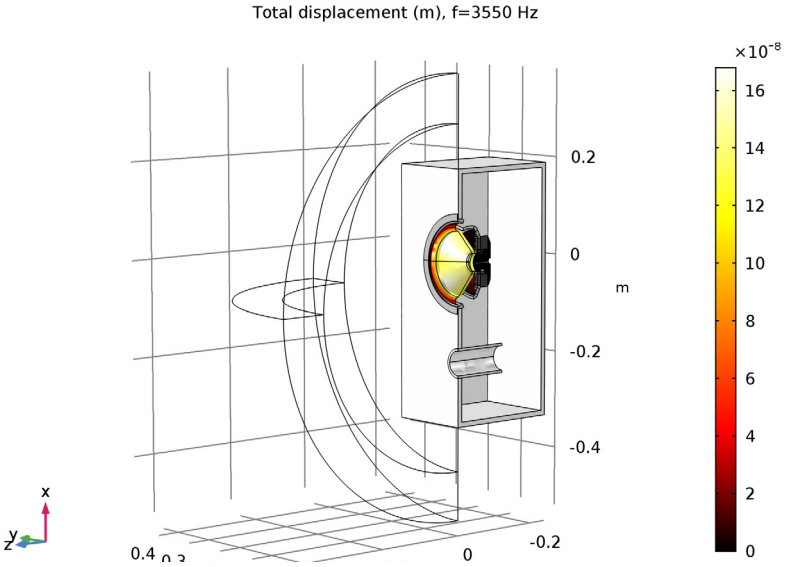


Figure 3: The geometry of the loudspeaker. The front of the box is set in an infinite baffle (not shown). The figure also shows the displacement of the moving parts at 3550 Hz.

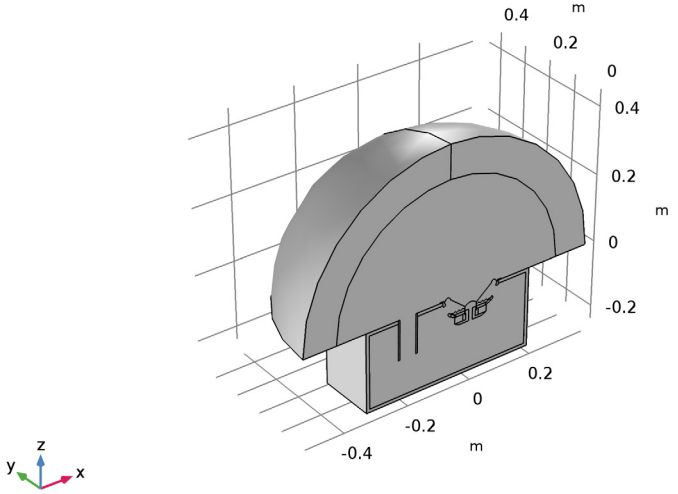


Figure 4: The complete model geometry. Thanks to the symmetry with respect to the xz-plane, the model consists of only one half of the speaker and the outside air.

The model is driven by an electromagnetic force acting on the voice coil:

$$F_e = \frac{BLV_0}{Z_b} - v\frac{(BL)^2}{Z_b}$$

as derived in the documentation for the Loudspeaker Driver model. BL is the so-called force factor of the voice coil and Z_b is its blocked impedance; the electric impedance as measured when the coil is at stand-still. While BL is a constant 7.55 N/A, Z_b is complex-valued function of the frequency. Both BL and Z_b are taken directly from the Loudspeaker Driver, the latter through interpolation from text files with the resistive and inductive contributions listed versus the frequency.

V_0 is the applied driving voltage. The definition of sensitivity assumes a driving power that equals 1 W when the total impedance of the loudspeaker is at its nominal value. The modeled driver has a nominal impedance of 8 Ω, which translates to a driving voltage of $V = V_0 e^{i\omega t}$ with the amplitude V_0 = 4 V. The second term in the expression for the driving force contains the velocity of the voice coil, v, which is unknown prior to the computation.

The electromagnetic force is applied as a homogeneous force density over the boundaries constituting the voice coil. As the cone and the suspension move along and deform, their local normal acceleration is automatically coupled over to become a sound source in the Pressure Acoustics equation. The computed acoustic pressure acts back as a load on the shell.

For a discussion of the material and damping parameters used in the moving parts of the driver, see the Loudspeaker Driver model documentation.

Results and Discussion

The model produces the local stresses and strains in all the moving parts, as well as the sound pressure distribution inside and outside the enclosure at all frequencies solved for. As an example of how you can visualize the deformations, Figure 3 includes them as a deformed surface plot.

One way of looking at the sound distribution is as in Figure 5, which displays the pressure at 1000 Hz as an isosurface plot. An alternative option is shown in Figure 6, where the sound pressure level in dB is plotted on a slice near the symmetry plane. In Figure 7 and Figure 8, the same plot types illustrate the solution at the highest frequency, 3550 Hz. At this frequency, there is a complex standing wave pattern inside the enclosure.

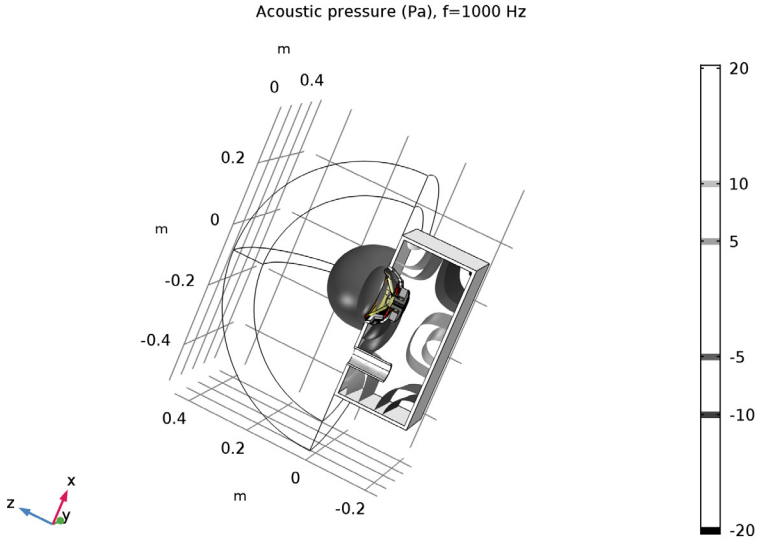


Figure 5: Isosurface plot of the sound pressure at 1000 Hz.

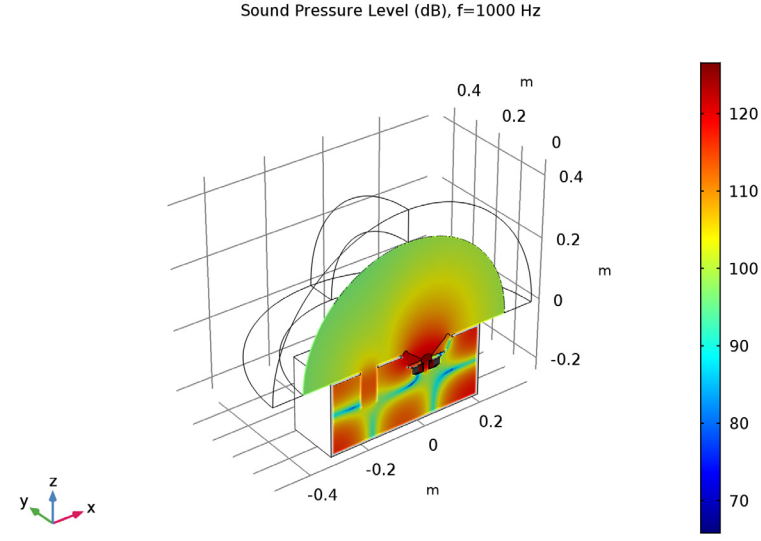


Figure 6: Slice plot of the sound pressure level at 1000 Hz.

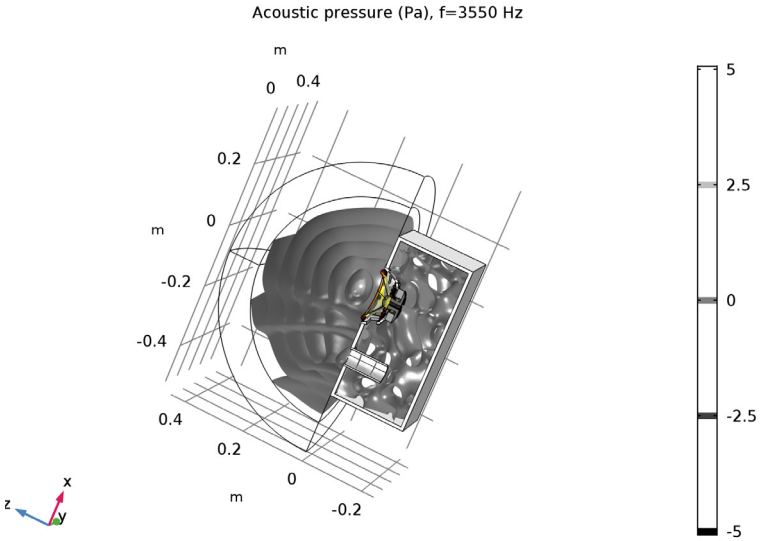


Figure 7: Isosurface plot of the sound pressure at 3550 Hz.

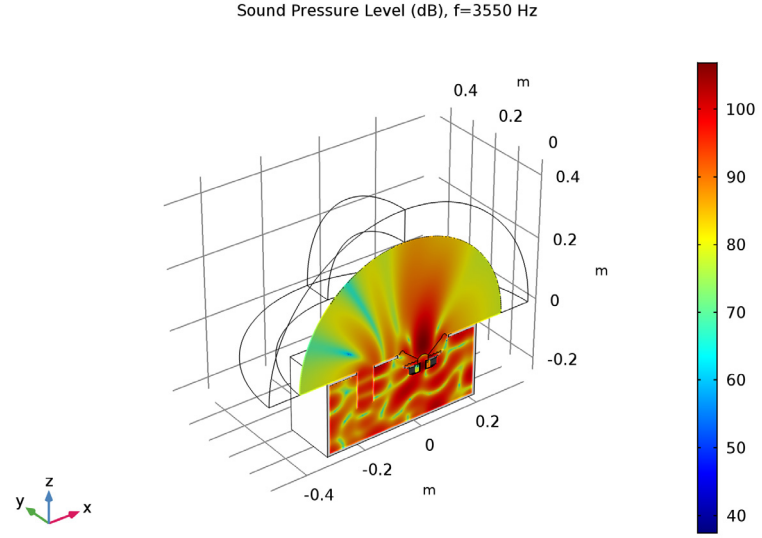


Figure 8: Slice plot of the sound pressure level at 3550 Hz.

Thanks to the full integral far-field evaluation available in COMSOL, you can evaluate the pressure not only inside the computational domain but also anywhere outside the domain. This makes it possible to, for instance, plot the sound pressure level at a given distance versus the elevation angle, or evaluate the directivity. The step-by-step instructions for this model show you how to plot the sensitivity (Figure 9) and the directivity (Figure 11).

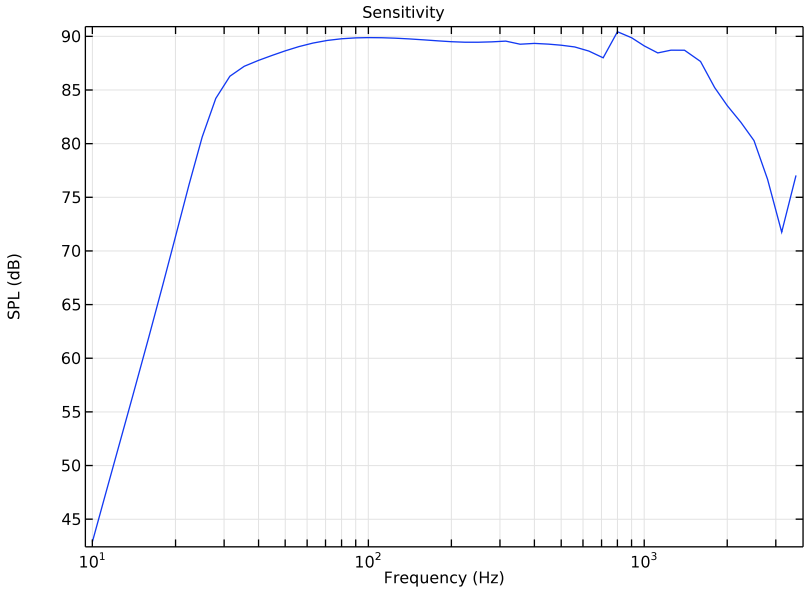


Figure 9: Loudspeaker sensitivity measured as the on-axis sound pressure level (dB) at a distance of 1 m from the unit. The pressure is evaluated using an RMS input signal of 2.83 V, corresponding to a power of 1 W at 8 Ω. Note the logarithmic scale for the frequency.

Compared to the sensitivity of the baffled driver alone (Figure 8 in the Loudspeaker Driver model), adding the enclosure clearly results in a "boost" for the lower frequencies, roughly in the range between 30 Hz and 100 Hz. As in the other model, the dip at around 700 Hz is due to a Helmholtz resonance in the cavity beneath the voice coil. This resonance would be less pronounced if the model were to account for the thermoviscous losses in the thin air gap between the magnet and the pole piece.

When using the far-field calculation feature it is also possible to use the dedicated far-field plots to display the spatial response of the speaker. They exist as both 1D polar plots, as 2D plots, and as 3D polar plots. A slightly modified version of the default generated 3D far-field polar plot is shown in Figure 10. The plot, evaluated for f = 1000 Hz, clearly shows that the speaker has a notch in the horizontal plane and a boost upwards and downward. The sensitivity curve in Figure 9 is evaluated at a fixed point 1 m in front of

the speaker for all frequencies and, thus, does not provide any spatial information about the speaker performance. This spatial information is visualized the 3D far-field plot (one frequency at the time) and is of course an important component in speaker design and optimization.

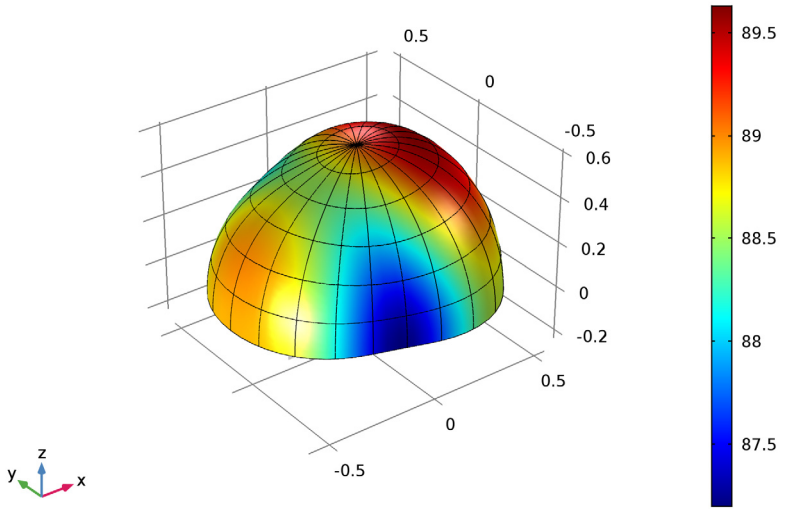


Figure 10: 3D polar plot of the speaker sensitivity at 1 m evaluated at 1000 Hz.

A plot that gives a mixed frequency and spatial information is the so-called directivity plot depicted in Figure 11. The response of the speaker is evaluated on a half circle 1 m in front of the speaker, the data is collected for all frequencies and plotted in this contour-like plot. The plot is predefined in the Acoustics Module and is simply called *Directivity*.

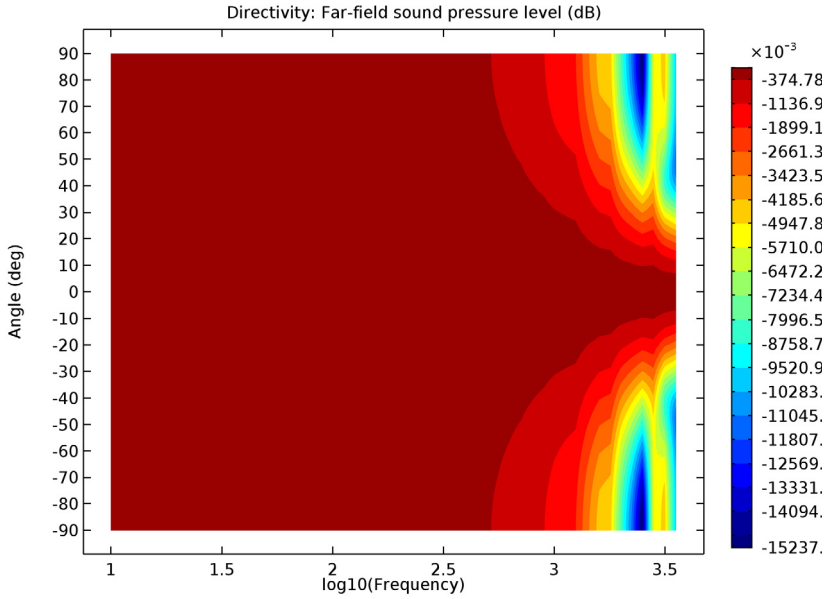


Figure 11: Directivity plot for the loudspeaker.

Notes About the COMSOL Implementation

This application requires the Acoustics Module and the Structural Mechanics Module.

The model instructions let you import the entire geometry, including the air and PML domains. In a similar real-life modeling situation it is usually straightforward to add air and PML domains to your own CAD drawing of the enclosure.

Application Library path: `Acoustics_Module/Electroacoustic_Transducers/vented_loudspeaker_enclosure`

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 **Acoustics>Acoustic-Structure Interaction>Acoustic-Shell Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

6 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 On the **Home** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `vented_loudspeaker_enclosure.mphbin`.

5 Click **Import**.

DEFINITIONS

To make it easier to select some important sets of domains and boundaries in a complicated geometry such as the one you are looking at, it is good modeling practice to begin by defining selections. When working on your own model, these selections are most conveniently defined by clicking and selecting directly in the geometry. The instructions however refer to them by numbers. Whenever a selection is made, it is therefore recommended that you use the Paste Selection button. Enter the number or list of numbers in the text field that appears. Input such as 5, 5-8, 13, or 4, 7, and 9 is accepted.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Model Domain` in the **Label** text field.

3 Select Domains 1, 2, and 5 only.

Because the magnetic engine and the walls of the enclosure will be considered rigid structures, there is no need to include their interior in the simulation.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Air Domain` in the **Label** text field.

3 Select Domain 2 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `PML Domain` in the **Label** text field.

3 Select Domains 1 and 5 only.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Cone` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

With wireframe rendering, you can see through boundaries and get a better view of which ones you are selecting.

5 Select Boundaries 50, 89, 97, and 108 only.

Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Spider` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 49, 55, 56, 64, 74, 77, 80, 81, 85, 107, 112, 116, 117, 119, 120, 126, 130, and 131 only.

Explicit 6

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Surround` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 35, 37, 133, and 135 only.

Explicit 7

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Apex` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 87, 88, 105, and 106 only.

Union 1

1 On the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `Shell Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Cone**, **Spider**, **Surround**, and **Apex**.

6 Click **OK**.

Next, define interpolation functions to bring in the blocked resistance and inductance from the model of the driver.

Interpolation 1 (int1)

1 On the **Definitions** toolbar, click **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `vented_loudspeaker_enclosure_Rb.txt`.

6 Click **Import**.

7 In the **Function name** text field, type `Rb`.

8 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

9 In the **Function** text field, type `ohm`.

10 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Nearest function**.

Set the extrapolation such that the results extend correctly just above the maximal frequency of 3500 Hz in the imported data. The simulation will run up to 3550 Hz.

Interpolation 2 (int2)

1 On the **Definitions** toolbar, click **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `vented_loudspeaker_enclosure_Lb.txt`.

6 Click **Import**.

7 In the **Function name** text field, type `Lb`.

8 Locate the **Units** section. In the **Arguments** text field, type `Hz`.

9 In the **Function** text field, type `H`.

10 Locate the **Interpolation and Extrapolation** section. From the **Extrapolation** list, choose **Nearest function**.

To enable extraction of the velocity and application of the electric force on the apex, define average and integration operators.

Integration 1 (intop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_apex` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Apex**.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, type `av_apex` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Apex**.

GLOBAL DEFINITIONS

Define parameters for the driving voltage, the BL factor from the loudspeaker driver model, the frequency at which the material losses are specified, and the wavelength at 3500 Hz (used to set the mesh size).

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
V0	4[V]	4 V	Driving Voltage
BL	7.55[N/A]	7.55 Wb/m	Force factor from loudspeaker driver model
f_loss	40[Hz]	40 Hz	Frequency at which loss factor is given
omega_loss	2*pi*f_loss	251.33 Hz	Angular frequency at which loss factor is given
lambda_min	343[m/s]/3500[Hz]	0.098 m	Wavelength at 3500 Hz (used in mesh)

DEFINITIONS

Next, create the expressions used in defining the electric driving force. The Description field is optional, but helps you keep track of what you are doing.

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
v0	av_apex(shell.u_tZ)	m/s	Cone velocity
vol_apex	2*int_apex(shell.d)	m^3	Full coil volume
Zb	Rb(freq)+acpr.iomega*Lb(freq)	Ω	Blocked coil impedance
Fe	BL*V0/Zb-v0*BL^2/Zb	N·m/m	Electric driving force

Before creating the materials for use in this model, it is a good idea to specify where you want to solve the pressure acoustics equation and also which boundaries constitute moving parts and thus are to be modeled as shells. Using this information, the software can detect which material properties are needed.

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 **Model Domain**.

The selection you just made removes the rigid structures from the volume where the pressure acoustics equation will be solved.

SHELL (SHELL)

On the **Physics** toolbar, click **Pressure Acoustics, Frequency Domain (acpr)** and choose **Shell (shell)**.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.

2 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Shell Boundaries**.

ADD MATERIAL

1 On 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>Air**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Material 2 (mat2)

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 `Composite` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Cone**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`140[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.42`	1	Basic
Density	rho	720	kg/m^3	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Cloth` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Spider**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`0.58[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	650	kg/m³	Basic

Material 4 (mat4)

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Foam` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Surround**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`1.6[MPa]`	Pa	Basic
Poisson's ratio	nu	`0.4`	1	Basic
Density	rho	67	kg/m³	Basic

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Copper**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat5)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Boundary**.

3 From the **Selection** list, choose **Apex**.

NOTE: In the 2D model of the driver, the apex and the voice coil that is wound around it are two separate domains. In the model at hand, they are lumped together and treated as Copper. Because the apex is not expected to deform considerably, this is fine as long as the shell thickness is tuned so that the total mass becomes the same as in the 2D model.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 On the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **PML Domain**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

5 Find the **Center coordinate** subsection. In the table, enter the following settings:

Xm (m)	Ym (m)	Zm (m)
-0.07	0	0

The default **Polynomial** stretching is kept as it ensures better convergence of the iterative solver used in the study.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

With the materials defined, it is now time to set up the remaining physics of the model. Begin by specifying the symmetry condition in the acoustics domain.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 Select Boundaries 1, 4, and 44 only.

The way the geometry is set up, there are two boundaries just outside the surround, separating the air inside and outside the enclosure. To avoid the air leaking through these boundaries, turn them into (interior) hard walls.

Interior Sound Hard Boundary (Wall) 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Interior Sound Hard Boundary (Wall)**.

2 Select Boundaries 33 and 136 only.

The acoustics physics is now ready, add a far-field calculation.

Far-Field Calculation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Far-Field Calculation**.

2 Select Boundaries 6 and 45 only.

3 In the **Settings** window for **Far-Field Calculation**, locate the **Far-Field Calculation** section.

4 Select the **Symmetry in the y=0 plane** check box.

5 Select the **Symmetry in the z=0 plane** check box.

6 From the **Type of integral** list, choose **Full integral**.

You have now supplied a source boundary encompassing all local sound sources and applied symmetry planes to account for the infinite baffle and the geometric symmetry. The Full integral setting lets you compute the pressure field (including phase) at any finite distance from speaker.

SHELL (SHELL)

Now set up the shell physics. Begin by specifying the individual thicknesses and damping properties of the moving parts of the driver.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Linear Elastic Material 1**.

Damping 1

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cone**.

4 Locate the **Damping Settings** section. From the **Damping type** list, choose **Isotropic loss factor**.

5 From the η_s list, choose **User defined**. In the associated text field, type `0.04`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Linear Elastic Material 1**.

Damping 2

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Spider**.

4 Locate the **Damping Settings** section. In the β_{dK} text field, type `0.14/omega_loss`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Linear Elastic Material 1**.

Damping 3

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Surround**.

4 Locate the **Damping Settings** section. In the β_{dK} text field, type `0.46/omega_loss`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Linear Elastic Material 1**.

Damping 4

1 On the **Physics** toolbar, click **Attributes** and choose **Damping**.

2 In the **Settings** window for **Damping**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Apex**.

4 Locate the **Damping Settings** section. From the **Damping type** list, choose **Isotropic loss factor**.

5 From the η_s list, choose **User defined**. In the associated text field, type `0.04`.

Change Thickness 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Change Thickness**.

2 In the **Settings** window for **Change Thickness**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cone**.

4 Locate the **Change Thickness** section. In the d text field, type `1[mm]`.

Change Thickness 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Change Thickness**.

2 In the **Settings** window for **Change Thickness**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Spider**.

4 Locate the **Change Thickness** section. In the d text field, type `0.35[mm]`.

Change Thickness 3

1 On the **Physics** toolbar, click **Boundaries** and choose **Change Thickness**.

2 In the **Settings** window for **Change Thickness**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Surround**.

4 Locate the **Change Thickness** section. In the d text field, type `1.4[mm]`.

Change Thickness 4

1 On the **Physics** toolbar, click **Boundaries** and choose **Change Thickness**.

2 In the **Settings** window for **Change Thickness**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Apex**.

4 Locate the **Change Thickness** section. In the d text field, type `0.166[mm]`.

Next, apply the electric load onto the apex. Because the load is entered as a volume force density, you need to divide the force by the volume of the apex.

Body Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Body Load**.

2 In the **Settings** window for **Body Load**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Apex**.

4 Locate the **Force** section. Specify the $\mathbf{F}_V$ vector as

`0`	`x`
`0`	`y`
`Fe/vol_apex`	`z`

Fix the outer rim of the spider and the surround.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.

2 Select Edges 64, 94, 264, and 275 only.

Keep in mind that you can use the **Paste Selection** button for all selections. This is also good idea for the following edge setting, which serves to impose the mirror symmetry.

Symmetry 1

1 On the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 Select Edges 63, 68, 93, 95, 107, 109, 124, 139, 147, 155, 157, 164, 169, 171, 173, 196, 311–314, 318, 322, 323, 332, 336, 345, 357, 358, 368, and 376 only.

3 In the **Settings** window for **Symmetry**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Global coordinate system**.

Note that the **Axis** to use as symmetry plane normal is set to 2. This means that the solution will be symmetric with respect to the xz-plane.

You have now set up all the relevant physics. This may be a good time to have a second look at all the features of the model, to verify that all settings make sense and apply to the right domains or boundaries. In particular, you can check the **Multiphysics>Acoustic-Structure Boundary 1** node, which has been set automatically.

MESH 1

The mesh needs to resolve the fine details of the geometry as well as the waves at all frequencies. Furthermore, the PML should have at least 5 elements across its thickness (for the rational scaling). To improve the far-field evaluation add a thin boundary layer, of thickness `lambda_min/5/10`, adjacent to the PML domain. Turn off the smooth transition option - the single layer is used to get a good normal gradient evaluation. These requirements are all met by the following settings.

Boundary Layers 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Air Domain**.

5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.

3 In the **Number of boundary layers** text field, type `1`.

4 Select Boundaries 6 and 45 only.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `lambda_min/5/10`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, click to expand the **Element size parameters** section.

3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambda_min/5`.

This gives you a minimum of 5 elements per wavelength at the highest frequency, 3500 Hz. In practice, remembar that a mesh convergence analysis is always recommended.

4 In the **Minimum element size** text field, type `1[mm]`.

The shortest edges of the geometry have length 1 mm, which you will resolve with this setting. You can increase the **Minimum element size** setting to get a coarser mesh in the narrow region. This can trigger a warning as COMSOL checks if the minimum size is larger than a gometric entity. The model will solve when a warning is present. It is up to the user to decide if the warning is relevant for the quality of the simulation results.

Use a swept mesh for the PML (with the size settings you will get 6 elements in the thickness). For more details on meshing PMLs see the modeling sections in the Acoustics Module User's Guide.

Swept 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **PML Domain**.

Distribution 1

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type `8`.

Use the recommended 8 mesh elements in the PML when using the polynomial stretching type.

Finally, mesh the loudspeaker box. The mesh here is only for post processing purposes to be able to plot the box as a solid structure. It is not used in the computation.

Free Tetrahedral 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

STUDY 1

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** check box.

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 Click **Range**.

4 In the **Range** dialog box, choose **ISO preferred frequencies** from the **Entry method** list.

5 In the **Start frequency** text field, type 10.

6 In the **Stop frequency** text field, type 3550.

7 From the **Interval** list, choose **1/6 octave**.

8 Click **Replace**.

This gives you frequencies with a 1/6 octave resolution from 10 to 3550 Hz with values specificed by the ISO standard. The model takes about 5 GB of RAM to solve and will solve in around 40 min (depending on your hardware). If you are short on time you can, for example, select 1/3 octave or simply octave spacing.

Generate the default solver sequence, then expand and enable the suggested iterative solve for this acoustic-shell interaction problem. The iterative solver is both faster and more memory efficient than the default direct solver.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (GMRES with Direct Precon.) (asb1)** and choose **Enable**.

5 On the **Study** toolbar, click **Compute**.

RESULTS

3D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 Select the **Allow evaluation of expressions** check box.

5 In the **Precision** text field, type 4.

6 In the **Title** text area, type `Total displacement (m), f=eval(freq,Hz) Hz`.

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>Shell>Displacement> shell.disp - Total displacement**.

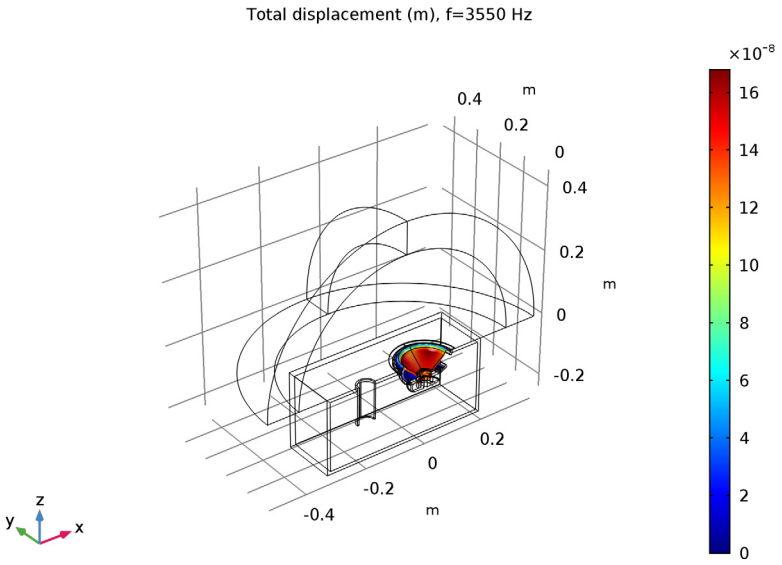
Surface 1

1 Right-click **Results>Displacement>Surface 1** and choose **Deformation**.

2 In the **Model Builder** window, under **Results>Displacement** click **Surface 1**.

3 On the **Displacement** toolbar, click **Plot**.

4 Click the **Go to Default View** button on the **Graphics** toolbar.



This first plot shows how the shell displaces. The colors represent the total displacement and the deformation shows the position of the membrane at zero phase. Try looking at a couple of different solutions to see how the displacement amplitude and phase change with the frequency.

5 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

6 From the **Color table** list, choose **Thermal**.

7 On the **Displacement** toolbar, click **Plot**.

Displacement

1 In the **Model Builder** window, under **Results** click **Displacement**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **10**.

4 On the **Displacement** toolbar, click **Plot**.

5 From the **Parameter value (freq (Hz))** list, choose **3550**.

6 On the **Displacement** toolbar, click **Plot**.

Data Sets

In order to display the box and reproduce Figure 3, create a new data set defined only in the solid parts of the model.

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Data Sets** node.

Study 1/Solution 1 (2) (sol1)

1 Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

2 In the **Settings** window for **Solution**, type `Study 1/Solution 1 - Speaker box` in the **Label** text field.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 3, 4, 6, and 7 only.

Volume 1

1 In the **Model Builder** window, under **Results** right-click **Displacement** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 - Speaker box (sol1)**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Geometry>dom - Entity index**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

6 Clear the **Color legend** check box.

7 Select the **Reverse color table** check box.

8 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

By plotting the domain number in grayscale, you have now added a black and white visualization of the enclosure and the solid parts of the driver. Lowering the resolution makes the visualization faster. If you rotate the plot a little, it should now resemble Figure 3.

Next, create a plot of the sound pressure level in a centered slice of the geometry. It is convenient to copy and paste the plot group from the first plot.

Displacement 1

1 Right-click **Displacement** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Sound Pressure Level` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Sound Pressure Level (dB), f=eval(freq,Hz) Hz`.

Surface 1

1 In the **Model Builder** window, expand the **Results>Sound Pressure Level** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

Slice 1

1 In the **Model Builder** window, under **Results** right-click **Sound Pressure Level** and choose **Slice**.

2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Pressure Acoustics, Frequency Domain>Pressure and sound pressure level>acpr.Lp - Sound pressure level**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 Click to expand the **Quality** section. From the **Smoothing** list, choose **None**.

The smoothing algorithm will attempt to make the pressure continuous across the membrane. You get a sharper plot if you remove it.

6 On the **Sound Pressure Level** toolbar, click **Plot**.

The plot range is dominated by the large pressure drop in the PML. You can hide the PML domain by letting the solution data set be defined only in the air.

Study 1/Solution 1 (3) (sol1)

1 On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

2 In the **Settings** window for **Solution**, type `Study 1/Solution 1 - Air domain` in the **Label** text field.

Selection

1 On the **Results** toolbar, click **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Air Domain**.

Sound Pressure Level

1 In the **Model Builder** window, under **Results** click **Sound Pressure Level**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Data set** list, choose **Study 1/Solution 1 - Air domain (sol1)**.

4 On the **Sound Pressure Level** toolbar, click **Plot**.

5 Click the **Go to Default View** button on the **Graphics** toolbar.

The plot should now like Figure 8. Switch to 1000 Hz to reproduce Figure 6.

6 From the **Parameter value (freq (Hz))** list, choose **1000**.

7 On the **Sound Pressure Level** toolbar, click **Plot**.

Create a third plot group to show the 3D far-field polar plot. It is the spatial sensitivity characteristic evaluated at a distance of 1 m and at 1000 Hz.

3D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Far-Field Sound Pressure Level` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **1000**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Far Field 1

1 On the **Far-Field Sound Pressure Level** toolbar, click **More Plots** and choose **Far Field**.

2 In the **Settings** window for **Far Field**, locate the **Expression** section.

3 In the **Expression** text field, type `(acpr.ffc1.Lp_pfar-77)/20`.

4 Clear the **Use as color expression** check box.

5 Locate the **Evaluation** section. Find the **Angles** subsection. In the **Number of elevation angles** text field, type `30`.

6 In the **Number of azimuth angles** text field, type `60`.

7 From the **Restriction** list, choose **Manual**.

8 In the θ **range** text field, type `90`.

9 Locate the **Coloring and Style** section. From the **Grid** list, choose **Fine**.

The first expression defines the shape of the 3D polar plot. In this case it is modified to set the reference at 77 dB and then scaled by a factor of 20 in order to visualize it relative to the model geometry. Changing the reference to 77 dB enhances the visualization of the notches. The surface color (second expression, enabled with the check box) gives the true SPL at 1 m from the speaker. Change the evaluation frequency to see how the directivity of the speaker changes with frequency. Set the expression back to `acpr.ffc1.Lp_pfar` and use the **Zoom Extents** tool.

Volume 1

In the **Model Builder** window, under **Results>Displacement** right-click **Volume 1** and choose **Copy**.

Volume 1

1 In the **Model Builder** window, under **Results** right-click **Far-Field Sound Pressure Level** and choose **Paste Volume**.

2 On the **Far-Field Sound Pressure Level** toolbar, click **Plot**.

3 Click the **Go to Default View** button on the **Graphics** toolbar.

Rotate the plot in order to reproduce Figure 10.

Now, plot the sensitivity versus the frequency and reproduce Figure 9 as follows.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Sensitivity` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Sensitivity`.

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Frequency (Hz)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `SPL (dB)`.

Octave Band 1

1 On the **Sensitivity** toolbar, click **More Plots** and choose **Octave Band**.

2 In the **Settings** window for **Octave Band**, locate the **Selection** section.

3 From the **Geometric entity level** list, choose **Global**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `pfar(0,0,1)`.

5 Locate the **Plot** section. From the **Style** list, choose **Continuous**.

This will plot the sound pressure level using the reference pressure, which defaults to the commonly used value 20 μPa. The `pfar(x,y,z)` operator extracts the sound pressure in (x,y,z) = (0,0,1[m]).

You can change the style to 1/3 octave or octave to plot the sensitivity in bands if needed.

6 On the **Sensitivity** toolbar, click **Plot**.

The next set of instructions show you how to create an isosurface plot of the pressure distribution in and outside of the speaker. The last plot is the directivity plot for the speaker.

Displacement 1

1 In the **Model Builder** window, under **Results** right-click **Displacement** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Acoustic Pressure` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Acoustic pressure (Pa), f=eval(freq,Hz) Hz`.

4 Locate the **Data** section. From the **Data set** list, choose **Study 1/Solution 1 - Air domain (sol1)**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Acoustic Pressure** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

Isosurface 1

1 In the **Model Builder** window, under **Results** right-click **Acoustic Pressure** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, locate the **Levels** section.

3 From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `-5 -2.5 0 2.5 5`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

6 On the **Acoustic Pressure** toolbar, click **Plot**.

7 Click the **Zoom Extents** button on the **Graphics** toolbar.

The plot should resemble Figure 5. To reproduce Figure 7, change the frequency and the levels for the isosurface plot.

Acoustic Pressure

1 In the **Model Builder** window, under **Results** click **Acoustic Pressure**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **1000**.

Isosurface 1

1 In the **Model Builder** window, under **Results>Acoustic Pressure** click **Isosurface 1**.

2 In the **Settings** window for **Isosurface**, locate the **Levels** section.

3 In the **Levels** text field, type `-20 -10 -5 5 10 20`.

4 On the **Acoustic Pressure** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Finally, create the directivity plot depicted in Figure 11 for the loudspeaker.

2D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Directivity` in the **Label** text field.

Directivity 1

1 On the **Directivity** toolbar, click **More Plots** and choose **Directivity**.

2 In the **Settings** window for **Directivity**, 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 **Number of angles** text field, type `90`.

7 Find the **Reference direction** subsection. In the **x** text field, type `0`.

8 In the **y** text field, type `0`.

9 In the **z** text field, type `1`.

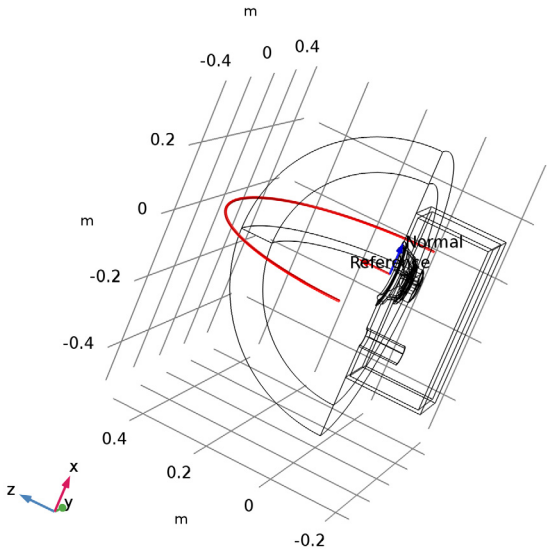
10 Find the **Normal** subsection. In the **x** text field, type 1.

11 In the **y** text field, type 0.

12 In the **z** text field, type 0.

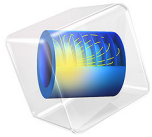
These settings give a directivity plot taken in the yz-plane where 0 deg. correspond to the z-axis direction. The evaluation plane can easily be visualized using the **Preview Evaluation Plane** feature. The radius of the red evaluation circle displayed is normalized relative to the geometry.

13 Click **Preview Evaluation Plane**.



14 On the **Directivity** toolbar, click **Plot**.

15 Click the **Zoom Extents** button on the **Graphics** toolbar.



Vibrating Micromirror with Viscous and Thermal Damping

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Micromirror systems are part of small MEMS (micro-electro-mechanical systems) systems for integrated scanning micromirror systems in telecommunications and consumer applications. The mirrors have low power consumption and are low cost to manufacture. Their application range from microscopy applications, medical imaging, scanners, and heads up displays, to some fiber optics applications.

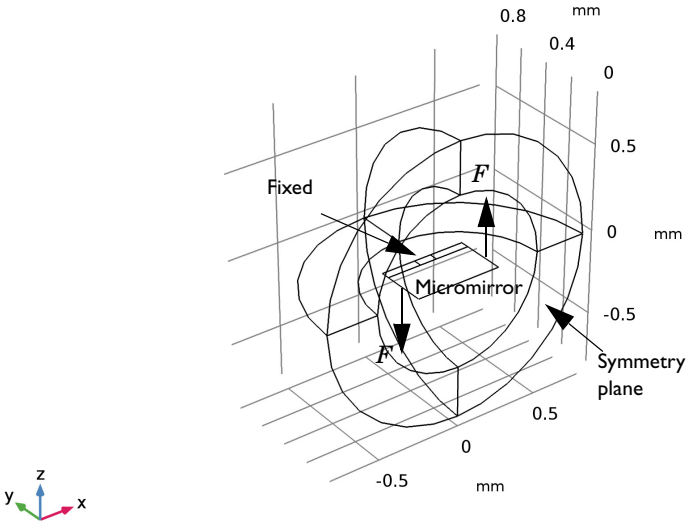


Figure 1: Geometry of the micromirror system including the fixed constraint, the torquing force components F, and the symmetry plane.

The dynamic vibrating behavior of the micromirror and especially the damping is an important factor affecting the design of their actuator systems. In the development process of new micromirror systems, modeling the damping correctly, and thus predicting the performance of the system, can save both time and money.

Model Definition

A sketch of the idealized micromirror system modeled here is shown in Figure 1. The mirror is made of silicon and is surrounded by air. It is 0.5 mm by 0.5 mm (only half is shown in the figure because of symmetry) and has a thickness of 1 μm.

The solid mirror is modeled using shell elements and the surrounding air is modeled using thermoviscous acoustics. A pressure acoustics layer is used to truncate the computational domain.The model combines the Shell interface, the Thermoviscous Acoustics, Frequency Domain interface, and the Pressure Acoustics, Frequency Domain interface. The three physics interfaces are set up and combined using multiphysics couplings: the Acoustics-Thermoviscous Acoustics Boundary coupling and the Thermoviscous Acoustics-Structure Boundary coupling features, respectively.

One goal with the model is to get a correct assessment of the damping the mirror experiences. Therefore, the detailed thermoviscous acoustics is used to model the air domain surrounding the micromirror. The interface includes thermal and viscous damping explicitly as it solves the full linearized Navier-Stokes, continuity, and energy equations, implemented in the Thermoviscous Acoustics, Frequency Domain interface.

An important parameter in such models is the thickness of the viscous and thermal boundary layers (also called the penetration depths). It is in these layers that the energy is dissipated (viscous drag and thermal conduction). For air, the layers are of approximately the same size. The thickness of the viscous boundary layer δ_v is given by the expression

$$\delta_v = \sqrt{\frac{2\mu}{\omega_0 \rho_0}} = 0.22[\text{mm}] \sqrt{\frac{100[\text{Hz}]}{f_0}} \tag{1}$$

since $\delta_v = 0.22$ mm at 100 Hz. The boundary layer thickness may also be evaluated in post processing through the variables `tash.d_visc` and `tash.d_therm`.

The model is both solved using a frequency-domain sweep and using an eigenfrequency study. The frequency sweep results in a frequency response where the displacement (or velocity) is evaluated at the tip of the mirror (on the symmetry plane).

The response at the resonance yields the Q-factor Q_r and the resonance frequency f_r. Their expected values are given in the parameters list. The Q-factor is defined as

$$Q = \frac{f_0}{\Delta f}$$

where f_0 is the resonance frequency and Δf is the peak width at half power (which corresponds to the 3 dB down width). The Q-factor is related to the attenuation rate α through

$$\alpha = \frac{\omega_0}{2Q} \qquad \omega_0 = 2\pi f_0$$

The result of the eigenfrequency study is a complex valued eigenfrequency f_c which relates to the other parameters through

$$f_0 = \mathrm{Re}(f_c) \qquad \alpha = 2\pi \mathrm{Im}(f_c) \tag{2}$$

Finally, the response of an underdamped harmonic oscillator can be written as

$$\frac{v}{F} = \frac{\omega_0^2}{(i\omega)^2 + 2\alpha i\omega + \omega_0^2} \tag{3}$$

where F is the actuating force and v is the velocity of the oscillator. This expression is used as a model equation for the frequency response in the vicinity of the resonance frequency; based on parameters identified in the eigenfrequency analysis (using the relations in Equation 2).

Results and Discussion

The displacement of the micromirror with the torquing actuation is shown in Figure 2 at a frequency of 10 kHz. The acoustic temperature variations and the acoustic pressure distribution at 11 kHz is shown in Figure 3. The instantaneous local velocity (acoustic variations in the velocity) is shown in Figure 4. A high velocity region is seen near the edge of the micromirror. The extent of this region into the surrounding air is given by the scale of the viscous boundary layer (also known as the viscous penetration depth, Equation 1). The thermal boundary layer (thermal penetration depth) can in the same way be identified in Figure 3 (top).

The displacement response of the system is plotted in Figure 5 (top). This plot shows the absolute value of the z-component of the displacement field $|w_{\mathrm{shell}}|$ evaluated at the tip of the micromirror. The bottom part of this figure shows the (scaled) power response, on the dB scale, by plotting

$$20\log(|i\omega w_{\mathrm{shell}}|)$$

and comparing to the model expression given in Equation 3. A manually set scaling factor, accounting for the magnitude of the actuation force, is subtracted to give the same magnitudes. The parameters used in the model expression stem from the eigenfrequency derived in the eigenfrequency study. Note that the two curves do not match perfectly. This is because the value of the eigenfrequency f_c used in the model curve (`f_num` from the parameters list **Global Definitions > Parameters**), was derived using a finer mesh than in the default setup of this model. The size of the mesh on the surface of the mirror is set to

`1.2*dvisc` which is not fine enough; a value of `0.5*dvisc` is better suited to well resolve the acoustic boundary layer. The coarse mesh is selected here to reduce the required memory to about 4 GB of RAM. The model would require about 13.5 GB of RAM using the fine mesh suggestion. The step-by-step instructions describe how to enable and use one of the suggested iterative solvers for the frequency domain model.

Some of the resulting modes from the eigenfrequency study are shown in Figure 6. The torquing mode studied in the frequency sweep is the top figure where f_c = 10560+145i, while a symmetric vibrating mode is found at about 13 kHz.

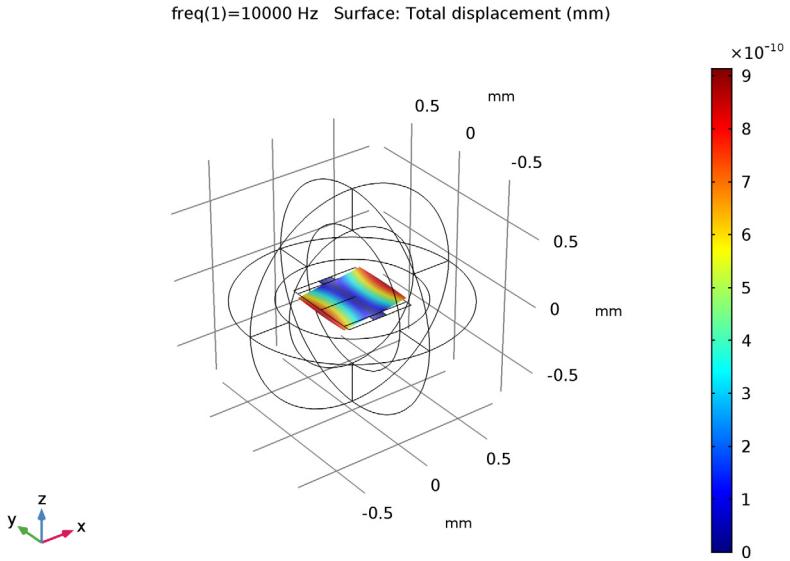


Figure 2: Displacement of the micromirror at 10 kHz subject to the torquing force.

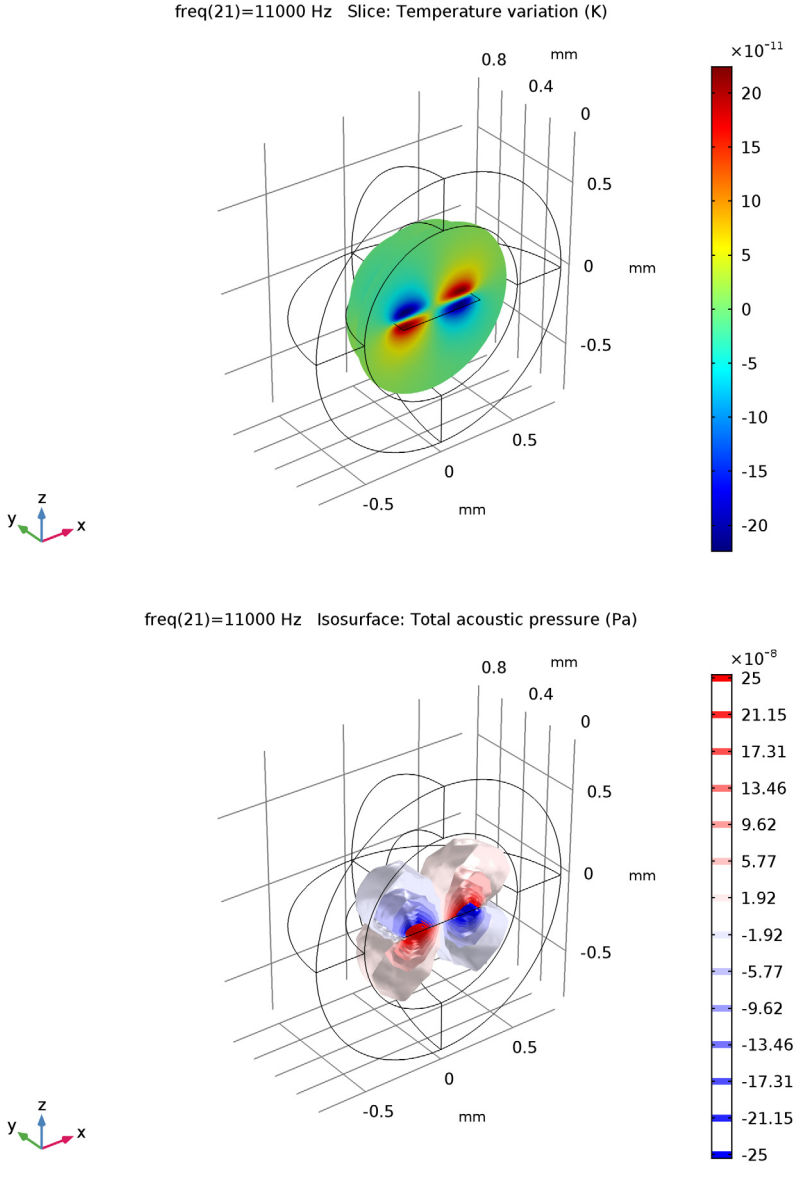


Figure 3: (top) Temperature field in the thermovicous acoustics domain around the micromirror, and (bottom) pressure isosurfaces showing the anti-symmetric pressure distribution.

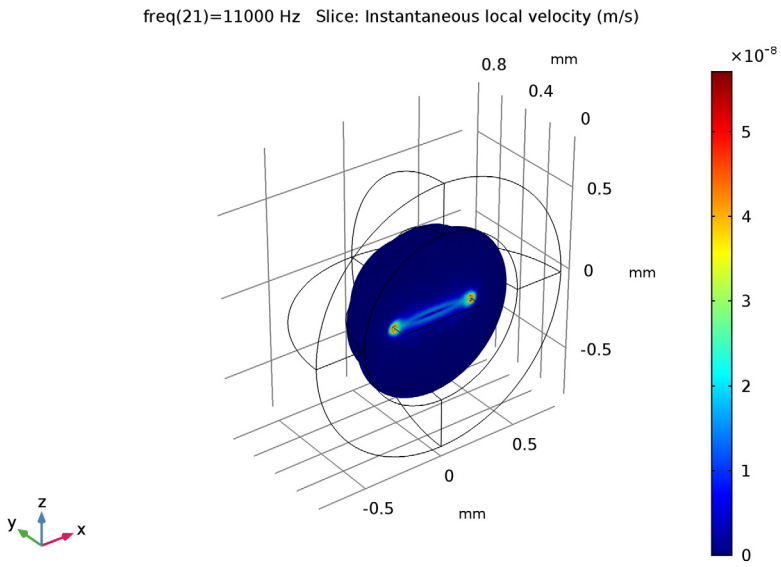


Figure 4: The instantaneous local velocity, with clear high velocity regions near the mirror edges. The extent of the high velocity region is given by the scale of the acoustic viscous boundary layer.

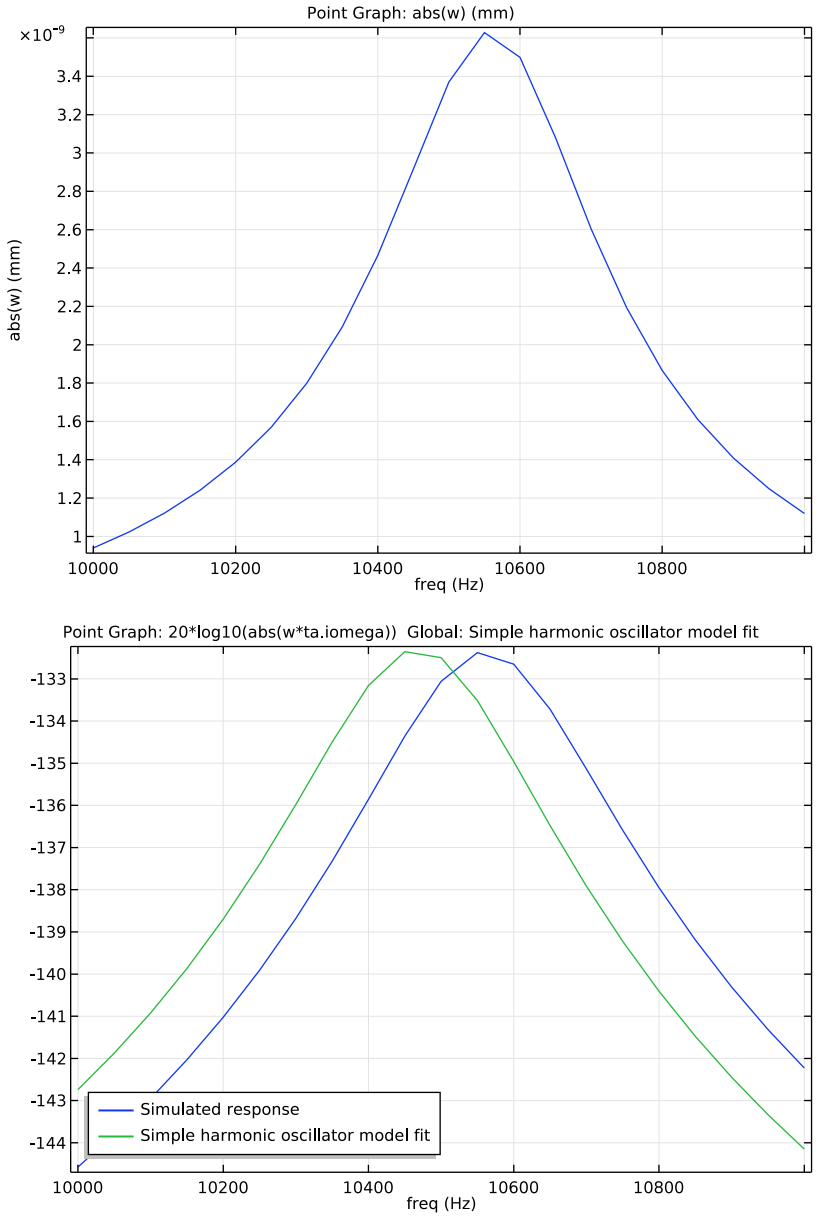


Figure 5: (top) Displacement response, and (bottom) power response with model curve.

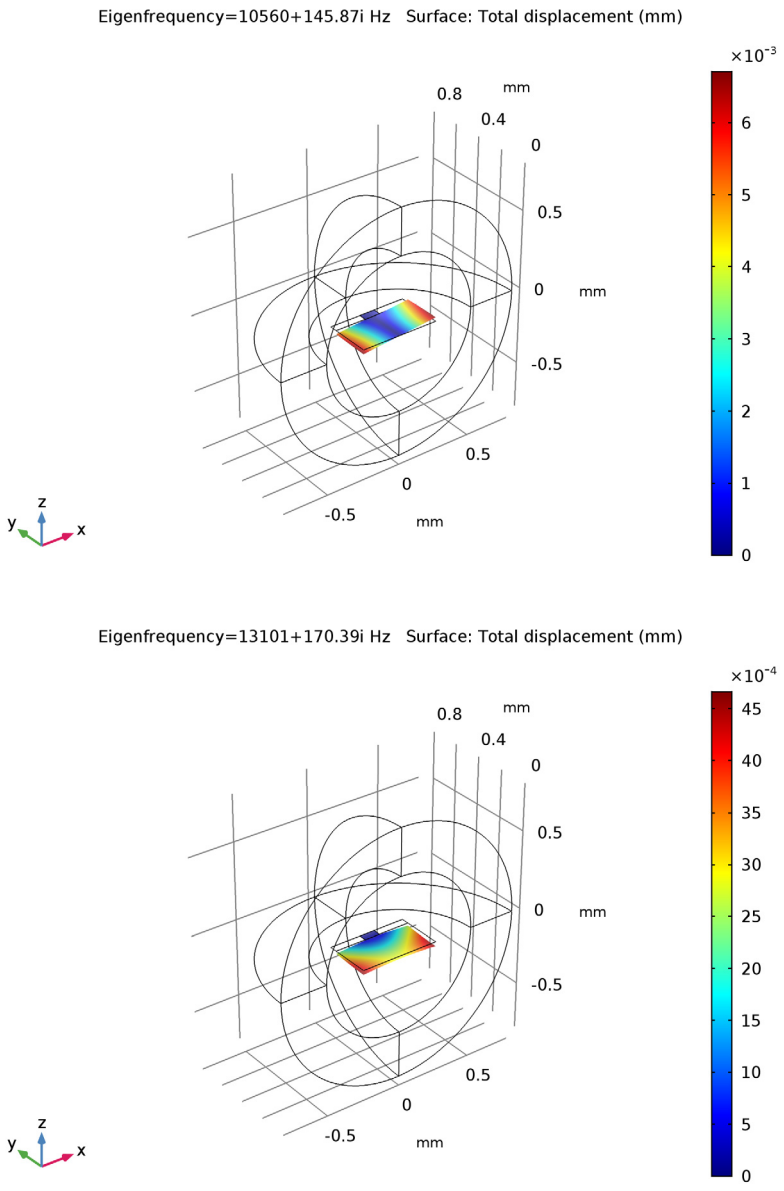


Figure 6: Torquing (top) and symmetric (bottom) vibrating modes of the micromirror.

Application Library path: `Acoustics_Module/Vibrations_and_FSI/vibrating_micromirror`

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>Shell (shell)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics> Thermoviscous Acoustics, Frequency Domain (ta)**.
5. Click **Add**.
6. In the **Select Physics** tree, select **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.
7. Click **Add**.
8. Click **Study**.
9. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.
10. Click **Done**.

GLOBAL DEFINITIONS

Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters**.
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 `vibrating_micromirror_parameters.txt`.

GEOMETRY 1

The geometry sequence for the model (see Figure 1) is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Appendix—Geometry Modeling Instructions. section. Otherwise, insert the geometry sequence as follows:

1. On the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `vibrating_micromirror_geom_sequence.mph`.
3. On the **Geometry** toolbar, click **Build All**.
4. Click the **Go to Default View** button on the **Graphics** toolbar.
5. Click the **Wireframe Rendering** button on the **Graphics** toolbar.

DEFINITIONS

Explicit 1

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Shell` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 9 and 11 only.

Explicit 2

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Symmetry` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 1, 2, 6, 13, and 17 only.

Explicit 3

1. On the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Symmetry Edge` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.
4. Select Edge 9 only.

ADD MATERIAL

1. On 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>Air**.
4. Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	0	Pa·s	Basic

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Silicon**.

3 Click **Add to Component** in the window toolbar.

4 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Silicon (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Geometric entity level** list, choose **Boundary**.

3 From the **Selection** list, choose **Shell**.

SHELL (SHELL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Shell (shell)**.

2 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Shell**.

4 Locate the **Thickness** section. In the d text field, type `h_mirror`.

Fixed Constraint 1

1 On the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.

2 Select Edge 15 only.

Symmetry 1

1 On the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Symmetry Edge**.

Body Load 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Body Load**.

2 In the **Settings** window for **Body Load**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Shell**.

4 Locate the **Force** section. Specify the $\mathbf{F}_V$ vector as

0	x
0	y
x/(0.25[mm])*1[N/m^3]	z

This corresponds to a torquing force proportional to the x coordinate acting in the z direction. A load of 0 N/m^3 act in the middle (at $x = 0$) rising to 1 N/m^3 at the mirror edge.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thermoviscous Acoustics, Frequency Domain (ta)**.

2 In the **Settings** window for **Thermoviscous Acoustics, Frequency Domain**, locate the **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 3 only.

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

2 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry**.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 Select Domains 1, 2, 4, and 5 only.

3 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.

4 In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.

5 From the **Selection** list, choose **Symmetry**.

Spherical Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Spherical Wave Radiation**.

2 Select Boundaries 4, 5, 14, and 19 only.

Proceed to set up the Multiphysics couplings that couple Thermoviscous Acoustics to Pressure Acoustics and couple the interior shell to the surrounding thermoviscous acoustics domain.

MULTIPHYSICS

Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)

1 On the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic-Thermoviscous Acoustic Boundary**.

2 In the **Settings** window for **Acoustic-Thermoviscous Acoustic Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

When you select **All boundaries** COMSOL automatically detects where a thermoviscous acoustics domain and a pressure acoustics domain meet and applies the coupling there.

Thermoviscous Acoustic-Structure Boundary 1 (tsb1)

1 On the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Thermoviscous Acoustic-Structure Boundary**.

2 In the **Settings** window for **Thermoviscous Acoustic-Structure Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Shell**.

Selecting **All boundaries** here will result in the same setup as using the Shell selection.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Shell**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 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. Select the **Maximum element size** check box.

5 In the associated text field, type `1.2*dvisc`.

This is an important mesh setting as it determines the resolution of the acoustic boundary layer. A coarse setting is of the order `1.2*dvisc` and will solve with about 3 to 4 GB of RAM. A well resolved setting of maybe `0.5*dvisc` will result in a model that requires about 12 GB of RAM to solve.

6 Select the **Minimum element size** check box.

7 In the associated text field, type `dvisc/10`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** 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 `343[m/s]/f0/5`.

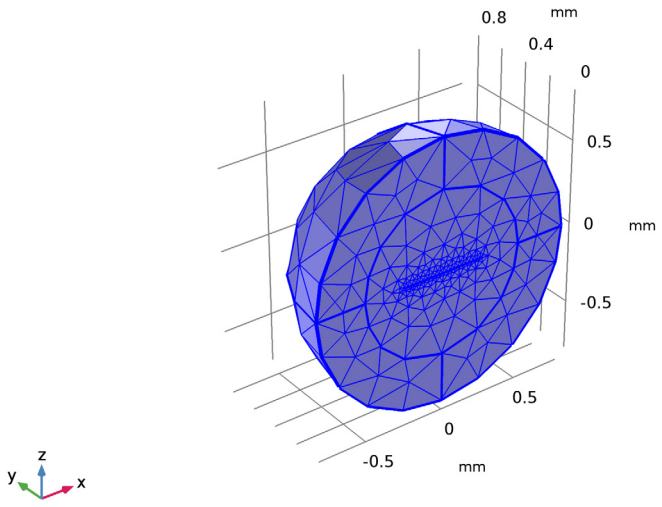
5 In the **Resolution of narrow regions** text field, type `1`.

Free Tetrahedral 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

The geometry with mesh should look like the figure below.



STUDY 1

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(f0-deltaf,50,f0+deltaf)`.

Turn off the generation of default plots and only create the ones you need. Plot the acoustic variations in temperature, pressure, and velocity as well as response curves. If turned on, the default plots for each physics interface will be generated.

3 In the **Model Builder** window, click **Study 1**.

4 In the **Settings** window for **Study**, locate the **Study Settings** section.

5 Clear the **Generate default plots** check box.

Now, generate and show the default sover to take a look at the solver suggestions automatically generated.

Solution 1 (sol1)

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

In this model, that couples pressure acoustics, thermoviscous acoustics and shell, the default solver is a direct solver. The second and third suggestions can be selected for the frequency domain sweep, but they are not recommended for the eigenfrequency study. Select and enable the first iterative suggestion. It is faster and more memory efficient than the direct solver. With this solver, you can also try and refine the surface mesh to the suggested `0.5*dvisc`. Solving the model with the finer mesh will require about 13.5 GB of RAM.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (GMRES with Direct Precon.) (tsb1_atb1)** and choose **Enable**.

5 On the **Study** toolbar, click **Compute**.

Solving the model takes, depending on your hardware, around 6 minutes and requires about 3.8 GB of RAM with the current mesh. Solving with the default direct solver requires about 4 GB of RAM and takes around 10 min (again depending on your hardware).

RESULTS

Data Sets

In the **Model Builder** window, expand the **Results** node.

Mirror 3D 1

1 On the **Results** toolbar, click **More Data Sets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 From the **Plane** list, choose **XZ-planes**.

3D Plot Group 1

1 On 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 **Data set** list, choose **Mirror 3D 1**.

Displacement

1 Right-click **Displacement** and choose **Surface**.

2 In the **Model Builder** window, under **Results>Displacement** right-click **Surface 1** and choose **Deformation**.

3 In the **Model Builder** window, under **Results** click **Displacement**.

4 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

5 From the **Parameter value (freq (Hz))** list, choose **10000**.

6 On the **Displacement** toolbar, click **Plot**.

The plot should look like the one in Figure 2.

3D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Temperature` in the **Label** text field.

Slice 1

1 Right-click **Temperature** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `T`.

4 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.

5 On the **Temperature** toolbar, click **Plot**.

The plot should look like the one in Figure 3 top.

3D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Pressure` in the **Label** text field.

Isosurface 1

1 Right-click **Pressure** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, locate the **Expression** section.

3 In the **Expression** text field, type `ta.p_t`.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `range(-2.5e-7,5.0e-7/13,2.5e-7)`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

Isosurface 2

1 Right-click **Results>Pressure>Isosurface 1** and choose **Duplicate**.

2 In the **Settings** window for **Isosurface**, locate the **Expression** section.

3 In the **Expression** text field, type `acpr.p_t`.

4 Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Isosurface 1**.

5 On the **Pressure** toolbar, click **Plot**.

The plot should look like the one in Figure 3 bottom.

Proceed to setting up two 1D curves that represent the frequency response of the system by probing the displacement at the symmetry plane of the micromirror. The first plot gives the abs value of the displacement. The second plot depicts the power (proportional to the velocity squared) on the dB scale and compared to a simple harmonic oscillator model tuned using parameters extracted from an eigenfrequency study of the system (instructions follow below). The parameters are given in the parameters list. Note that they are based on a fine mesh resolution of `0.5*dvisc`.

Temperature 1

1 In the **Model Builder** window, under **Results** right-click **Temperature** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Velocity` in the **Label** text field.

Slice 1

1 In the **Model Builder** window, expand the **Results>Velocity** node, then click **Slice 1**.

2 In the **Settings** window for **Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `ta.v_inst`.

4 On the **Velocity** toolbar, click **Plot**.

The plot should look like the one in Figure 4.

1D Plot Group 5

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Displacement response` in the **Label** text field.

Point Graph 1

1 Right-click **Displacement response** and choose **Point Graph**.

2 Select Point 16 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `abs(w)`.

5 On the **Displacement response** toolbar, click **Plot**.

The plot should look like the one in Figure 5 top.

1D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Power response` in the **Label** text field.

3 Click to expand the **Legend** section. From the **Position** list, choose **Lower left**.

Point Graph 1

1 Right-click **Power response** and choose **Point Graph**.

2 Select Point 16 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `20*log10(abs(w*ta.iomega))`.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Simulated response`

Global 1

1 In the **Model Builder** window, under **Results** right-click **Power response** 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	Description
`20*log10(abs(omega_r^2/((ta.iomega)^2+2*alpha_num*ta.iomega+omega_r^2)))-163.5`	`Simple harmonic oscillator model fit`

4 On the **Power response** toolbar, click **Plot**.

The plot should look like the one in Figure 5 bottom.

Proceed to set up an eigenfrequency study. The eigenfrequency including the complex part yields information about the attenuation in the system. The value of `f_num` as given in the **Definitions>Parameters** list corresponds to a finer mesh than the default used here.

ADD STUDY

1 On 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 **Preset Studies> Eigenfrequency**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Eigenfrequency

1 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

2 In the **Search for eigenfrequencies around** text field, type `10000`.

3 From the **Eigenfrequency search method around shift** list, choose **Larger real part**.

4 Select the **Desired number of eigenfrequencies** check box.

5 In the associated text field, type `3`.

These selections will search for the first three eigenmodes above 10 kHz. As we know from the frequency domain analysis, the first eigenfrequency is at about 10500 Hz. The eigenfrequency search condition will make the solution converge faster. Selecting the **Closest in absolute value** will search for modes with both lower and larger eigenfrequencies.

Again, turn off the generation of the default plots.

6 In the **Model Builder** window, click **Study 2**.

7 In the **Settings** window for **Study**, locate the **Study Settings** section.

8 Clear the **Generate default plots** check box.

9 On the **Home** toolbar, click **Compute**.

RESULTS

Displacement 1

1 In the **Model Builder** window, under **Results** right-click **Displacement** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement Modes` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 On the **Displacement Modes** toolbar, click **Plot**.

The exact value of the calculated eigenfrequency will converge for better mesh resolutions. That is, changing the boundary resolution from the selected `1.2*dvisc` to,

for example, `0.5*dvisc`. The latter finer resolution produces the value of `f_num` found in the **Definitions>Parameters** list.

The default (first) selected eigenvalue is the vibrating torque mode at about 10500 Hz that was studied in the frequency domain study.

The first mode should look like the one in Figure 6 top.

The next eigenmode represents a symmetric vibration in the transverse direction happening at about 13100 Hz.

5 From the **Eigenfrequency (Hz)** list, choose **13101+170.39i**.

6 On the **Displacement Modes** toolbar, click **Plot**.

The second mode should look like the one in Figure 6 bottom.

The next mode happens at nearly 40 kHz and can be seen by selecting the last of the three eigenfrequencies.

Appendix—Geometry Modeling Instructions.

If you wish to create the geometry yourself, follow these steps.

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)

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Rectangle 1 (r1)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.5`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Rectangle 2 (r2)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.05`.

5 Locate the **Position** section. In the **xw** text field, type `-0.05`.

6 In the **yw** text field, type `0.25`.

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **yw** text field, type `-0.3`.

Rectangle 4 (r4)

1 On the **Work Plane** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.5`.

4 In the **Height** text field, type `0.6`.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 Right-click **Rectangle 4 (r4)** and choose **Build All Objects**.

7 In the **Model Builder** window, click **Geometry 1**.

Sphere 1 (sph1)

1 On the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `0.8`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	0.3

Union 1 (uni1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

To better see the entire geometry use the **Zoom Extents** tool.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 Right-click **Union 1 (uni1)** and choose **Build Selected**.

Block 1 (blk1)

1 On the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type 2.

4 In the **Depth** text field, type 2.

5 In the **Height** text field, type 2.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **y** text field, type -1.

8 Right-click **Block 1 (blk1)** and choose **Build Selected**.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

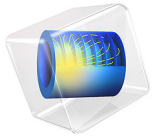
2 Select the object **uni1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **blk1** only.

6 Click **Build All Objects**.



Vibrating Particle in Water—Correct Thermoviscous Acoustic Material Parameters

This model is licensed under the COMSOL Software License Agreement 5.3a.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This is a model of a small vibrating particle in water, see Figure 1. It shows how to set up the thermoviscous acoustic material parameters in order to get the correct acoustic behavior of the system. The Thermoviscous Acoustics, Frequency Domain interface is used for modeling the propagation of acoustic waves in small devices, where it is important to include losses in a detailed way. This is useful when modeling acoustics and vibrations in, for example, microphones, hearing aids, or MEMS devices. The interface provides a detailed way of solving the equations governing the propagation of acoustic waves in any fluid, including thermal conduction and viscous damping.

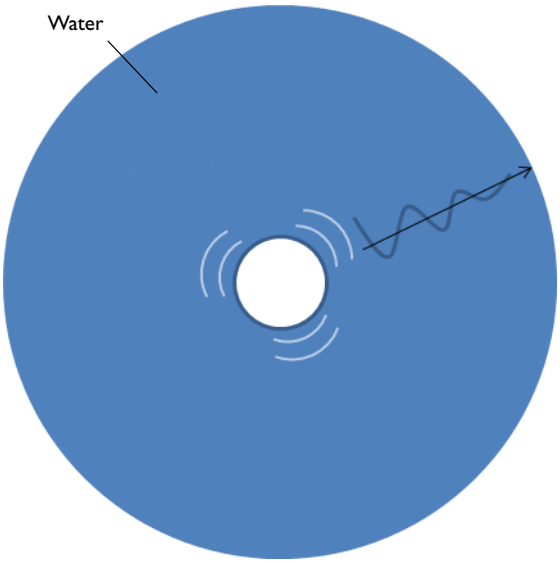


Figure 1: Sketch of the radially vibrating particle.

Acoustic waves are compressible waves and in the framework of the Thermoviscous Acoustics interface a linearized equation of state is set up. This constitutive equation relates the acoustic variations in pressure p, density ρ, and temperature T through the expression

$$\rho = \rho_0(\beta_T p - \alpha_p T)$$

where ρ_0 is the equilibrium density, β_T is the isothermal compressibility, and α_p is the coefficient of thermal expansion. If one of these coefficients is wrong or simply evaluates

to zero, the resulting solution is also wrong. Typically, no compressible acoustic waves appear or the speed of sound of the waves is wrong.

Having the option to easily set these parameters allows the user to model the propagation of waves in any desired fluid.

This model shows how to easily set up the coefficients for the case where the fluid is water. The model shows a small sphere of radius 1 mm which is vibrating at 50 kHz. The sphere is modeled in 2D axisymmetry.

Note: Details about the governing equations are found in the theory section of the thermoviscous acoustics physics interface documentation. See **File>Help>Documentation** and search or open the *Acoustics Module User's Guide* to the thermoviscous acoustics theory.

Model Definition

In the model the material water is used as the domain fluid. The equilibrium density ρ_0 is here only a function of the ambient equilibrium temperature T_0. This may be seen by either:

- Expanding the materials tree **Materials>Water, liquid>Basic>Piecewise 3 {rho}** and noting that only T is an argument here.
- Looking at the **Thermoviscous Acoustics Model 1** and noting that only the **Equilibrium temperature** input field is active, meaning that the material parameters are not depending on the pressure.

The fact that the density is independent of the pressure results in the isothermal compressibility to evaluate to zero with the default settings **From equilibrium density**. This is because it is calculated as

$$\beta_T = \frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial p}\right]_T$$

The same type of problem may occur with the coefficient of thermal expansion when the equilibrium density is not a function of the temperature.

The options are now to enter the expression for the compressibility as a user defined quantity, to pick it up from the material, or to compute it using the speed of sound (this option is chosen in this model) using the thermodynamic relation:

$$\beta_T = \frac{1}{\rho_0 c^2}\frac{\gamma}{} \tag{1}$$

Results and Discussion

The acoustic pressure variations in the entire model domain are plotted in Figure 2 and in Figure 3 the pressure variations are plotted along the r-axis. Figure 3 compares the correctly set up model, showing the expected pressure variations for spherical waves, with a model with default settings where the compressibility has evaluated to 0.

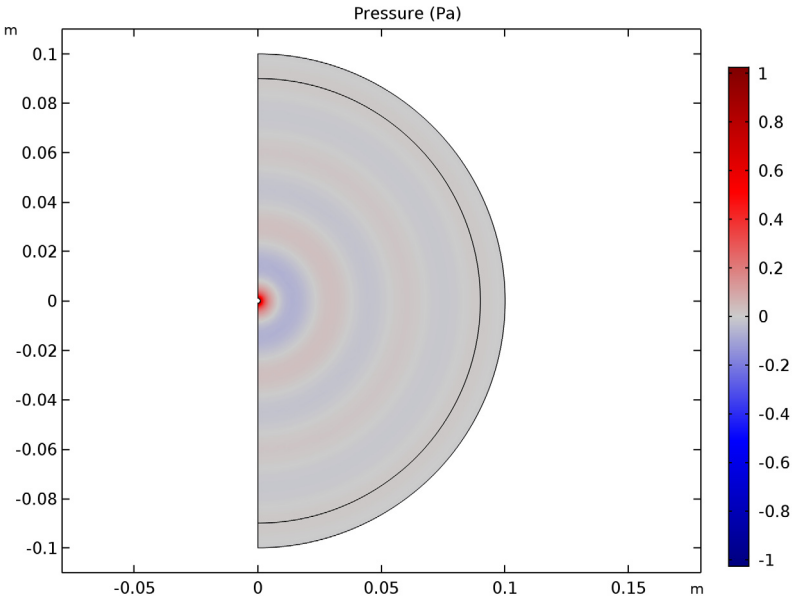


Figure 2: Pressure variations in the water outside the small vibrating particle using a correctly set up thermoviscous acoustic model.

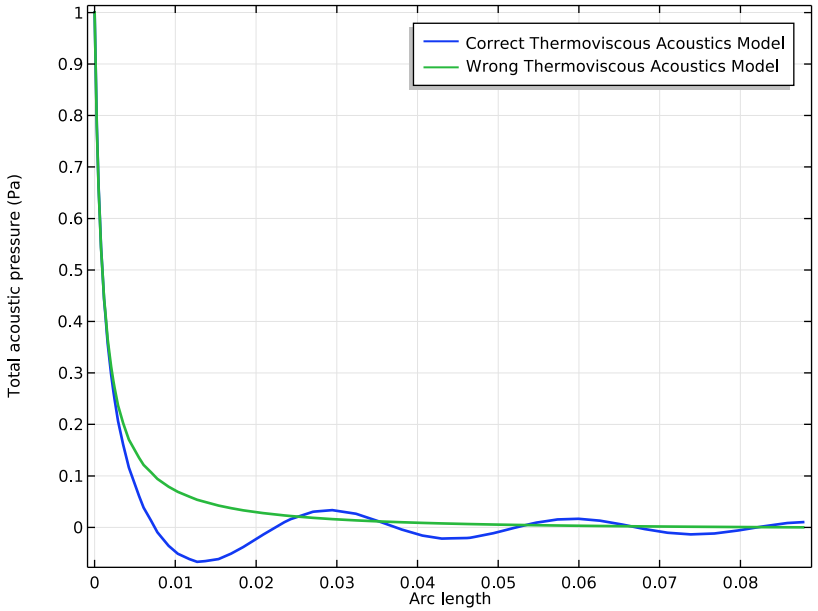


Figure 3: Pressure variation along the r-axis comparing the correctly set up material model with the default model showing no pressure fluctuations.

Application Library path: `Acoustics_Module/Tutorials/vibrating_particle_water`

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>Thermoviscous Acoustics>Acoustic-Thermoviscous Acoustic Interaction, Frequency Domain**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
a_s	1[mm]	0.001 m	Source radius
a_ta	90[mm]	0.09 m	Thermoaoucstic domain radius
a_tot	100[mm]	0.1 m	Total radius
f0	50[kHz]	50000 Hz	Study frequency
dvisc	55[um]* sqrt(100[Hz]/f0)	2.4597E-6 m	Viscous boundary layer thickness at f0
lambda0	1500[m/s]/f0	0.03 m	Wavelength at f0

GEOMETRY 1

Circle 1 (c1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type a_s.

4 In the **Sector angle** text field, type 180.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.

6 Right-click **Circle 1 (c1)** and choose **Build Selected**.

Circle 2 (c2)

1 Right-click **Circle 1 (c1)** and choose **Duplicate**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type a_ta.

Circle 3 (c3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Circle 1 (c1)** and choose **Duplicate**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `a_tot`.

4 Click **Build All Objects**.

5 Click the **Zoom Extents** button on the **Graphics** toolbar.

Difference 1 (dif1)

1 On the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **c2** and **c3** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.

5 Select the object **c1** only.

6 Click **Build All Objects**.

ADD MATERIAL

1 On 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>Water, liquid**.

4 Click **Add to Component** in the window toolbar.

5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Water, liquid (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk viscosity	muB	0	Pa·s	Basic

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

Thermoviscous Acoustics is only active in the inner domain, Pressure Acoustics will be added to the outer domain.

1 Click the **Zoom Extents** button on the **Graphics** toolbar.

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Thermoviscous Acoustics, Frequency Domain (ta)**.

3 Select Domain 2 only.

4 In the **Settings** window for **Thermoviscous Acoustics, Frequency Domain**, locate the **Sound Pressure Level Settings** section.

5 From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

Thermoviscous Acoustics Model 1

Set the compressibility to be evaluated according to Equation 1 using the speed of sound.

1 In the **Model Builder** window, under **Component 1 (comp1)>Thermoviscous Acoustics, Frequency Domain (ta)** click **Thermoviscous Acoustics Model 1**.

2 In the **Settings** window for **Thermoviscous Acoustics Model**, locate the **Thermal Expansion and Compressibility** section.

3 From the β_T list, choose **From speed of sound**.

It is necessary to add both a mechanical and a thermal condition on the particle surface. The default wall condition has both conditions built in. You can combine a mechanical and a thermal conditions to create a customized behavior at a boundary.

Normal Stress 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Normal Stress**.

2 Select Boundaries 7 and 8 only.

3 In the **Settings** window for **Normal Stress**, locate the **Normal Stress** section.

4 In the σ_n text field, type `1`.

Isothermal 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Isothermal**.

2 Select Boundaries 7 and 8 only.

Thermoviscous Acoustics Model 2

1 On the **Physics** toolbar, click **Domains** and choose **Thermoviscous Acoustics Model**.

2 In the **Settings** window for **Thermoviscous Acoustics Model**, type `Wrong Thermoviscous Acoustics Model` in the **Label** text field.

3 Select Domain 2 only.

Add a thin layer of pressure acoustics around the fluid domain including a radiation condition to mimic a non-reflecting boundary condition.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.

2 Select Domain 1 only.

3 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Sound Pressure Level Settings** section.

4 From the **Reference pressure for the sound pressure level** list, choose **Use reference pressure for water**.

Spherical Wave Radiation 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Spherical Wave Radiation**.

2 Select Boundaries 5 and 10 only.

Go to the **Multiphysics** node and click on the **Acoustics-Thermoviscous Acoustics Boundary**. You will note that the **Boundary Selection** is automatically set to **All boundaries** (this is the default when the coupling is added automatically in a multiphysics interface). Looking at the Graphics window it is evident that the coupling only is active at the interface between **Pressure Acoustics** and **Thermoviscous Acoustics**.

MESH 1

Now, proceed and set up the mesh.

Size

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.

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 `lambda0/8`.

Boundary Layers 1

1 Right-click **Mesh 1** and choose **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Click to expand the **Transition** section. Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Boundary Layers 1** click **Boundary Layer Properties**.

2 Select Boundaries 7 and 8 only.

3 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.

4 In the **Number of boundary layers** text field, type `5`.

5 From the **Thickness of first layer** list, choose **Manual**.

6 In the **Thickness** text field, type `0.5*dvisc`.

7 Click **Build All**.

STUDY 1

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `f0`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Thermoviscous Acoustics, Frequency Domain (ta)>Wrong Thermoviscous Acoustics Model**.

5 Click **Disable**.

6 On the **Home** toolbar, click **Compute**.

ROOT

Now, solve the model with the wrong thermoacosutics model (where the compressibility evaluated to 0), add a second study to do this.

ADD STUDY

1 On 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 **Preset Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

1. In the **Settings** window for **Study**, locate the **Study Settings** section.
2. Clear the **Generate default plots** check box.
3. In the **Model Builder** window, click **Step 1: Frequency Domain**.
4. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
5. In the **Frequencies** text field, type `f0`.
6. On the **Home** toolbar, click **Compute**.

Modify the first default plot to also show the pressure in the pressure acoustics domain.

RESULTS

Surface

1. In the **Model Builder** window, expand the **Acoustic Pressure (ta)** node, then click **Surface**.
2. In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3. From the **Color table** list, choose **Wave**.
4. Select the **Symmetrize color range** check box.

Surface 2

1. In the **Model Builder** window, under **Results** right-click **Acoustic Pressure (ta)** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `p2`.
4. Click to expand the **Inherit style** section. Locate the **Inherit Style** section. From the **Plot** list, choose **Surface**.
5. On the **Acoustic Pressure (ta)** toolbar, click **Plot**.

Acoustic Pressure (ta)

1. In the **Model Builder** window, under **Results** click **Acoustic Pressure (ta)**.
2. In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.
3. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `Pressure (Pa)`.

   The figure should look like the one in Figure 2.

There are several more default plots, they are labeled depending on if they stem from **Thermoviscous Acoustics (ta)** or **Pressure Acoustics (acpr)**. Proceed and create a cross section plot using a cut line data set.

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **R** to `a_s`.

4 In row **Point 2**, set **R** to `a_tot`.

Cut Line 2D 2

1 Right-click **Cut Line 2D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

1D Plot Group 8

1 On the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Cut Lines` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Line Graph 1

1 Right-click **Cut Lines** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type `2`.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Correct Thermoviscous Acoustics Model`

Line Graph 2

1 Right-click **Results>Cut Lines>Line Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 2D 2**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Wrong Thermoviscous Acoustics Model`

5 On the **Cut Lines** toolbar, click **Plot**.

The figure should look like the one in Figure 3.