COMSOL
MULTIPHYSICS

ACOUSTICS MODULE

MODEL LIBRARY

VERSION 3.5

COMSOL

How to contact COMSOL:

Benelux
COMSOL BV
Röntgenlaan 19
2719 DX Zoetermeer
The Netherlands
Phone: +31 (0) 79 363 4230
Fax: +31 (0) 79 361 4212
info@comsol.nl
www.comsol.nl

Denmark
COMSOL A/S
Diplomvej 376
2800 Kgs. Lyngby
Phone: +45 88 70 82 00
Fax: +45 88 70 80 90
info@comsol.dk
www.comsol.dk

Finland
COMSOL OY
Arabianranta 6
FIN-00560 Helsinki
Phone: +358 9 2510 400
Fax: +358 9 2510 4010
info@comsol.fi
www.comsol.fi

France
COMSOL France
WTC, 5 pl. Robert Schuman
F-38000 Grenoble
Phone: +33 (0)4 76 46 49 01
Fax: +33 (0)4 76 46 07 42
info@comsol.fr
www.comsol.fr

Germany
COMSOL Multiphysics GmbH
Berliner Str. 4
D-37073 Göttingen
Phone: +49-551-99721-0
Fax: +49-551-99721-29
info@comsol.de
www.comsol.de

Italy
COMSOL S.r.l.
Via Vittorio Emanuele II, 22
25122 Brescia
Phone: +39-030-3793800
Fax: +39-030-3793899
info.it@comsol.com
www.it.comsol.com

Norway
COMSOL AS
Søndre gate 7
NO-7485 Trondheim
Phone: +47 73 84 24 00
Fax: +47 73 84 24 01
info@comsol.no
www.comsol.no

Sweden
COMSOL AB
Tegnérgatan 23
SE-111 40 Stockholm
Phone: +46 8 412 95 00
Fax: +46 8 412 95 10
info@comsol.se
www.comsol.se

Switzerland
FEMLAB GmbH
Technoparkstrasse 1
CH-8005 Zürich
Phone: +41 (0)44 445 2140
Fax: +41 (0)44 445 2141
info@femlab.ch
www.femlab.ch

United Kingdom
COMSOL Ltd.
UH Innovation Centre
College Lane
Hatfield
Hertfordshire AL10 9AB
Phone:+44-(0)-1707 636020
Fax: +44-(0)-1707 284746
info.uk@comsol.com
www.uk.comsol.com

United States
COMSOL, Inc.
1 New England Executive Park
Suite 350
Burlington, MA 01803
Phone: +1-781-273-3322
Fax: +1-781-273-6603

COMSOL, Inc.
10850 Wilshire Boulevard
Suite 800
Los Angeles, CA 90024
Phone: +1-310-441-4800
Fax: +1-310-441-0868

COMSOL, Inc.
744 Cowper Street
Palo Alto, CA 94301
Phone: +1-650-324-9935
Fax: +1-650-324-9936

info@comsol.com
www.comsol.com

For a complete list of international representatives, visit
www.comsol.com/contact

Company home page
www.comsol.com

COMSOL user forums
www.comsol.com/support/forums

Acoustics Module Model Library

Version: September 2008 COMSOL 3.5

Part number: CM020202

CONTENTS

Chapter 1: Introduction

Chapter 2: Tutorial Models

Chapter 3: Industrial Models

Chapter 4: Benchmark Models

1

Introduction

The *Acoustics Module Model Library* consists of a set of models from various areas of acoustics engineering simulation. Its purpose is to assist you in learning, by example, how to model sophisticated acoustics systems and effects. Through the library models, you can tap the expertise of the top researchers in the field, examining how they approach some of the most difficult modeling problems you might encounter. You can thus get a feel for the power that COMSOL Multiphysics offers as a modeling tool. In addition to serving as references, the models can give you a head start if you are developing a model of a similar nature.

We have divided the models into three groups:

- *Benchmark models*—this category consists of models for which you can compare the COMSOL Multiphysics solution with either an analytical solution or some reference numerical solution
- *Industrial models*—these are models in applied fields of acoustics with direct industrial relevance
- *Tutorial models*—the models in this group are deemed particularly suitable for learning how to model with the Acoustics Module

The models also illustrate the use of the various acoustics-specific application modes from which we built them. These specialized modes are not available in the base COMSOL Multiphysics package, and they come with their own graphical user interfaces that make it quick and easy to access their power. You can even modify them for custom requirements. COMSOL Multiphysics itself is very powerful, and with sufficient expertise in a given field you could certainly develop these modes by yourself. But why spend the hundreds or thousands of hours that would be necessary when our team of experts has already done the work for you?

Note that the model descriptions in this book do not contain every detail on how to carry out every step in the modeling process. Before tackling these in-depth models, we urge you to first read the second book in the Acoustics Module documentation set. Titled the *Acoustics Module User's Guide*, it introduces you to the basic functionality in the module, covers basic modeling techniques, and includes reference material of interest to those working in acoustics. For more information on how to work with the COMSOL Multiphysics graphical user interface, please refer to the COMSOL Multiphysics User's Guide or the COMSOL Multiphysics Quick Start manual. An explanation of how to model with a programming language is available in yet another book, the *COMSOL Multiphysics Scripting Guide*.

The book in your hands, the *Acoustics Module Model Library*, provides details about a large number of ready-to-run models that illustrate real-world uses of the module. Each entry comes with theoretical background as well as instructions illustrating how to set it up. They were written by our staff engineers who have years of experience in acoustics; they are your peers, using the language and terminology needed to get across the sophisticated concepts in these advanced topics.

Finally, note that we supply these models as COMSOL Multiphysics Model MPH-files so that you can import them into COMSOL Multiphysics for immediate execution.

Model Library Guide

The table below summarizes key information about the entries in the *Acoustics Module Model Library.* A series of columns states the application mode (such as Pressure Acoustics) used to solve the corresponding model. The solution time is the elapsed time measured on a machine running Windows Vista with a 2.6 GHz AMD Athlon X2 Dual Core 500 CPU and 2 GB of RAM. For models with a sequential solution strategy, the Solution Time column shows the total combined solution time for all solution steps. Additional columns point out the modeling features that a given example highlights. The categories here include the type of analysis (such as time harmonic or transient) and whether multiphysics or parametric studies are included.

TABLE 1-1: ACOUSTICS MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODES	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TRANSIENT	EIGENFREQUENCY/EIGENMODE	SENSITIVITY/OPTIMIZATION	MULTIPHYSICS	PARAMETRIC STUDY
TUTORIAL MODELS										
Bessel Panel	8	Pressure Acoustics	2 min		√					
Doppler Shift	136[*]	Aeroacoustics	2 s		√					
Hollow Cylinder	19	Solid, Stress-Strain; Pressure Acoustics	41 s		√				√	
Optimizing the Shape of a Horn[¤]	34	Pressure Acoustics, Moving Mesh (ALE) Optimization	42 s		√			√		√
Jet Pipe	48	Aeroacoustics (acae, acab)	12 s		√					
Piezoacoustic Transducer	59	Piezo; Pressure Acoustics	1 s		√		√		√	
Cylindrical Subwoofer	14[*]	Pressure Acoustics	21 s		√					
Transient Gaussian Explosion	69	Pressure Acoustics	3 min			√				
Ultrasound Scattering Off a Cylinder	78	Pressure Acoustics, UWVF	9 s		√					
INDUSTRIAL MODELS										
Absorptive Muffler	88	Pressure Acoustics (acpr, acbm)	8 min		√		√			√

TABLE 1-1: ACOUSTICS MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODES	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TRANSIENT	EIGENFREQUENCY/EIGENMODE	SENSITIVITY/OPTIMIZATION	MULTIPHYSICS	PARAMETRIC STUDY
Car Interior	104	Pressure Acoustics	88 min		√					√
Flow Duct	115	Compressible Potential Flow, Aeroacoustics (acae, acab)	32 s	√	√		√		√	
Loudspeaker Driver[†]	145	Pressure Acoustics; AC Power Electromagnetics; Axial Symmetry, Stress-Strain	15 min		√				√	√
Loudspeaker Suspension[†]	145	(Pressure Acoustics; AC Power Electromagnetics) Axial Symmetry, Stress-Strain	57 s	√	√					
Loudspeaker Driver in a Vented Enclosure	175	Pressure Acoustics	35 min							
Muffler with Perforates	193	Pressure Acoustics	27 min		√					√
SAW Gas Sensor	210	Piezo Plane Strain	42 s				√		√	
BENCHMARK MODELS										
Vibrations of a Disk Backed by an Air-Filled Cylinder[‡]	226	Pressure Acoustics, Mindlin Plate	12 s				√		√	

TABLE 1-1: ACOUSTICS MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODES	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TRANSIENT	EIGENFREQUENCY/EIGENMODE	SENSITIVITY/OPTIMIZATION	MULTIPHYSICS	PARAMETRIC STUDY
Open Pipe	238	Pressure Acoustics	22 s		√					√
Scattering from a Plate with Ribs	252	Pressure Acoustics	5 min		√					√

* This page number refers to the *Acoustics Module User's Guide*.

¤ This model requires the Optimization Lab.

† This model requires the AC/DC Module.

‡ This model requires the COMSOL Multiphysics Structural Mechanics Module.

Typographical Conventions

All COMSOL manuals use a set of consistent typographical conventions that should make it easy for you to follow the discussion, realize what you can expect to see on the screen, and know which data you must enter into various data-entry fields. In particular, you should be aware of these conventions:

- A **boldface** font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL graphical user interface (for toolbar buttons in the corresponding tooltip). For instance, we often refer to the **Model Navigator**, which is the window that appears when you start a new modeling session in COMSOL; the corresponding window on the screen has the title **Model Navigator**. As another example, the instructions might say to click the **Multiphysics** button, and the boldface font indicates that you can expect to see a button with that exact label on the COMSOL user interface.
- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user interface during geometry modeling. However, nowhere on the screen will you see

the term "Draw" referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).

- The symbol **>** indicates a menu item or an item in a folder in the **Model Navigator**. For example, **Physics>Equation System>Subdomain Settings** is equivalent to: On the **Physics** menu, point to **Equation System** and then click **Subdomain Settings**. **COMSOL Multiphysics>Heat Transfer>Conduction** means: Open the **COMSOL Multiphysics** folder, open the **Heat Transfer** folder, and select **Conduction**.
- A `Code` (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as "Type `1.25` in the **Current density** edit field." The monospace font also indicates COMSOL Script codes.
- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

2

Tutorial Models

In this chapter you can find a selection of tutorial models that show how to use the features in the Acoustics Module to solve common acoustics problems.

Bessel Panel

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 of Bessel panels. The solution is compared with analytical results.

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 2-1 is a drawing of this assembly and the input to each speaker.

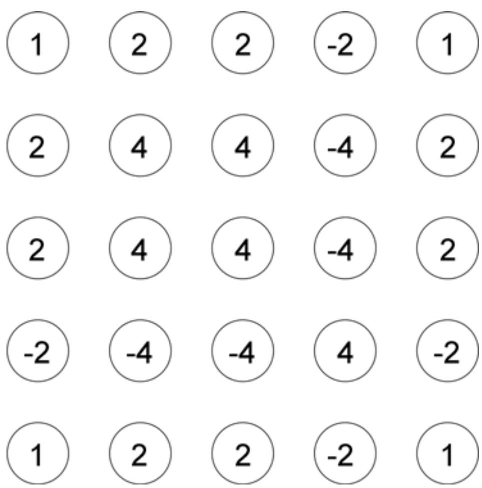


Figure 2-1: The Bessel panel combination used in the model. The circles represent the speakers and the numbers represent their input. Each row and each column is a Bessel panel in itself.

For the harmonic sound waves of acoustic pressure $p(\mathbf{x}, t) = p(\mathbf{x})e^{i\omega t}$ that you study in this model, the following frequency-domain Helmholtz equation applies for $p(\mathbf{x})$:

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

Here ρ_0 is the density of the medium (kg/m^3), $\omega = 2\pi f$ denotes the angular frequency (rad/s), c_s refers to the speed of sound (m/s), and Q_L (1/s^2) is a monopole source representing a loudspeaker.

For air, $\rho_0 = 1.25$ kg/m^3 and $c_s = 343$ m/s. For the frequency use $f = 100$ Hz. Each loudspeaker, L, is represented by a point source emitting a flow of strength $S_L = 10^{-2} n_L$ m^3/s, where n_L is the weight factor shown in Figure 2-1. It holds that

$$Q_L = \omega S_L \, \delta^{(3)}(\mathbf{R} - \mathbf{R}_L)$$

where $\delta^{(3)}$ refers to the 3D Dirac delta function and $\mathbf{R}_L$ is the location of the speaker L.

See Figure 2-2 for the model geometry. The distance between two neighboring loudspeakers is 0.5 m. A sphere of radius of 5 m represents an infinite air domain surrounding the loudspeakers.

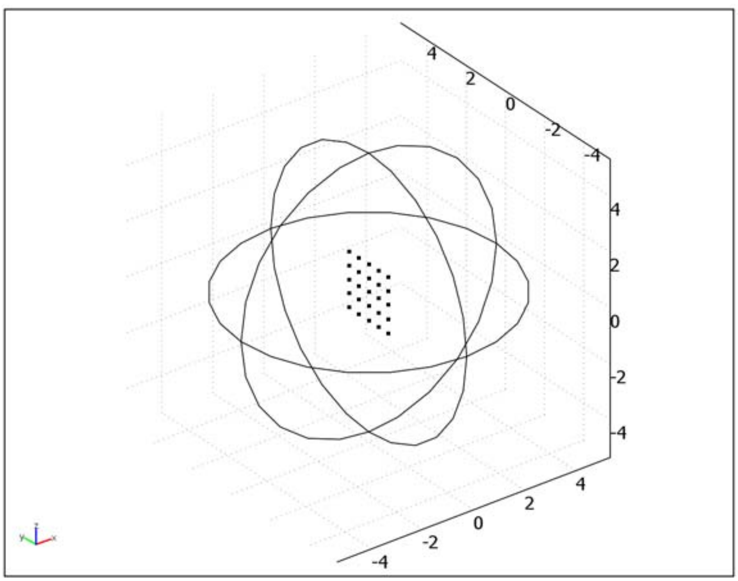


Figure 2-2: The model geometry.

The predefined *radiation condition* within the *spherical wave* option minimizes reflections on the exterior boundaries of the air sphere. This boundary condition allows a spherical wave to travel out of the system while generating only minimal reflections for the wave's non-spherical components. The radiation boundary condition is useful when the surroundings are simply a continuation of the domain.

For mathematical details on the radiation boundary condition, see the description under the heading "Radiation Boundary Condition" on page 79 of the *Acoustics Module User's Guide*.

Results and Discussion

Figure 2-3 shows the sound pressure distribution in a slice of the sphere close to the loudspeakers. In this immediate vicinity of the sources, the sound field is still very inhomogeneous.

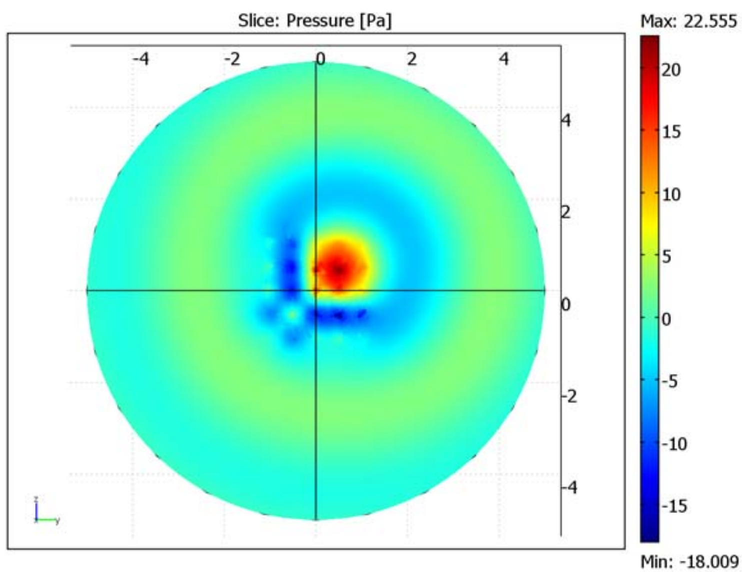


Figure 2-3: Slice plot of the sound pressure distribution at 500 Hz. The slice is parallel with the yz-plane and situated at x = 0.2 m.

Another way of visualizing the near sound field is as an isosurface plot. Figure 2-4 shows the isobar for the absolute value of the sound pressure of 4 Pa.

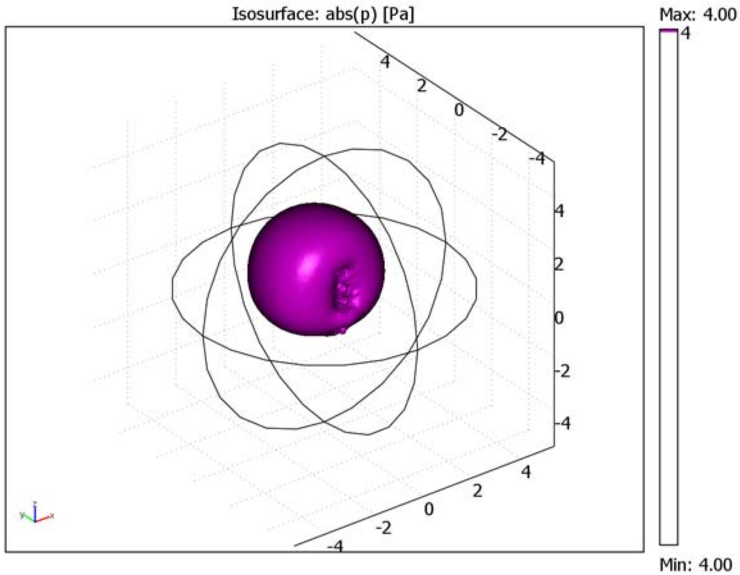


Figure 2-4: Isosurface plot showing the location of the isobar abs(p) = 4 Pa.

Figure 2-5 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 2.3 dB.

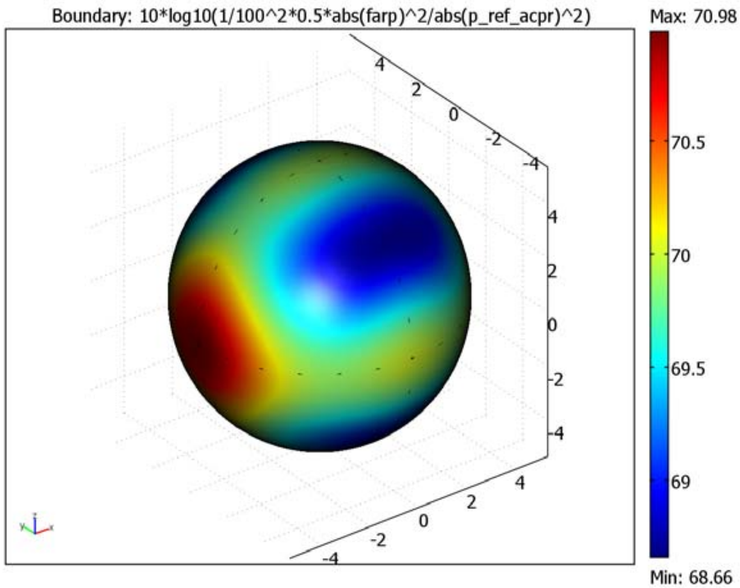


Figure 2-5: Sound pressure level (dB) at a distance of 100 m from the loudspeakers.

Figure 2-6 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 close to the analytical solution. Besides refining the mesh, you can refine the accuracy by adding perfectly matched layers outside the computational domain; for more information see page 73 of the *Acoustics Module*

User's Guide. Finally, the accuracy is bounded by the far-field transformation itself: the longer the distance from the sources, the better the accuracy.

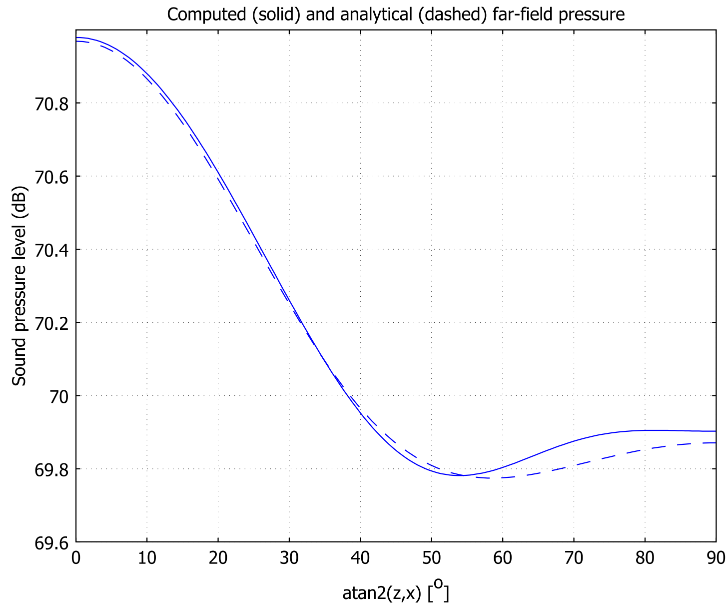


Figure 2-6: Sound pressure level (dB) at a radial distance of 100 m in the xz-plane (zero azimuthal angle) as a function of the polar angle from the xy-plane. The solid line represents the computed solution and the dashed line the analytical solution.

Modeling in COMSOL Multiphysics

Use the Pressure Acoustics application mode of the Acoustics Module to set up the model of the Bessel panel. The GMRES solver with the Geometric multigrid preconditioner ensures low memory consumption at a high mesh resolution. In an optional exercise that requires COMSOL Script or MATLAB, you run a script to calculate the analytical solution for comparison.

Reference

1. "Bessel panels—high-power speaker systems with radial sound distribution," *Technical publication 091*, Philips Export BV, 1983.

Model Library path: `Acoustics_Module/Tutorial_Models/bessel_panel`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator** select **3D** from the **Space dimension** list.

2 From the list of application modes select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis**.

3 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1 Choose **Draw>Sphere**. In the **Radius** edit field type `5`, then click **OK**.

2 Choose **Draw>Point**. Add a point to the geometry by typing the following values in the **Coordinates** edit fields, then click **OK**.

Coordinates x	0
Coordinates y	-1
Coordinates z	-1

3 With the point selected, choose **Draw>Modify>Array**. Enter the following data, then click **OK**.

Displacement x	0
Displacement y	0.5
Displacement z	0.5
Array size x	1
Array size y	5
Array size z	5

OPTIONS AND SETTINGS

Choose **Options>Constants** and define the following constant.

NAME	EXPRESSION	DESCRIPTION
S	0.01[m^3/s]	Flow source

PHYSICS SETTINGS

Subdomain Settings

The Bessel loudspeaker array is modeled in air. Because this is the default medium no changes are needed.

Boundary Conditions

1 Choose **Physics>Boundary Settings**.

2 Select all the boundaries, then select **Radiation condition** from the **Boundary condition** list.

3 From the **Wave type** list select **Spherical wave**.

In preparation for studying the far field, you must supply a postprocessing variable. Plotting on boundaries gives you the far field for all angles. When plotting on edges you are limited to fixed azimuthal or polar angles, but as there is a lower number of computations involved these plots display much faster.

4 Go to the **Far-Field** page.

5 Select Boundaries 5–8.

6 In the top **Name** edit field type `farp`. When you select the next edit field in the table the default values for the field and the normal derivative appears.

7 Select the **Symmetry planes: x=0** check box. Keep the default **Symmetric pressure**.

8 Click **OK** to close the dialog box.

Point Settings

1 Choose **Physics>Point Settings**.

2 For all points, select **Flow** from the **Type of source** list. Assign the values of the flow according to the table below.

SETTINGS	POINTS 3, 7, 25, 29	POINTS 4, 8, 12, 26	POINTS 5, 6, 14, 18, 20, 24, 27, 28	POINTS 9, 16, 17, 22, 23	POINTS 10, 11, 15, 21
iS	S	-2*S	2*S	4*S	-4*S

The minus signs correspond to sources with the opposite phase.

3 Click **OK** to close the dialog box.

GENERATING THE MESH

1 From the **Mesh** menu open the **Free Mesh Parameters** dialog box. On the **Global** page, select **Custom mesh size** and type `0.85` in the **Maximum element size** edit field.

2 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

1. Choose **Solve>Solver Parameters** or click the corresponding button on the Main toolbar to open the **Solver Parameters** dialog box.
2. From the **Linear system solver** list select **GMRES**, and from the **Preconditioner** list select **Geometric multigrid**.
3. Click the **Settings** button. In the dialog box that appears click **Preconditioner**, then select **Refine mesh** from the **Hierarchy generation method** list.
4. Click **OK** twice to close both the **Linear System Solver Settings** dialog box and the **Solver Parameters** dialog box. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the acoustic pressure on five equidistant slices along the x-axis. The central slice comprises the point sources, resulting in singular pressure values. For a better view of the local pressure distribution close to the sources, try a single slice offset by a small distance from the sources.

1. Choose **Postprocessing>Plot Parameters** or click the corresponding button on the Main toolbar.
2. On the **Slice** page find the **Slice positioning** area. Click the **Vector with coordinates** option button for the **x levels** entry and type `0.2` in the corresponding edit field.
3. Click **OK** to generate the plot.
4. Click the **Go to YZ View** button. The plot should now resemble the one in Figure 2-3.

The near-field radiation pattern is well represented by an isosurface plot of the absolute value of the sound pressure.

1. Click the **Go to Default 3D View** button.
2. Return to the **Plot Parameters** dialog box and click the **General** tab. Clear the **Slice** check box and select the **Isosurface** check box.
3. On the **Isosurface** page, type `abs(p)` in the **Expression** edit field on the **Isosurface Data** page.
4. Select the **Vector with isolevels** option button and type `4` in the corresponding edit field.
5. Select the **Uniform color** option button. Click the **Color** button, select a violet hue and click **OK**.

6 Click **OK** to generate the plot, then click the **Headlight** button on the Camera toolbar to get a clearer view.

The far-field pressure `farp` is normalized to give the pressure at a distance of 1 m from the source. The sound intensity, I, is proportional to p^2, and the total power radiated, $P = 4\pi I r^2$, is independent of the distance, r, from the source. It then follows that p scales as r^{-1}. Thus, to get the sound pressure level in dB at a distance of 100 m from the panel, proceed as follows.

1 Choose **Postprocessing>Domain Plot Parameters** and click the **Line/Extrusion** tab.
2 Select Edge 11 and type `10*log10(1/100^2*0.5*abs(farp)^2/abs(p_ref_acpr)^2)` in the **Expression** edit field in the **y-axis data** area.
3 Click the **Expression** option button and then click the **Expression** button in the **x-axis data** area.
4 In the dialog box that appears, type `atan2(z,x)` in the **Expression** edit field and select the degree sign (°) from the **Unit** list.
5 Click **OK** twice to see the 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 solid line in Figure 2-6.

You can visualize the far field in all possible directions using the boundaries of the sphere. Please note that it takes approximately 30 minutes before this plot shows. If you do not want to see the far field on the boundaries, skip the next two steps.

1 In the **Plot Parameters** dialog box click the **Boundary** tab. In the **Expression** edit field type `10*log10(1/100^2*0.5*abs(farp)^2/abs(p_ref_acpr)^2)`.
2 Select the **Boundary plot** check box, then click **OK** to see the plot.

When the far-field boundary plot eventually shows up, the values should vary between 68.7 dB and 71.0 dB.

The plot that appears when you open this model in the Model Library is a combination of the isosurface plot that you already have and a surface plot of the local sound pressure level:

1 Choose **Options>Suppress>Suppress Boundaries**. In the dialog box that appears, select Boundary 2 and click **OK**.
2 In the **Plot Parameters** dialog box click the **Boundary** tab. Select **Sound pressure level** from the **Predefined quantities** list and click **OK**.

POSTPROCESSING WITH COMSOL SCRIPT

If you have access to COMSOL Script or MATLAB, you can easily compare the results from your simulation with the analytical solution. The `bessel_pressure` script returns the pressure at the coordinates (`xin`, `yin`, `zin`). The script is stored in the `multiphysics` folder as `bessel_pressure.m`.

To compare the simulation results with the analytical solution as in Figure 2-6, follow these steps:

1. Choose **Options>Functions**.
2. Click the **New** button. In the **Function name** edit field type `bessel_pressure`, then click **OK** to return to the **Functions** dialog box.
3. In the **Arguments** edit field type `x, y, z`, and in the **Expression** edit field type `bessel_pressure(x,y,z)`.
4. Select the **May produce complex output for real arguments** check box.
5. Click **OK** to close the dialog box.
6. Choose **Solve>Update Model**.
7. Choose **Postprocessing>Domain Plot Parameters**.
8. If you have closed the far-field plot, recreate it by selecting Edge 11 on the **Line/Extrusion** tab and then clicking **Apply**.
9. On the **General** page select the **Keep current plot** check box and click the **Title/Axis** button.
10. In the **Title/Axis Settings** dialog box go to the **Title** edit field and type `Computed (solid) and analytic (dashed) far-field pressure`, then in the **Second axis label** edit field type `Sound pressure level (dB)`. Click **OK** to close the dialog box.

    The radius of the model geometry is 5 m. To get the sound level at a distance of 100 m from the source, you therefore must multiply the input coordinates with a factor of 20.
11. Go to the **Line/Extrusion** page of the **Domain Plot Parameters** dialog box.
12. In the **Expression** edit field type `10*log10(0.5*abs(bessel_pressure(20*x,20*y,20*z))^2/abs(p_ref_acpr)^2)`.
13. Click the **Line Settings** button. From the **Line style** list select **Dashed line**, then click **OK**.
14. Click **OK** to see the plot, which should resemble the one in Figure 2-6.

Hollow Cylinder

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
- 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.

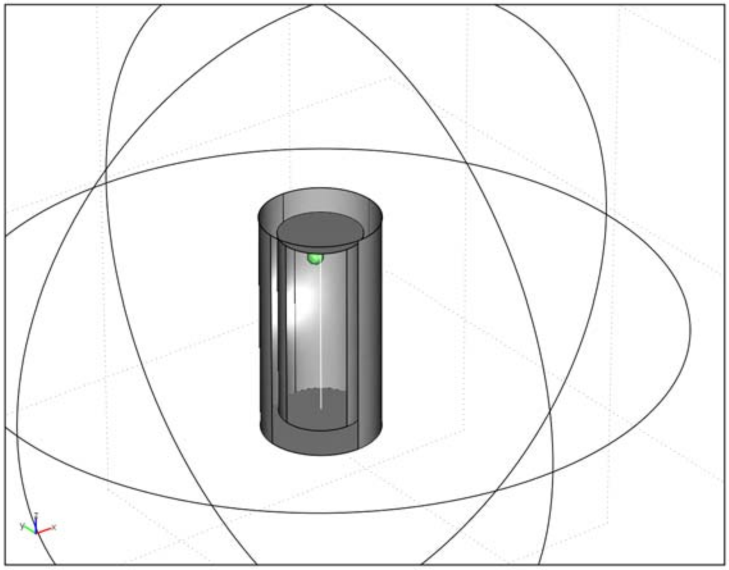


Figure 2-7: A hollow aluminum cylinder is immersed in water. The white line inside the cylinder indicates the line source, and the tiny sphere next to the line shows the position of the point source. The simulation domain is bounded by a large sphere.

Figure 2-7 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 Subdomain

For harmonic sound waves we use the frequency-domain Helmholtz equation for sound pressure:

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

Here, the acoustic pressure is a harmonic quantity, $p = p_0 e^{i\omega t}$ (N/m^2), ρ_0 is the density of the medium (kg/m^3), $\mathbf{q}$ is an optional *dipole source* (N/m^3), ω is the angular frequency (rad/s), and c_s is the speed of sound (m/s). In the present model, no dipole source is included.

TABLE 2-1: ACOUSTICS DOMAIN DATA

QUANTITY	VALUE	DESCRIPTION
ρ_0	997 kg/m^3	Density
c_s	1500 m/s	Speed of sound
p_{wref}	10^{-6} Pa	Reference sound pressure
$f=\omega/2\pi$	60,000 Hz	Frequency

In the above table, p_{wref} denotes the standard reference pressure used when defining the sound pressure level in water; its value differs from that of air, which is the default setting in COMSOL Multiphysics.

Solid Subdomain

You calculate the harmonic stresses and strains inside the solid cylinder walls using a frequency response analysis in the 3D Solid, Stress-Strain application mode. 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 2-7), 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 minimal 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 subsection “Radiation Boundary Conditions” on page 79 of the *Acoustics Module User’s Guide*.

Cylinder-Water Interface

To couple the acoustic pressure wave to the solid cylinder, set the boundary load $\mathbf{F}$ (force/unit area) on the 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.

To couple the frequency response of the solid back to the acoustics problem, use the boundary condition that the normal acceleration

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

equal that of the solid structure. Here, $\mathbf{n}_a$ is the outward-pointing unit normal vector seen from inside the acoustics domain.

These boundary conditions are available as boundary groups using the Acoustic-Structure Interaction predefined multiphysics coupling.

EDGE AND POINT SETTINGS

In the two cases considered, the sound waves are generated by either a point source or a line source. A line source along the z-axis is defined as follows:

$$\nabla \cdot \left(-\frac{1}{\rho_0}\nabla p\right) = 2\sqrt{\frac{P\omega}{\rho_0}}\delta^{(2)}(\mathbf{r})$$

Here P is the power per unit length of an infinitely long line source 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.

For a point source of power P located at the point $\mathbf{R} = \mathbf{R}_0$ in an infinite homogeneous space, the definition is

$$\nabla \cdot \left(-\frac{1}{\rho_0}\nabla p\right) = 2\sqrt{\frac{\pi P c_s}{\rho_0}}\delta^{(3)}(\mathbf{R} - \mathbf{R}_0)$$

where $\delta^{(3)}(\mathbf{R})$ is the Dirac delta function in three dimensions. Any type of confinement will result in higher power usage.

Results

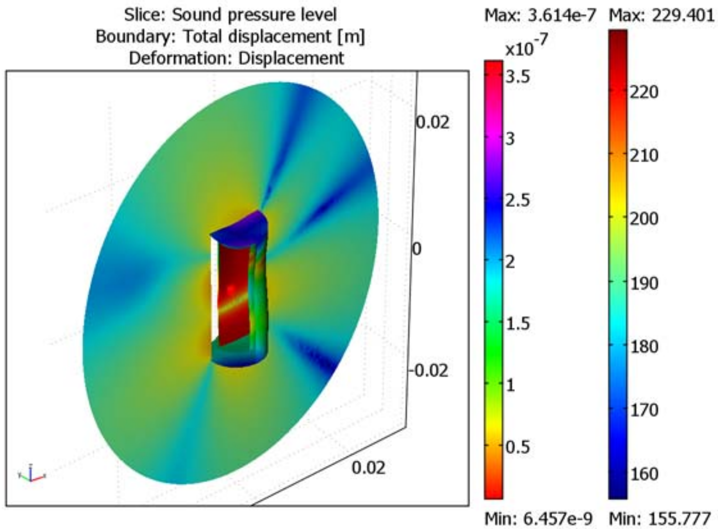


Figure 2-8: Sound-pressure plot (dB) of the acoustic waves in the coupled problem, using a point source inside the cylinder. The surfaces of the cylinder show its deformation (m). Some of the surfaces are hidden to reveal the pressure distribution inside the cylinder.

Figure 2-8 shows the sound pressure in the near field as a slice plot, for the case of an off-center point source. Far-field results are shown in the "Postprocessing and Visualization" section of the step-by-step instructions.

Modeling in COMSOL Multiphysics

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. Thanks to 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.

Model Library path: `Acoustics_Module/Tutorial_Models/hollow_cylinder`

MODEL NAVIGATOR

1 Start COMSOL Multiphysics.

2 In the **Model Navigator**, select **3D** from the **Space dimension** list.

3 Select **Acoustics Module>Acoustic-Structure Interaction** from the list of application modes.

4 Click **OK**.

OPTIONS

1 Open the **Constants** dialog box from the **Options** menu and enter the following values (the descriptions are optional):

NAME	EXPRESSION	DESCRIPTION
Freq	60[kHz]	Frequency
rhow	997[kg/m^3]	Water density
cw	1500[m/s]	Speed of sound in water
pw_ref	1[uPa]	Reference sound pressure in water
R	3[cm]	Radius of modeling domain
edgeL	1.7[cm]	Length of line source

2 Click **OK**.

3 Choose **Physics>Scalar Variables** and enter the following values:

NAME	EXPRESSION	DESCRIPTION
freq_acsld	Freq	Excitation frequency
freq_acpr	Freq	Excitation frequency
p_ref_acpr	pw_ref	Pressure reference

4 Click **OK**.

GEOMETRY MODELING

All the buttons used in creating the geometry of this model are located in the leftmost of the vertical toolbars.

1 Click the **Cylinder** button. In the dialog box that appears, enter property values according to the following table:

PROPERTY	VALUE
Radius	0.005
Height	0.02
Axis base point, z	-0.01

Let all other entries retain their default values. Click **OK** to close the dialog box.

2 Click the **Cylinder** button once more and create a cylinder with the following specifications; when done, click **OK**.

PROPERTY	VALUE
Radius	0.0035
Height	0.017
Axis base point, z	-0.0085

3 Click the **Line** button. Create a line with the following endpoints; when done, click **OK**.

PROPERTY	VALUE
x	0 0
y	0 0
z	-0.0085 0.0085

4 Click the **Point** button. Create a point located at the following coordinate; when done, click **OK**.

PROPERTY	VALUE
x	0.001
y	0.002
z	0.005

5 Click the **Sphere** button. In the dialog box that appears, type 0.03 in the **Radius** edit field, and let the other entries retain their default values. Click **OK** to close the dialog box.

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

PHYSICS SETTINGS

Subdomain Settings—Solid

1 From the **Multiphysics** menu, select **Solid, Stress-Strain (acsld)**.
2 From the **Physics** menu, select **Subdomain Settings**. Select Subdomains 1 and 3, then select **Fluid domain** from the **Group** list.
3 Select Subdomain 2, then select **Solid domain** from the **Group** list.
4 On the **Material** page, click **Load**. Select **Aluminum 3003-H18** under the **Basic Material Properties** entry in the **Materials** list. Click **OK**.
5 Click the **Damping** tab, then select **No damping** from the **Damping model** list.
6 Click **OK**.

Boundary Conditions—Solid

1 In the **Boundary Settings** dialog box select all the exterior boundaries (5–12, 15–16, and 19–20).
2 From the **Group** list, select **Fluid load**.

   The variables `nx_acpr`, `ny_acpr`, and `nz_acpr` in the predefined face load components are the Cartesian components of the normal vector directed outward from the subdomains where the **Pressure Acoustics (acpr)** application mode is active.
3 Click **OK**.

Subdomain Settings—Pressure Acoustics

1 From the **Multiphysics** menu, select **Pressure Acoustics (acpr)**.
2 Choose **Physics>Subdomain Settings**.
3 Select Subdomain 2 and then select **Solid domain** from the **Group** list.
4 Select Subdomains 1 and 3 and then select **Fluid domain** from the **Group** list. Enter the following data for the density and the speed of sound:

QUANTITY	VALUE/EXPRESSION
ρ_0	`rhow`
c_s	`cw`

5 Click **OK**.

Boundary Conditions—Pressure Acoustics

1 Select **Physics>Boundary Settings**. Hold down the Ctrl key and select Boundaries 1–4, 13–14, and 17–18. Select **Radiation condition** from the **Boundary condition** list, and choose **Spherical wave** as the wave type.

2 Select the **Select by group** check box, then select Boundary 5 to get a group selection of all the remaining boundaries.

3 From the **Group** list, select **Structural acceleration**.

4 Click **OK**.

The variables `u_tt_acsld`, `v_tt_acsld`, and `w_tt_acsld` in the predefined setting for the normal acceleration are the acceleration components from the **Solid, Stress-Strain (acsld)** application mode.

Edge Settings—Pressure Acoustics

1 Select **Physics>Edge Settings**.

2 In the dialog box that appears, select Edge 26.

3 Select **Power** in the **Type of source** list, then set **P** to `1/edgeL`.

4 Click **OK**.

Note that the presence of a Dirac delta function is not explicitly indicated on the right-hand side of the equation in the **Equation** area; instead, the localization of the source to the selected edge is implicitly understood.

GENERATING THE MESH

1 From the **Mesh** menu open the **Free Mesh Parameters** dialog box. On the **Global** page select **Custom mesh size** and type `0.005` in the **Maximum element size** edit field.

   This value corresponds to $0.2L$, where $L = c_s/f$ is the wavelength of the sound waves in the acoustics domain. Combined with the (default) choice of second-order elements, it follows that the rule-of-thumb minimum of ten to twelve degrees of freedom per wavelength for the solution to be reliable is satisfied.

2 Click the **Subdomain** tab. Select Subdomain 2 and set the **Maximum element size** to `0.002`.

3 Click **Remesh**, then click **OK** to close the dialog box.

COMPUTING THE SOLUTION

1 Select **Solve>Solver Parameters**.

2 From the **Solver** list, select **Stationary**.

3 From the **Linear system solver** list, select **GMRES**.

   The GMRES solver by default uses the geometric multigrid preconditioner to solve the system in two iterations: first with 1st-order elements and then with 2nd-order elements. In solving the present model, GMRES is faster and requires less memory (about 400 MB versus 900 MB) than the SPOOLES linear solver.

4 Click **OK**.

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

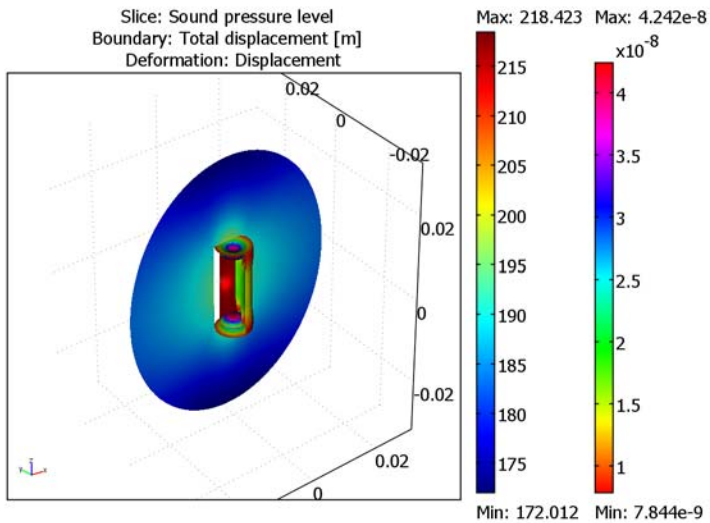


Figure 2-9: The sound pressure level (dB) inside and outside the cylinder, and the deformations (m) of the cylinder when using a line pressure source along the axis of the cylinder.

The default plot is a slice plot of the von Mises stress. There is much more information hidden in the solution, and there is a lot you can do to improve the result's readability and enhance its looks. For example, to arrive at the plot in Figure 2-9, do as follows:

1 Choose **Options>Suppress>Suppress Boundaries**.

2 Select Boundaries 5–6 and 9–10 from the list. Click **OK**.

3 Choose **Postprocessing>Plot Parameters**. On the **General** page, select the check boxes for **Slice**, **Boundary**, and **Deformed shape** in the **Plot type** area. Leave all the other check boxes cleared.

4 Click the **Slice** tab, then select **Pressure Acoustics (acpr)>Sound pressure level** among the **Predefined quantities**. In the **Slice positioning** area type `0` in all three **Number of levels** edit fields. On the **y levels** line, click the **Vector with coordinates** button, then type `0.002` in the corresponding edit field.

5 Click the **Boundary** tab and select **Solid, Stress-Strain (acsld)>Total displacement** in the list of **Predefined quantities**. Click the **Colormap** button and select **hsv** from the corresponding list.
6 Click the **Deform** tab. In the **Domain types to deform** area, make sure that only the **Boundary** check box is selected. Click the **Boundary Data** tab, and verify that the selection in the **Predefined quantities** list is **Displacement**.
7 Click **OK**.

To refine the visual quality of the model, do as follows:

1 Click both the **Headlight** button and the **Scene Light** button on the Main toolbar.
2 Choose **Options>Visualization/Selection Settings**.
3 On the **Camera** page click the **Perspective** button in the **Projection** area.
4 Click the **Lighting** tab. In the **Scene light** area click all four light sources and clear the **Enabled** check box on each one of them.
5 Click the **New** button. Select **Spot** in the **Type** list. Click **OK**.
6 Specify the light source as in the following table, then click **OK**.

PROPERTY	VALUE
Position	-0.01 -0.01 0
Direction	0 1 0
Spread angle	90
Concentration	0.05

You can experiment with the viewing angle by clicking the **Zoom** and **Dolly In/Out** buttons on the Plot toolbar and clicking and dragging the geometry.

Point-Source Version

Once you are done solving and postprocessing the line-source version of this model, save it and proceed to set up a point-source version. Whereas the line source gives an axially symmetric pressure field, the point source is displaced from the origin and thus motivates a 3D model. Starting from your line source model, do the following to shift to a point source version.

PHYSICS SETTINGS

Point Settings—Pressure Acoustics

Choose **Physics>Point Settings**. In the dialog box that appears select Point 20. In the **Type of source** list select **Power** and set **P** to `1` W.

Edge Settings—Pressure Acoustics

To turn off the line source choose **Physics>Edge Settings**. In the dialog box that appears select Edge 26 and set **P** to `0`.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The location of the slice plot now coincides with the point source, where the pressure field is singular. You get a nicer plot if you move the slice away from the source:

1. In the **Plot Parameters** dialog box, click the **Slice** tab.
2. In the **y levels** edit field, type `0` in the **Vector with coordinates** column.
3. Click **OK**.

Your plot should now resemble Figure 2-8 on page 23.

Far-Field Postprocessing

1. Choose **Physics>Boundary Settings** and go to the **Far-Field** page.
2. Select Boundaries 1–4, 13–14, and 17–18.
3. Define a variable with the **Name** `pfar`.
4. Click **OK** to close the dialog box.
5. Choose **Solve>Update Model**.

   You can now plot the far pressure field in the direction from the origin towards any point To see, for instance, the field as a function of the elevation angle along Edge 16, do the following:
6. Choose **Postprocessing>Domain Plot Parameters**.
7. On the **Line/Extrusion** page, select Edge 16.
8. From the **Predefined quantities** list, select **Pressure Acoustics (acpr)>Sound pressure level for pfar**.

9 For the **x-axis data** click the **Expression** button and specify the expression `atan2(-y,z)`. Click **OK** to close the dialog box and then click **OK** again to see the plot. Verify that it looks similar to Figure 2-10.

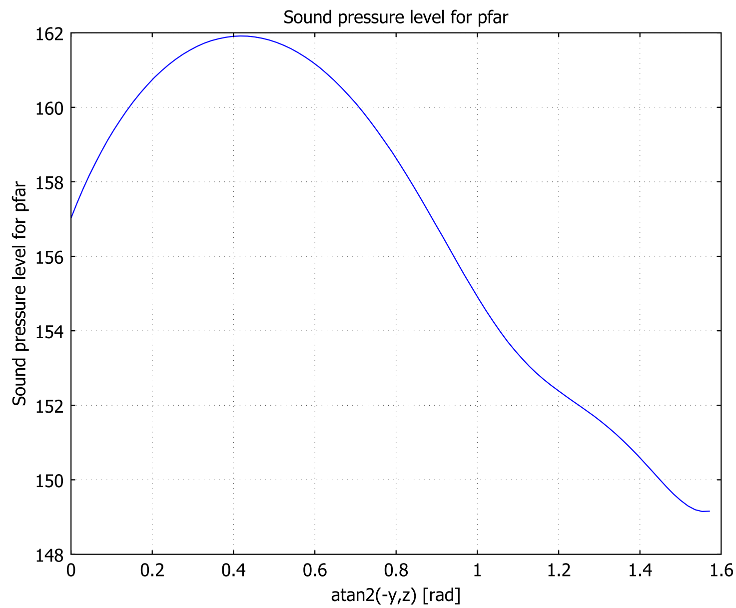


Figure 2-10: The far-field sound pressure level in dB as a function of the polar angle from 0 to $\pi/2$ *at the fixed azimuthal angle of* $-\pi$ *and a distance of 1 m from the source.*

If you are patient and have some spare time, you can try to plot the far-field radiation pattern at all possible angles around the sphere. This is typically done as a deformed surface plot. Because postprocessing the far field is computationally heavy this might take several minutes.

1 Choose **Options>Constants**. Add the following constants; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`Lpfarmin`	`122`	`Min sound pressure level (dB)`
`Lpfarmax`	`163`	`Max sound pressure level (dB)`

2 Choose **Options>Expressions>Boundary Expressions**.

3 On Boundaries 1–4, 13–14, and 17–18, define the variable `deformation` as `(Lpfar_acpr-Lpfarmax)/(Lpfarmax-Lpfarmin)`.

4 Click **OK**.

5 Choose **Solve>Update Model**.

6 Open the **Plot Parameters** dialog box. On the **General** page, select the check boxes for **Boundary** and **Deformed shape** only.

7 Click the **Boundary** tab. From the **Predefined quantities** list, select **Pressure Acoustics (acpr)>Sound pressure level for pfar**.

8 Click the **Deform** tab, then click the **Boundary Data** tab. In the **Domain types to deform** area select the **Boundary** check box only. Enter the following **Boundary Data**:

x component	`deformation*x`
y component	`deformation*y`
z component	`deformation*z`

9 Click **OK** and wait for a few minutes to see the plot.

10 Rotate the plot to explore the directional dependence of the pressure in the far-field region. If you prefer, you can manually enter settings from **Options>Visualization/ Selection Settings**. For a nice view, click the **Camera** tab and enter the following data:

PROPERTY	VALUE
Camera position	`0.18 0.39 0.29`
Camera target	`0 0 0`
Camera up vector	`0.11 0.55 -0.83`
Camera view angle	`11.6`

Figure 2-11 shows a zoom in of the resulting radiation pattern.

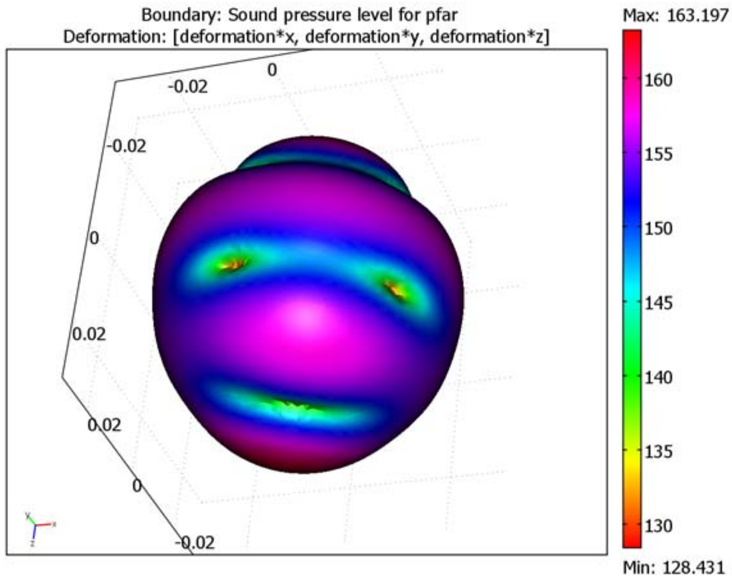


Figure 2-11: Radiation pattern of the far-field sound pressure level at a distance of 1 m from the source. The lower the pressure, the larger the inward deformation from the original spherical boundary.

This plot gives a useful overview of the far-field sound pressure level. The first thing to notice is that the plot has a mirror symmetry with respect to the plane spanned by the *z*-axis and the point of excitation. This may be easier to verify by inspection if you turn off the light sources. Furthermore, there are two distinct maxima, above and below the end caps of the cylinder, respectively. The minima approximately correspond to nodes in the structural deformation.

Optimizing the Shape of a Horn

Introduction

This model 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, Martin Berggren, and others (Ref. 1).

Note: This model requires the Optimization Lab.

Model Definition

A plane-wave mode feeds an axisymmetric horn radiating from an infinite baffle towards an open half space. 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.

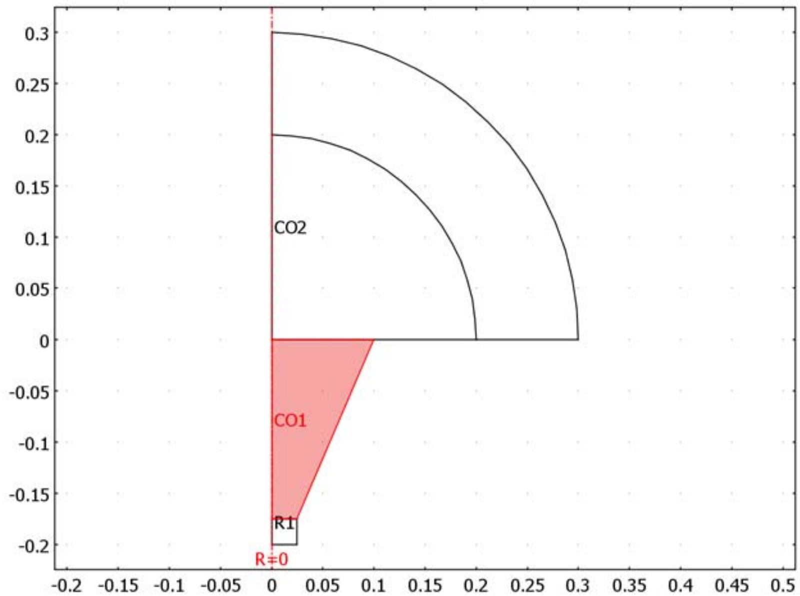


Figure 2-12: The initial configuration is a simple cone.

The surface is parameterized by assuming that the radius of the horn (as function of the distance from the baffle) deviates from the simple cone by a function of the form

$$d_r = \sum_{i=1}^{N} q_i d_i \sin(i\pi s) \tag{2-1}$$

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 will also make the optimization process more sensitive and may generate a shape which 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 will have to generate a scalar, real-valued quantity to be used as *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 COMSOL's Optimization Lab 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

$$\begin{aligned} \frac{d}{dz}|z| &= \frac{\bar{z}}{|z|} \\ \frac{d}{dz_1}(z_1\overline{z_2}) &= \overline{z_2} \\ \frac{d}{dz_2}(z_1\overline{z_2}) &= \overline{z_1} \end{aligned} \tag{2-2}$$

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 real(z_1·conj(z_2)) but differentiates according to Equation 2-2. In particular, as a measure of the transmission properties of the horn, you will use a an expression of the form realdot(p_m, p_m)/$p_0{}^2$, where p_m is the pressure measured at a specific point in front of the horn and p0 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 function of the angle from the axis.

COMSOL Multiphysics contains optimized code for evaluating such far-field integrals. This is, however, a pure post-processing feature which does not support the automatic differentiation required by the adjoint method. Therefore, you will have to return to the definition of the Helmholtz-Kirchhoff integral as given in its asymptotic axisymmetric form by Equation 3-5 on page 33 in the *Acoustics Module User's Guide*:

$$p_{\mathrm{far}}(\mathbf{R}) \cong -\frac{1}{2}\int_S r e^{ik\frac{zZ}{|\mathbf{R}|}}\left[J_0\left(\frac{krR}{|\mathbf{R}|}\right)\nabla p(\mathbf{r})\cdot\mathbf{n} - \frac{ikp(\mathbf{r})}{|\mathbf{R}|}\left(in_r R J_1\left(\frac{krR}{|\mathbf{R}|}\right) + n_z Z J_0\left(\frac{krR}{|\mathbf{R}|}\right)\right)\right]dS \tag{2-3}$$

If the infinite baffle is placed at $z = 0$, its effect is the same as if 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 2-3 cancel out, and all that is left is

$$p_m(\Theta) = \int_S r J_0(kr\sin(\theta))\frac{\partial p}{\partial z}dr \tag{2-4}$$

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 as an integration coupling variable.

Optimization as a rule implies many evaluations of the model for different designs, which may 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 evaluated for each frequency. In this tutorial, a single frequency of 4000 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 2-12.

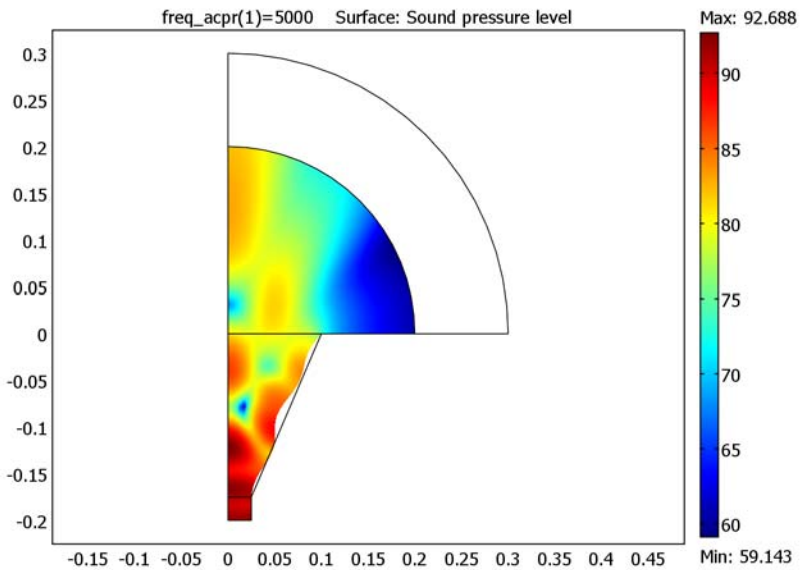


Figure 2-13: The final shape of the horn, optimized for on-axis SPL at 5000 Hz.

The improvement is rather small, because also the initial configuration shows a marked directivity, as can be seen from Figure 2-14. Obviously, the optimal shape with respect to on-axis SPL leads to deep undesirable minima in other directions.

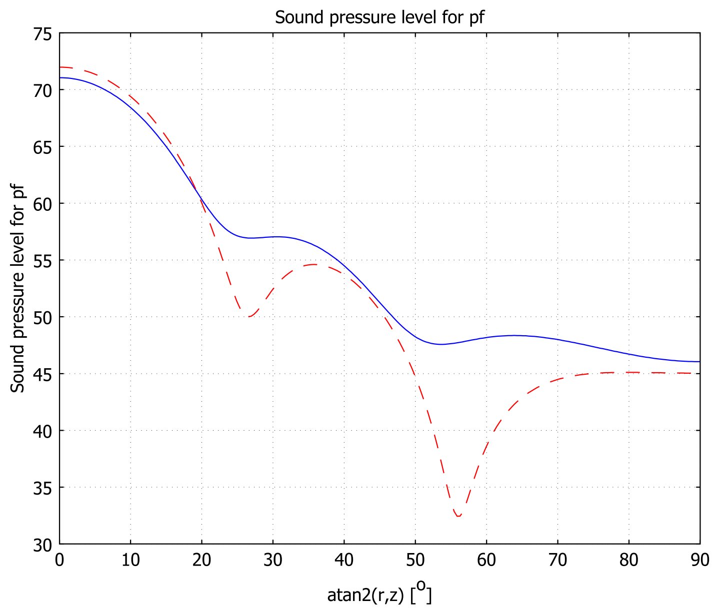


Figure 2-14: Far-field patterns for the original (solid blue) and final (dashed red) designs.

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.

Modeling in COMSOL Multiphysics

COMSOL Multiphysics implements the parametrization as a prescribed boundary displacement in a Moving Mesh application mode. 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 will be 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 will perform this sanity check as a final postprocessing step.

A time-harmonic Pressure Acoustics application mode 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 matched boundary condition on the waveguide attached to the narrow end feeds the horn with a plane wave of amplitude p_0.

An Optimization application mode adds five scalar optimization variables, q_1 to q_5, which 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 2-1. The pressure measured by an integration coupling variable according to Equation 2-4 is inserted into a scalar objective function contribution equal to the negative of the SPL value, or $-10 \cdot \log_{10}(0.5 \cdot \mathrm{realdot}(p_m, p_m)/(20 \cdot 10^{-6})^2)$.

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.

Model Library path: `Acoustics_Module/Tutorial_Models/horn_shape_optimization`

MODEL NAVIGATOR

1 In the **Model Navigator**, select **Axial Symmetry (2D)** from the **Space dimension** list, then go to the list of application modes and select **COMSOL Multiphysics>Deformed Mesh>Moving Mesh (ALE)**.

2 Click the **Multiphysics** button, then click **Add**.

3 Select **Acoustics Module>Pressure Acoustics** from the application mode list, then click **Add**.

4 Select **COMSOL Multiphysics>Optimization and Sensitivity>Optimization**, then click **Add**.

5 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1 Shift-click the **Rectangle/Square** button on the Draw toolbar. In the dialog box that appears, change the following settings; when done, click **OK**.

Size	Width	`0.025`
	Height	`0.025`
Position	z	`-0.2`

2 Create another rectangle with the following changes compared to the defaults:

Size	Width	`0.3`
	Height	`0.3`

3 Zoom in to the geometry by clicking the **Zoom Extents** button on the Main toolbar.

4 Click the **Ellipse/Circle (Centered)** button on the Draw toolbar. Right-click at the origin and drag to create a circle of radius 0.3.

5 Draw another circle of radius 0.2 inside the first one.

6 Shift-click to select both circles, then click the **Union** button on the Draw toolbar.

7 Select the joined circles together with the larger of the rectangles (R2), then click the **Intersection** button.

8 To draw the actual horn, click the **Line** button on the Draw toolbar; click consecutively at (0, 0), (0.1, 0), (0.025, -0.15), (0, -0.15), and (0, 0); and finally right-click once anywhere in the geometry to close the curve and turn it into a solid object.

OPTIONS AND SETTINGS

1 Choose **Options>Constants**.

2 Define the following constants; when finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
r0	0.025[m]	Radius of waveguide
p0	1[Pa]	Incident pressure amplitude
theta	0[deg]	Polar angle
d1	0.02[m]	Scale factor
d2	0.01[m]	Scale factor
d3	0.01[m]	Scale factor
d4	0.01[m]	Scale factor
d5	0.01[m]	Scale factor

3 Choose **Options>Expressions>Global Expressions** and add the expression for the radial displacement of the horn surface:

NAME	EXPRESSION	DESCRIPTION
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)	Radial displacement

4 Choose **Options>Integration Coupling Variables>Boundary Variables**.

5 Select Boundary 6 and create a variable with the following properties:

NAME	EXPRESSION	INTEGRATION ORDER	FRAME	GLOBAL DESTINATION
pm	r*besselj(0,k_acpr*r* sin(theta))*pz	4	ref	yes

6 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings—Moving Mesh (ALE)

1 From the **Multiphysics** menu, select the **Moving Mesh (ALE)** application mode.

2 Choose **Physics>Subdomain Settings** to open the **Subdomain Settings** dialog box.

3 Select Subdomains 1, 3, and 4, click the **No displacement** button, and then click **OK**.

Boundary Conditions—Moving Mesh (ALE)

4 From the **Physics** menu, open the **Boundary Settings** dialog box.

5 Select Boundaries 3, 4, and 6.

6 Select the **dr** and **dz** check boxes to lock these boundaries at their initial positions.

7 Select Boundary 9, again select the **dr** and **dz** check boxes, and enter `dr` in the **Mesh displacement, r direction** edit field.

8 Click **OK** to close the dialog box.

Subdomain Settings—Pressure Acoustics

The default medium in the Pressure acoustics application mode is air, so the only thing you have to do is activate the PML domain.

1 From the **Multiphysics** menu, choose the **Pressure Acoustics** application mode.

2 From the **Physics** menu, open the **Subdomain Settings** dialog box.

3 Select Subdomain 4 and click the **PML** tab.

4 From the **Type of PML** list, choose **Spherical**.

5 Select the **Absorbing in radial dir.** check box, then click **OK** to close the dialog box.

Boundary Conditions—Pressure Acoustics

The default boundary condition is *sound hard*, which is appropriate for the horn surface and the baffle, and does no harm on the outside of the PML or on the axis. Therefore, only the waveguide port requires the boundary condition to be changed. In addition, far-field postprocessing variables must also be defined in **Boundary Settings**.

1 Choose **Physics>Boundary Settings**.

2 Select Boundary 2. From the **Boundary condition** list, choose **Matched boundary** and set the **Pressure source** field to `p0`.

3 Click the **Interior boundaries** check box, then select Boundary 6.

4 On the **Far-Field** page, type `pf` in the first row of the **Name** column.

5 Press Tab to automatically fill in the remaining columns.

6 Select the **z=0** check box under **Symmetry planes** but leave the list box at **Symmetric Pressure**. This procedure accounts for the infinite baffle.

7 Click **OK** to close the dialog box.

Scalar Settings—Optimization

In the Optimization application mode, you define the objective function, declare optimization variables, and constraints.

8 Activate the **Optimization** application mode by selecting it from the **Multiphysics** menu or in the **Model Tree**.

9 Choose **Physics>Scalar Settings** to open the **Scalar Settings** dialog box.

10 In the **Scalar contribution** field on the **Objective** page, enter `-2*pi*realdot(pm,pm)/(pi*r0^2*p0^2)`.

11 Click the **Variables** tab. Enter **Variable** names `q1`, `q2`, `q3`, `q4`, and `q5` on separate lines in the table, all with **Init** value `0`.

12 Click the **Scalar Constraints** tab and limit the allowed value of each of the optimization variables to the range [−1, 1]. When you are done, the **Scalar constraints** table should look as follows:

LB	EXPRESSION	UB
-1	q1	1
-1	q2	1
-1	q3	1
-1	q4	1
-1	q5	1

13 Click **OK** to close the dialog box.

GENERATING THE MESH

The model will be run 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. 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.

1 Choose **Mesh>Free Mesh Parameters**.

2 Click the **Custom mesh size** option button, type `0.01` in the **Maximum element size** edit field, and then click **OK** to close the dialog box.

3 Go to **Subdomain Mode** by clicking the corresponding button on the main toolbar.

4 Select all subdomains except the PML, then click the **Mesh Selected (Free, Quad)** button on the Mesh toolbar.

5 To set the number of mesh layers in the PML, begin by selecting **Mapped Mesh Parameters** from the **Mesh** menu.

6 On the **Boundary** page, select Boundary 7 and then select the **Constrained edge element distribution** check box.

7 Set the **Number of edge elements** to `8`, then click **OK**.

8 Click the **Mesh Remaining (Mapped)** button on the Mesh toolbar to complete the mesh.

COMPUTING THE SOLUTION

Before starting the actual optimization solver it is good practice to check the model set-up by running the sensitivity solver once. This way, you can also study the reference state on which you intend to improve.

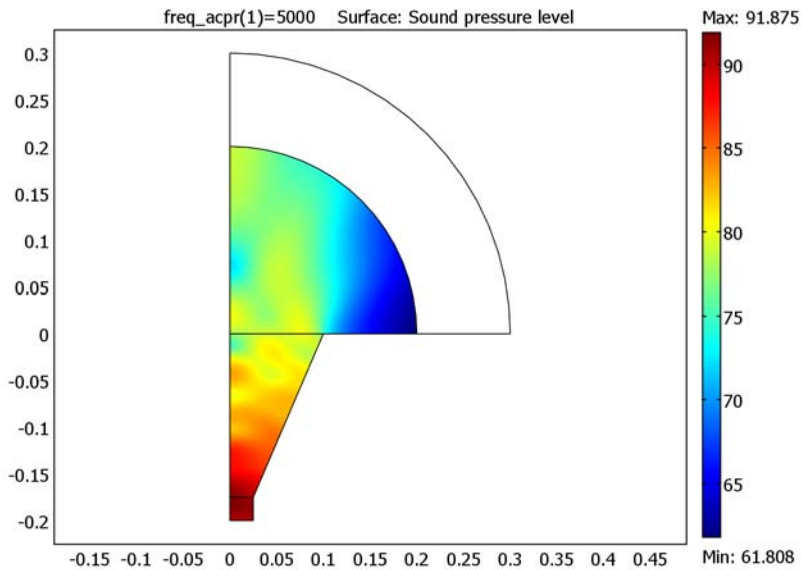
1 Choose **Solve>Solver Parameters** or click the corresponding button on the Main toolbar to open the **Solver Parameters** dialog box.
2 Select **Parametric** from the **Solver** list and select the **Optimization/Sensitivity** check box.
3 In the **Parameter Names** field, enter the name of the frequency variable: `freq_acpr`.
4 As **Parameter values**, enter just `5000`.
5 Click the **Optimization/Sensitivity** tab and verify that the **Analysis** is set to **Sensitivity** rather than **Optimization**.
6 Click **Apply** in the dialog box, then click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot in the main window shows the mesh displacement in the Moving Mesh application mode. Because the initial optimization variable values are zero, so is the displacement. The acoustical quantities are far more interesting for visualization, but for clarity, the PML zone should not be included in the plot.

1 Click the **Plot Parameters** button on the Main toolbar to open the dialog box with that name.
2 On the **Surface** page, select **Pressure Acoustics (acpr)>Sound pressure level** from the list of **Predefined quantities**.
3 Click **OK** to close the dialog box and display the SPL value.
4 Go to **Subdomain Mode** by clicking the corresponding button on the Main toolbar.
5 Select the PML domain, then click **Hide Selected Objects**.

6 Return to **Postprocessing Mode** using the button on the Main toolbar.



7 To see the far-field SPL pattern, first choose **Postprocessing>Domain Plot Parameters**.

8 On the **Line/Extrusion** page, select Boundary 12 and find **Pressure Acoustics (acpr)>Sound pressure level for pf** in the list of **Predefined quantities**.

9 Click the lower option button in the **x-axis data** area, then click the **Expression** button to open the **X-Axis Data** dialog box.

10 Type `atan2(r,z)` in the **Expression** field, press Tab, and select the degree sign from the **Unit** list.

11 Click **OK** to close the dialog box, then click **Apply** in the **Domain Plot Parameters** dialog box to display the far-field pattern.

COMPUTING THE SOLUTION

Now return to solving the actual optimization problem. In this tutorial, the optimization is performed only for a single frequency. This can also be seen as a preparation step before starting a time-consuming optimization over a frequency range.

1 Return to the **Optimization/Sensitivity** page in the **Solver Settings** dialog box and change the **Analysis** type to **Optimization**.

2 Increase the **Optimality tolerance** to `1e-4`, which is still stricter than the accuracy of this low-resolution finite element model.

3 Optionally, to follow the optimization solver's progress, select the **Plot while solving** check box. When you run the optimization, the software then plots the SPL field (according to the current settings in the **Plot Parameters** dialog box) in a separate figure window and updates it after each model evaluation.

4 Click **OK** to close the dialog box, then click the **Solve** button on the Main toolbar.

The solution requires about 20 model evaluations, which should not take more than a minute on a modern computer. You can follow the

POSTPROCESSING AND VISUALIZATION

When the solution is ready, you can immediately study the optimal (within the constraints imposed by the parametrization) shape of the horn. To see a direct comparison of the far-field pattern before and after optimization (Figure 2-14) do the following:

1 Return to the **Domain Plot Parameters** dialog box, select again Boundary 12 on the **Line/Extrusion** page, and click the **Line Settings** button.

2 In the **Line Settings** dialog box, select **Color** from the **Line color** list. Click **OK**.

3 Click the **General** tab. Select the **Keep current plot** check box, then click **Apply**.

The solution returned by the optimization solver is only guaranteed to be a local optimum within the given constraints. It is good practice to check whether any constraints imposed to keep the problem well posed (in this case avoid inverted mesh elements) are active at the optimum point. If some artificial constraint is active, try to relax it a little and restart the solution.

4 Choose **Postprocessing>Data Display>Global**.

5 From the **Predefined quantities** list, select **Optimization (opt)>q1**.

6 Click **Apply** and check that the value lies inside the specified range $-1 < q_1 < 1$.

7 Check q_2, q_3, q_4, and q_5 in the same way.

8 When done, click **OK**.

Because none of the parameters take on a limiting value, the obtained solution can be assumed independent of the artificial bounds.

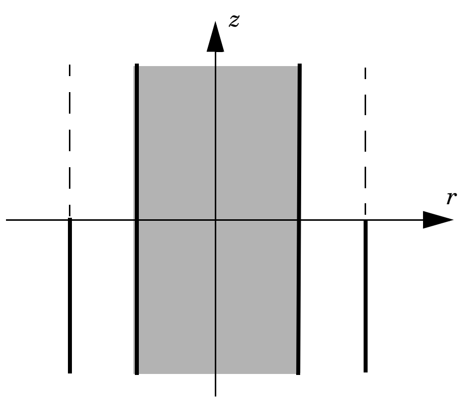
Jet Pipe

Introduction

This example models the radiation of fan noise from the annular duct of a turbofan aeroengine. When the jet stream exits 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.

Model Definition

The model is axisymmetric with the symmetry axis coinciding with the engine's centerline. The flows both inside and outside the duct are uniform mean flows, but because the flow velocities differ, a vortex sheet separates them.



The Aeroacoustics application mode 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 "Aeroacoustics" on page 107 of the *Acoustics Module User's Guide* for further details. However, the field equation is only valid when the 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 assemblies that are connected through pairs. The following boundary condition on the pair models the vortex sheet:

$$\rho(i\omega + M_1\nabla_T)w = -\mathbf{n}\cdot\left(\rho\nabla\phi_1 - \frac{\mathbf{V}}{c^2}\rho(i\omega\phi_1 + (\nabla\phi_1\cdot\mathbf{V}))\right)$$

$$\rho(i\omega + M_2\nabla_T)w = -\mathbf{n}\cdot\left(\rho\nabla\phi_2 - \frac{\mathbf{V}}{c^2}\rho(i\omega\phi_2 + (\nabla\phi_2\cdot\mathbf{V}))\right)$$

$$p_1 = p_2$$

In these equations, ω is the angular velocity, V is the mean flow velocity, w is the normal displacement, ϕ is the velocity potential, and p is the pressure. The subscripts 1 and 2 refer to the two sides of the boundary.

The velocity normal to the vortex sheet is zero, which implies that the last term in 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

$$(i\omega + M_1\nabla_T)w = \frac{\partial\phi_1}{\partial n}$$

$$(i\omega + M_2\nabla_T)w = \frac{\partial\phi_2}{\partial n}$$

$$p_1 = p_2$$

where M denotes the transverse Mach number.

The duct has a hard wall, which you also model using a boundary condition on a pair.

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, $n = 0$.

Results and Discussion

The boundary mode analysis made with the circumferential wave number $m = 4$, 17, and 24 gives several eigenmodes corresponding to different radial mode numbers. This example, as well as in Ref. 1, uses the following eigenmodes as incident waves in the duct.

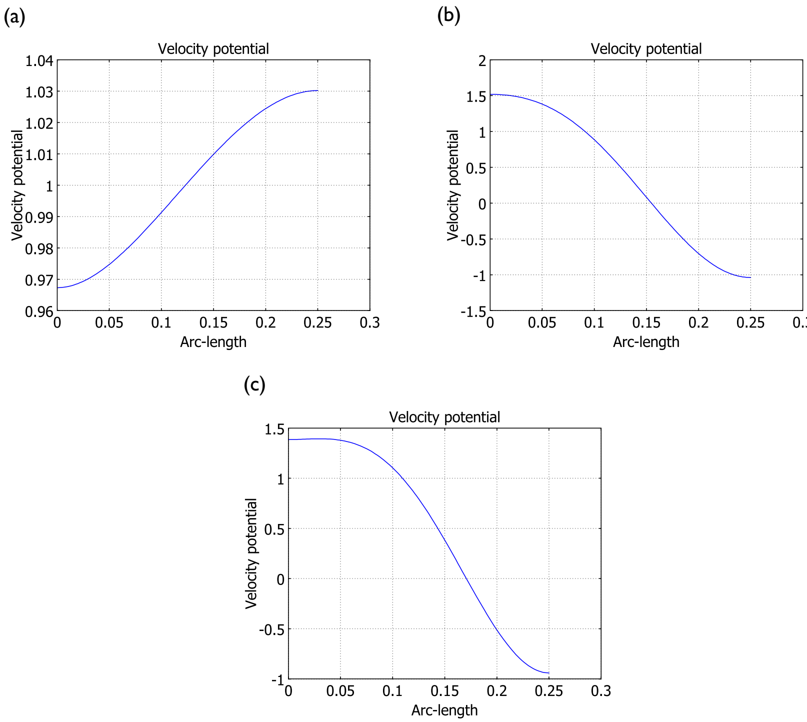


Figure 2-15: (a) Mode shape for $m = 4$, $n = 0$; (b) Mode shape for $m = 17$, $n = 1$; (c) Mode shape for $m = 24$, $n = 1$.

The near field around the duct obtained by COMSOL Multiphysics can be compared to the results for the near field in Ref. 1. Figure 2-16 to Figure 2-18 show the near-field solution for a Mach number equal to 0.45 in the pipe and 0.25 on the outside. The figures show the field for the different eigenmodes shown in Figure 2-15.

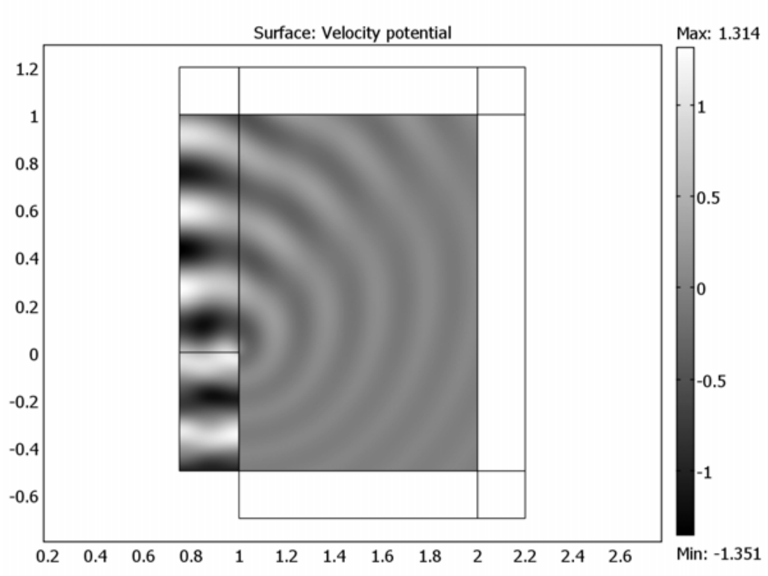


Figure 2-16: The near-field solution for m = 4 and n = 0.

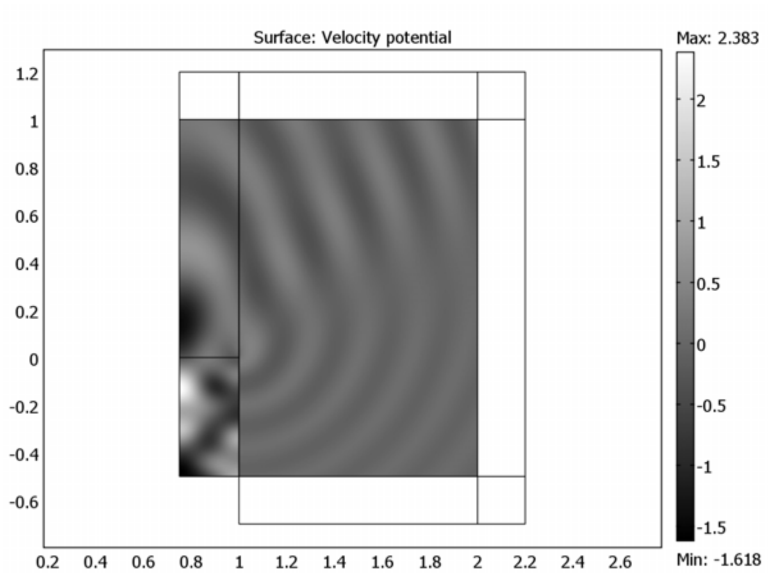


Figure 2-17: The near-field solution for m = 17 and n = 1.

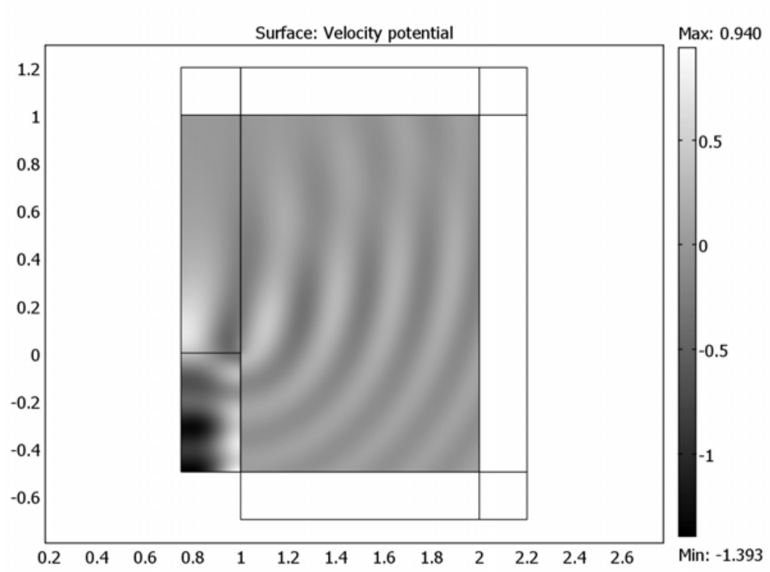


Figure 2-18: The near-field solution for m = 24 and n = 1.

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.

Model Library path: `Acoustics_Module/Tutorial_Models/jet_pipe`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator** go to the **Space dimension** list and select **Axial symmetry (2D)**, then click the **Multiphysics** button.
2. From the list of application modes select **Acoustics Module>Aeroacoustics>Boundary modal analysis**. In the **Dependent variables** edit field type `phi_b`. Click **Add**.

3 From the list of application modes select **Acoustics Module>Aeroacoustics>Time-harmonic analysis**. Click **Add**.

4 Click **OK**.

GEOMETRY MODELING

1 From the **Draw** menu select **Specify Objects>Rectangle**.

2 Specify the following rectangles, all with **Corner** as the **Position Base**.

RECTANGLE	WIDTH	HEIGHT	R	Z
R1	0.25	0.5	0.75	-0.5
R2	0.25	1	0.75	0
R3	0.25	0.2	0.75	1
R4	1	1.5	1	-0.5
R5	1.2	0.2	1	-0.7
R6	0.2	1.9	2	-0.7
R7	1.2	0.2	1	1

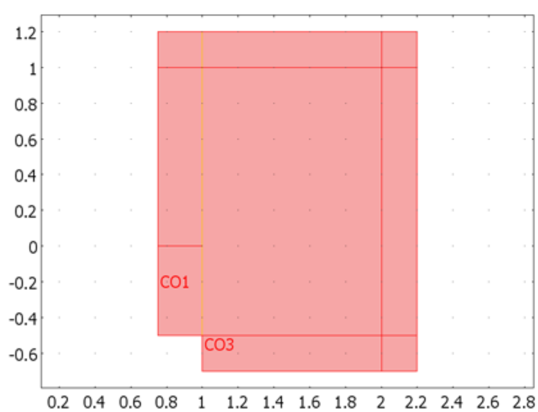
3 Click the **Zoom Extents** button.

4 Select R1, R2, and R3, then click the **Union** button on the **Draw** toolbar.

5 Select R4, R5, R6, and R7, then click the **Union** button.

6 Select CO1 and CO2, then click the **Create Pairs and Imprints** button.

This completes the geometry-modeling state. The geometry in the drawing area of the user interface on your screen should now like that in the figure below.



OPTIONS AND SETTINGS

Enter the following constants in the **Constants** dialog that you open from the **Options** menu. When finished, click **OK**.

NAME	VALUE	DESCRIPTION
`M0`	`0.25`	`Mach number outside the duct`
`M1`	`0.45`	`Mach number inside the duct`
`m`	`4`	`Circumferential wave number`

PHYSICS SETTINGS

Units

1 From the **Physics** menu, select **Model Settings**.

2 Set the **Base unit system** to **None**.

3 Click **OK**.

Identity Pairs

1 From the **Physics** menu, select **Identity Pairs>Identity Boundary Pairs**.

2 Click **New** and check Boundary 8 as source boundary and Boundary 13 as destination boundary.

3 Select **Pair 1** and clear the check boxes for Boundaries 8 and 13 that you selected in the previous step.

4 Click **OK**.

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.

2 Select Subdomains 1–3. Set $\mathbf{V_z}$ to `M1`, and both $\mathbf{c_s}$ and ρ to `1`.

3 Select Subdomains 4–9. Set $\mathbf{V_z}$ to `M0`, and both $\mathbf{c_s}$ and ρ to `1`.

4 Go to the **PML** page.

5 Select Subdomains 3, 4, 6, 7, and 9. Set **Type of PML** to **Cylindrical** and select the **Absorbing in z direction** check box.

6 Select Subdomains 7–9.

The settings that differ between the subdomains are now highlighted in yellow; these settings will not be affected unless you explicitly change them.

7 From the **Type of PML** list, select **Cylindrical**; this changes the setting for Subdomain 8. Select the **Absorbing in r direction** check box.

8 Click **OK**.

Boundary Conditions

1 From the **Multiphysics** menu, select **Aeroacoustics, Boundary Modal Analysis (acab)**.
2 From the **Physics** menu, open the **Boundary Settings** dialog box.
3 Select all boundaries and clear the **Active in this domain** check box.
4 Select Boundary 2 only, then select the **Active in this domain** check box.
5 Set $\mathbf{V_z}$ to `M1` and both $\mathbf{c_s}$ and ρ to `1`.
6 Click **OK**.

Application Scalar Variables

1 From the **Physics** menu, select **Scalar Variables**.
2 Set the **Excitation frequency** to `30/(2*pi)` and the **Circumferential wave number** to `m`.
3 Click **OK**.

GENERATING THE MESH

1 From the **Mesh** menu, select **Mapped Mesh Parameters**.
2 From the **Predefined mesh sizes** list, select **Extremely fine**. Click **Remesh**.
3 When the mesh is finished, click **OK** to close the dialog box.

COMPUTING THE SOLUTION

1 From the **Solve** menu, open the **Solver Parameters** dialog box.
2 Set **Desired number of propagation constants** to `10`.
3 Click **OK**.
4 From the **Solve** menu, open the **Solver Manager**.
5 On the **Solve For** page, select **Aeroacoustics, Boundary Modal Analysis (acab)**.
6 Click **Apply**, then click **Solve**.
7 On the **Initial Value** page click the **Store Solution** button and click **OK** in the dialog box that appears to store all solutions.
8 Click **OK** in the **Solver Manager** dialog box to close it.

POSTPROCESSING AND VISUALIZATION

1 From the **Postprocessing** menu, select **Domain Plot Parameters**.
2 On the **Line/Extrusion** page select Boundary 2 and make sure the **Expression** is `phi_b`.

3 On the **General** page select the value with the highest real part in the **Solutions to use** list. Click **Apply**.

   The figure that appears is the same as in Figure 2-15 (a). This corresponds to the lowest radial mode ($n = 0$).

4 You can look at higher modes by selecting values with lower real part in the **Solution to use** list.

   When you solve for $m = 17$ or 24 you can generate Figure 2-15 (b) or (c) by selecting the value with the second highest real part, $n = 1$.

5 Click **OK** to close the dialog when you are ready.

PHYSICS SETTINGS

Boundary Conditions

1 From the **Multiphysics** menu select **Aeroacoustics(acae)**.

2 Select Boundary 2. Set the **Boundary condition** to **Velocity potential** and ϕ_0 to `phi_b`.

3 Go to the **Pairs** page and select **Pair 1**. Set the **Boundary condition** to **Vortex sheet**.

4 Select **Pair 2** and set the **Boundary condition** to **Sound hard boundary (wall)**.

5 Click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu, open the **Solver Manager**.

2 Set the **Initial value** to **Stored solution** and the **Propagation constant** to the one with highest real part to get the lowest radial mode.

3 On the **Solve For** page, select **Aeroacoustics (acae)**.

4 Click **OK**.

5 Open the **Solver Parameters** dialog and set the solver to **Stationary**. Click **OK**.

6 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Follow these steps to generate Figure 2-16 on page 51.

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.

2 On the **Surface** page, select the **Surface plot** check box, then set the **Colormap** to **gray**.

3 On the **General** page, set the **Solution at angle (phase)** to `180`.

4 Clear the **Boundary** check box in the **Plot type** area.

5 Click **OK**.

6 From the **Options** menu, choose **Suppress>Suppress Subdomains**.
7 Select Subdomains 3, 4, and 6–9, then click **OK**.
8 Click first the **Postprocessing Mode** button and then the **Zoom Extents** button.

You have now solved the example for the eigenmode $m = 4$ and $n = 0$. To generate Figure 2-17 on page 51 you need to solve the example with $m = 17$ and $n = 1$ and to generate Figure 2-18 you need to use $m = 24$ and $n = 1$. To solve the example again with a different eigenmode you can follow these steps.

1 From the **Options** menu, open the **Constants** dialog box.
2 Change the value of m to 17 to generate Figure 2-17 or 24 to generate Figure 2-18 in the **Constants** dialog. Click **OK**.
3 From the **Physics** menu, select **Boundary Settings**.
4 Select Boundary 2 and set ϕ_0 to 0, then click **OK**.

Go through the steps to solve the model once again with the new value of the circumferential mode number. Start from the section "Computing the Solution" on page 55.

When choosing the eigenmode to use as incident wave you need to take the one with second highest real part to get the radial mode number n equal to 1.

Finally, the image you see when opening this model from the Acoustics Module model library is produced from the last obtained solution ($m = 24$, $n = 1$) with the following steps:

1 Click the **Plot Parameters** button on the Main toolbar.
2 On the **General** page, select the **Contour** check box and clear the one for **Geometry edges**.
3 Click the **Surface** tab.
4 From the **Predefined quantities** list on the **Surface Data** page, select **Aeroacoustics (acae)>Sound pressure level**.
5 From the **Colormap** list inside the **Surface color** area select **jet**.
6 On the **Height Data** page, select the **Height data** check box.
7 From the **Predefined quantities** list select **Aeroacoustics (acae)>Sound pressure level**.
8 Click the **Contour** tab.
9 From the **Predefined quantities** list on the **Contour Data** page select **Aeroacoustics (acae)>Sound pressure level**.

10 Inside the **Contour levels** area, select the option button next to the **Vector with isolevels** edit field. Type `linspace(60,125,66)` in this edit field to obtain a contour-line distance of 1 dB.

11 Clear the **Color scale** check box inside the **Contour color** area.

12 On the **Height Data** page, select the **Height data** check box.

13 From the **Predefined quantities** list, select **Aeroacoustics (acae)>Sound pressure level**.

14 Go to the **General** page and click the **Title** button.

15 Select the option button next to the edit field and enter the title `Sound pressure level (dB)`, then click **OK** to close the **Title** dialog box.

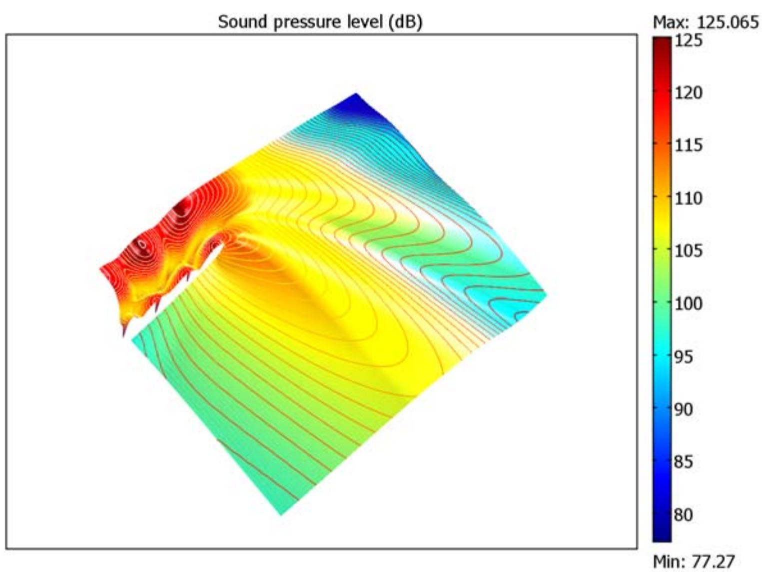
16 Click **OK** to generate the plot and close the **Plot Parameters** dialog box.

17 Click both the **Scene Light** button and the **Perspective Projection** button on the Camera toolbar to the left of the drawing area.

18 Double-click to clear the **AXIS** and **CSYS** buttons on the status bar at the bottom of the user interface.

19 Click the **Zoom Extents** button on the Main toolbar.

After rotating the view by clicking and dragging in the drawing area you should see a plot similar to that in the figure below.



Piezoacoustic Transducer

Introduction

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

Model Definition

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

This model simulates a single crystal plate in such a structure. The element is rotationally symmetric, making it possible to use an axisymmetric 2D application mode in COMSOL Multiphysics.

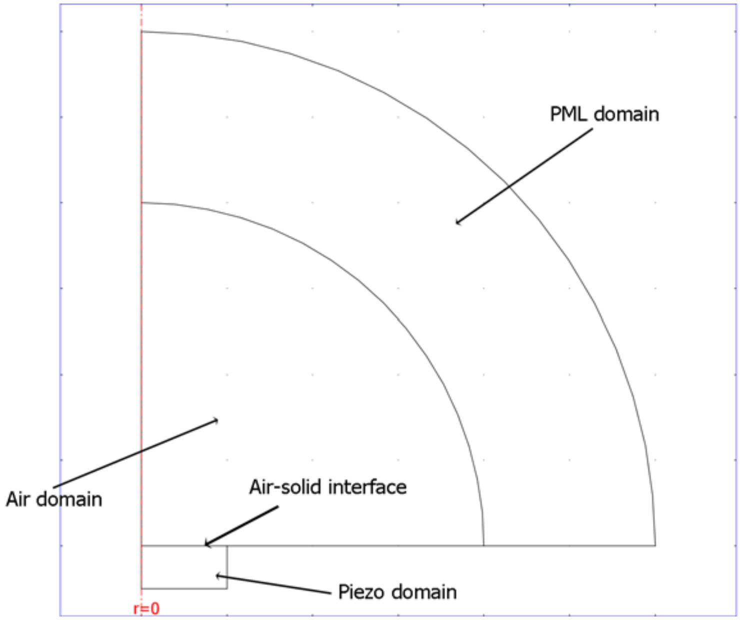


Figure 2-19: The model geometry.

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

$$\frac{1}{\rho_0 c_s^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) = Q \tag{2-5}$$

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

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

Hence Equation 2-5 simplifies to

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{\omega^2 p}{\rho_0 c_s^2} = Q \tag{2-6}$$

Because there are no sources present, Equation 2-6 simplifies further to

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

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

The frequency is set to 300 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

A voltage of 100 V is applied to the upper part of the transducer, while the bottom part is grounded. At the interface between the air and solid domain, the boundary condition for the acoustics application mode is that the pressure is equal to the normal acceleration of the solid domain

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

where a_n is the normal acceleration.

This drives the pressure in the air domain. The solid domain is on the other hand subjected to the acoustic pressure changes in the air domain. Because of the high voltage applied to the transducer, this load is probably negligible in comparison. Yet because the model is in 2D, it is possible to include this load and solve the full model simultaneously on any computer.

Results and Discussion

Figure 2-20 shows the pressure distribution in the air domain. This plot clearly shows how the PML (perfectly matched layer) absorbs the wave effectively.

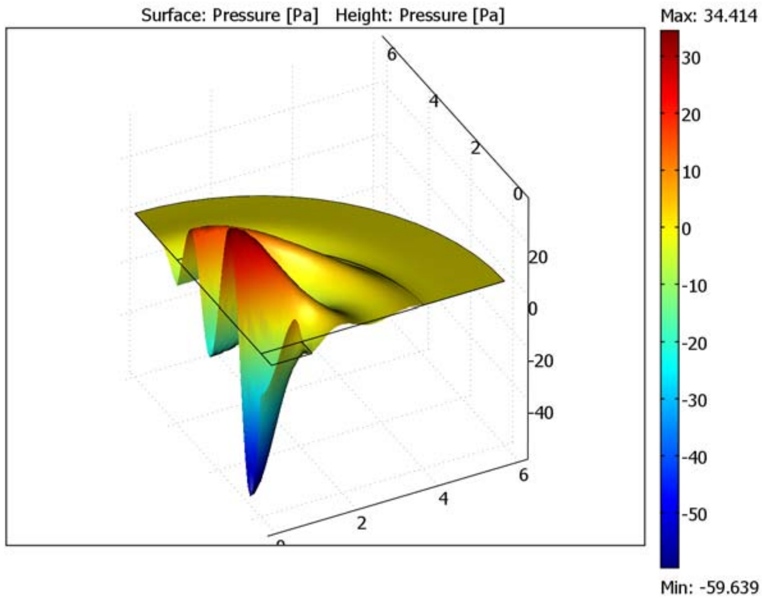


Figure 2-20: Surface and height plot of the pressure distribution.

Figure 2-21 shows the pressure distribution along the air-solid interface. The acoustic pressure load is small in comparison to the electrical load, which is plotted in Figure 2-22 on page 63.

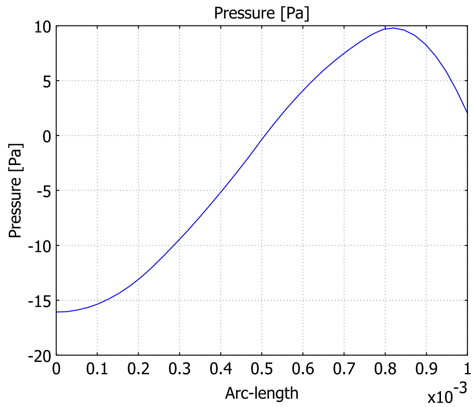


Figure 2-21: Acoustic pressure at the air-solid interface.

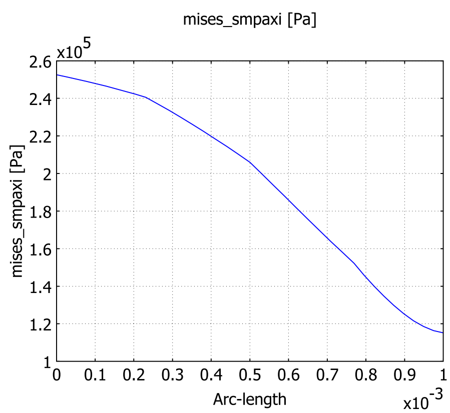


Figure 2-22: von Mises Stress along the air-solid interface.

The results from a far-field analysis appear in Figure 2-23 on page 64. This figure shows that the sound pressure level reaches a maximum right in front of the transducer. This result also shows that the sound pressure level is fairly low. Although humans cannot hear these high frequencies, it can be mentioned for comparison that 15 dB is about the same sound pressure level as rustling leaves. On the other hand, because this is just one element in an array of elements, a more detailed study is necessary in order to draw further conclusions.

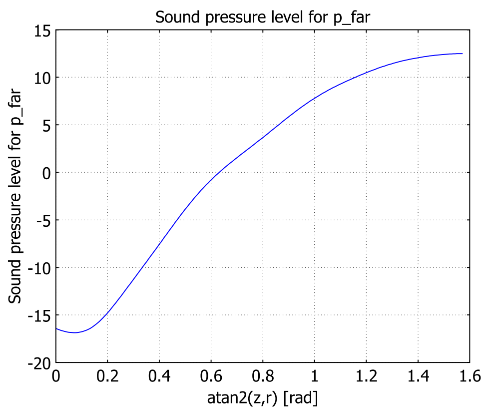


Figure 2-23: The far-field sound pressure level.

Model Library path: `Acoustics_Module/Tutorial_Models/piezoacoustic_transducer`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator**, begin by selecting **Axial symmetry (2D)** from the **Space dimension** list, then click the **Multiphysics** button.
2. Navigate to **Acoustics Module>Piezoelectric Effects>Piezo Axial Symmetry>Frequency response analysis**, then click **Add**.
3. Select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis**; then click **Add**.
4. Click **OK** to close the **Model Navigator**.

SCALAR VARIABLES

1. From the **Physics** menu, select **Scalar Variables**.
2. Select the **Synchronize equivalent variables** check box.
3. Enter a value of `200e3` Hz for the excitation frequency **freq_smpaxi**.

   COMSOL Multiphysics automatically updates the other excitation frequency, **freq_acpr**, to the same value.

4 Click **OK**.

GEOMETRY MODELING

1 Draw a rectangle, R1, by first selecting **Draw>Specify Objects>Rectangle** and then specifying the following properties; when done, click **OK**.

PROPERTY	VALUE
Width	`1e-3`
Height	`0.5e-3`
Position, base	Corner
Position, r	`0`
Position, z	`-0.5e-3`

2 Click the **Zoom Extents** button on the Main toolbar to automatically fit the geometry to your window.

The geometry for the transducer is now complete. Continue by creating the acoustics domain, which consists of two domains: one air domain and one PML domain.

3 Choose **Draw>Specify Objects>Circle**. Specify a **Radius** of `4e-3`, then click **OK**.

4 Choose **Draw>Specify Objects>Square**. Specify a **Width** of `4e-3`, then click **OK**.

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

6 Select the circle and the square, then click the **Intersection** button on the Draw toolbar.

7 Select the geometry object CO1. Press Ctrl+C to copy it, then press Ctrl+V and click **OK** in the dialog box that opens to paste the copy with no displacement.

8 Select the geometry object CO2, then click the **Scale** button. In the **Scale factor** area, type `1.5` in both the **r** and the **z** edit field. (You can select the geometry objects from a list in the **Create Composite Objects** dialog box.) Click **OK**.

9 Once again, click the **Zoom Extents** button on the Main toolbar to automatically fit the geometry to your window.

SUBDOMAIN SETTINGS—PIEZO AXIAL SYMMETRY

1 Select the **Piezo Axial Symmetry (smpaxi)** application mode from the **Model Tree** or from the **Multiphysics** menu.

2 From the **Physics** menu, choose **Subdomain Settings**.

3 Select Subdomains 2 and 3, then clear the **Active in this domain** check box.

4 Select Subdomain 1, then click the **Load** button.

5 From the **Basic Material Properties** library, select **Lead Zirconate Titanate (PZT-5H)**.

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

6 Click **OK** to close the **Materials/Coefficients Library** dialog box.

Note: For a piezoelectric material, you can specify an orientation and a coordinate system. In this model, use the default settings: the *xz*-plane in the global coordinate system.

---

7 Click **OK** to close the **Subdomain Settings** dialog box.

BOUNDARY CONDITIONS—PIEZO AXIAL SYMMETRY

1 From the **Physics** menu, choose **Boundary Settings**.

2 On the **Constraint** page, enter the following structural boundary conditions:

SETTINGS	BOUNDARY 1	BOUNDARY 2
Constraint condition	Symmetry plane	Roller

3 Click the **Electric BC** tab, and set the electric boundary condition as follows:

SETTINGS	BOUNDARY 1	BOUNDARY 2	BOUNDARY 4	BOUNDARY 6
Boundary condition	Axial symmetry	Ground	Electric potential	Zero charge/Symmetry
V_0			100	

4 Select Boundary 4.

5 On the **Load** page, type `-p` in the $\mathbf{F_z}$ edit field to specify the acoustic pressure load, and then click **OK**. `p` is the name of the dependent variable for pressure in the Pressure Acoustics application mode. The pressure acts from the air toward the piezo domain (in the negative *z* direction), which explains the minus sign in front of `p`. Click **OK** to close the dialog box.

SUBDOMAIN SETTINGS—PRESSURE ACOUSTICS

1 Select the **Pressure Acoustics (acpr)** application mode from the **Model Tree** or from the **Multiphysics** menu.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomain 1, then clear the **Active in this domain** check box.
4 Select Subdomain 3.
5 On the **PML** page, select **Spherical** from the **Type of PML** list.
6 Select the **Absorbing in radial dir.** check box, then enter a value of `2e-3`. This value corresponds to the PML's extension in the radial direction.
7 Set the inner PML radius, $\mathbf{R_0}$, to `4e-3`.
8 Click **OK** to close the dialog box.

Because the default values correspond to the properties of air, you do not have to specify the subdomain settings for Subdomain 2.

BOUNDARY CONDITIONS—PRESSURE ACOUSTICS

1 From the **Physics** menu, choose **Boundary Settings**.
2 Select Boundary 4, then select **Normal acceleration** as the boundary condition.
3 Set the value of the inward acceleration, $\mathbf{a_n}$, to `w_tt_smpaxi` (this is the second-order time derivative of the structural displacement).
4 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.
2 On the **Global** page, click the **Custom mesh size** option button.
3 Specify a **Maximum element size** of `(343/200e3)/5`. This value corresponds to 1/5th of the acoustic wavelength. For wave models it is important to use a mesh size sufficiently small to properly resolve the wavelength.
4 Click **Remesh**. When the mesher has finished, click **OK** to close the dialog box.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.
2 From the **Solver** list, select **Stationary**.
3 Click **OK** to close the dialog box.
4 Click the **Solve** button from the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To reproduce the plot in Figure 2-20, proceed as follows:

1 Click the **Plot Parameters** button on the Main toolbar.

2 Click the **Surface** tab.

3 On the **Surface Data** page, select **Pressure Acoustics (acpr)>Pressure** from the **Predefined quantities** list.

4 On the **Height Data** page, select the **Height data** check box.

5 From the **Predefined quantities** list, select **Pressure Acoustics (acpr)>Pressure**.

6 Click **OK** to generate the plot.

7 Click the **Headlight** button on the Camera toolbar for clearer visualization.

To create Figure 2-21 and Figure 2-22 do the following:

1 Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.

2 Go to the **Line/Extrusion** page.

3 In the **Expression** edit field, enter `p`.

4 From the **Boundary selection** list, select boundary 4 and then click on **Apply**.

5 To plot von Mises stress along the same boundary (Figure 2-22), enter `mises_smpaxi` in the **Expression** edit field, and then click **OK**.

To create Figure 2-23, you need to define a far-field variable.

1 Assure that the **Pressure Acoustics (acpr)** is selected from the **Model Tree**, then open the **Boundary Settings** dialog box.

2 Select Boundary 10, then click the **Far-Field** tab.

3 Type `p_far` in the **Name** edit field, then click **OK**.

4 From the **Solve** menu, choose **Update Model**.

5 From the **Postprocessing** menu, open the **Domain Plot Parameters** dialog box.

6 Go to the **Line/Extrusion** page. Select Boundary 10.

7 From the **Predefined quantities** list, select **Pressure Acoustics (acpr)>Sound pressure level for p_far**.

8 Click the **Expression** button in the **x-axis data** area, then click the **Expression** button.

9 In the **Expression** edit field, type `atan2(z,r)`. Click **OK**.

10 Click **OK** to create Figure 2-23.

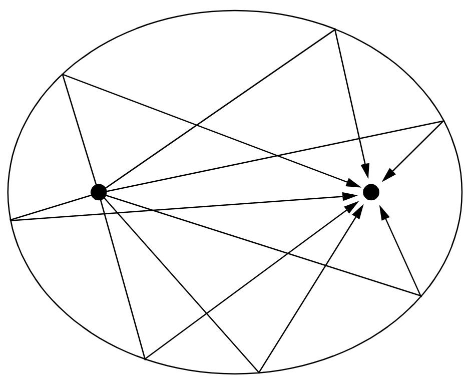
Transient Gaussian Explosion

Introduction

This model 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 (in meters) is the major axis length and c (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_s, the acoustic pressure field, $p(\mathbf{x}, t)$, inside the elliptical chamber is governed by the wave equation

$$\frac{1}{\rho c_s^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}\nabla p\right) = S(\mathbf{x}, t)$$

where the point-source term on the right-hand side is given by

$$S(\mathbf{x}, t) = \frac{dg}{dt}(t)\,\delta^{(2)}(\mathbf{x} - \mathbf{x}_0)$$

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 air flow (measured in 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 $f_0 = c/(Nh)$, where h is a typical 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 $N = 6$, but $N = 4$ should be acceptable for many purposes.

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.

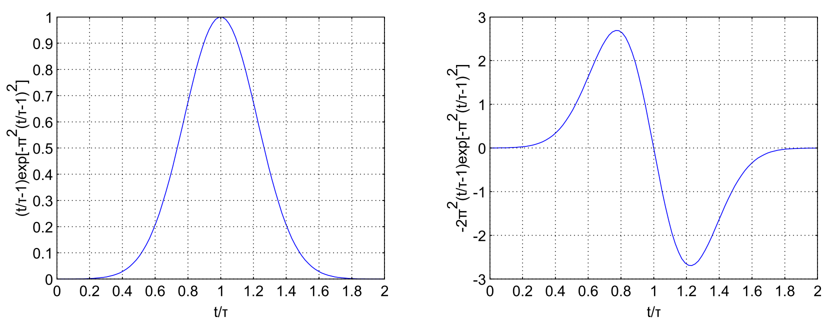


Figure 2-24: Normalized Gaussian pulse and its derivative.

A particularly interesting property of the Gaussian function is that its Fourier transform 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. The frequency was chosen on the basis of mesh-element size and resolving power of the spatial discretization, so, in practice, the pulse shape is a function of the mesh resolution. 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 and the amount of time the solver needs to take a single time step, the mesh resolution imposes a restriction on the time step size. 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.

COMSOL Multiphysics uses by default the implicit second-order accurate method generalized-α to solve transient acoustics problems (see the section "The Time-Dependent Solver" on page 391 of the *COMSOL Multiphysics User's Guide* for more details) and in space the default is 2nd-order 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 thus 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 the automatic time-step control 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. To check that the accuracy is acceptable, it is recommended that you run a short sequence of typical excitations with progressively smaller time steps and check the convergence.

Results and Discussion

Using the properties of air for the medium and selecting a mesh density based on the parameters $h = 0.15$ m and $N = 6$ gives a pulse half-width of $t_0 = 2.624$ ms. To be on the safe side, requiring $\mathrm{CFL} < 0.05$ gives a fixed time step of 0.02 ms. 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.

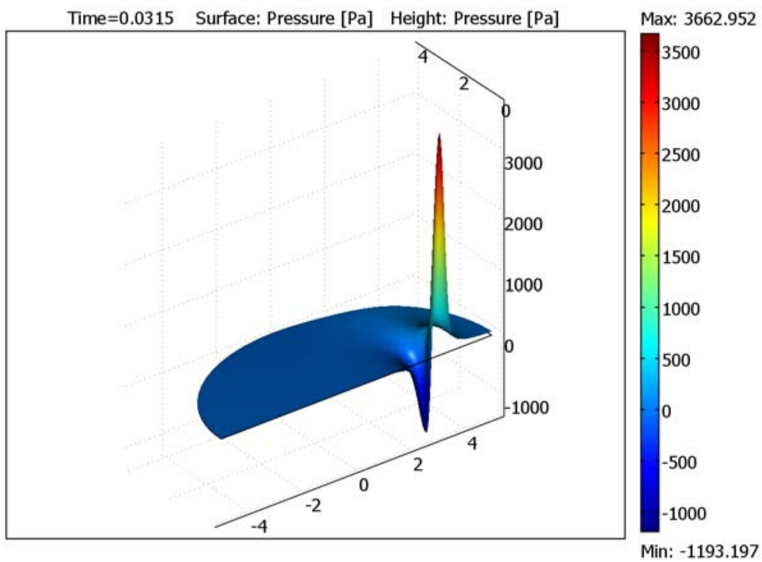


Figure 2-25: The refocusing occurs at roughly 0.0315 s. An animation gives a better feeling for the process.

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, while selecting a higher CFL number (and consequentially a larger time step) leads to an inaccurate solution.

The next figure shows the pressure along the left-hand part of the major axis at $t = 9$ ms—right after the wave is reflected from the apex at $x = -5$ m—for four different CFL numbers: 0.2, 0.1, 0.05, and 0.025. The difference between the last two is small enough to call the solution practically converged.

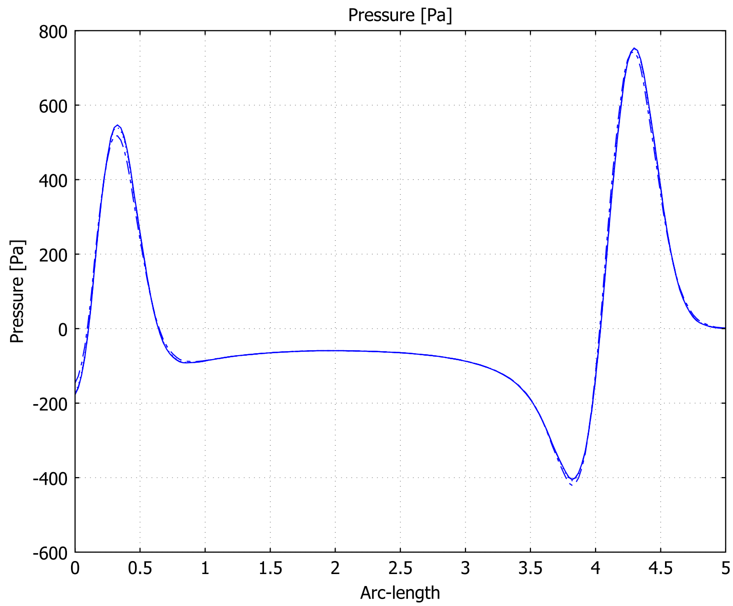


Figure 2-26: The CFL number has a pronounced effect on the accuracy of the final solution. The difference is marked between CFL = 0.2 (dash-dot), CFL = 0.1 (dotted), and CFL = 0.05 (dashed), but essentially indiscernible between the latter time and CFL = 0.025 (solid line).

To get a better view of the graphs in Figure 2-26, go to the COMSOL Help Desk and open the PDF version of the *Acoustics Module Model Library*. At a magnification of 300% the lines are readily distinguishable.

Modeling in COMSOL Multiphysics

You set up this tutorial model using the transient analysis type in the Pressure Acoustics application mode of the Acoustics Module. For material properties and boundary conditions use the default settings, which correspond to an air-filled chamber with sound-hard walls.

The modeling instructions are written for a parameter set that gives a converged and stable solution. You are encouraged to experiment with the resolution number, N, the CFL number, and other parameters to see their effects on the result.

References

1. B. Yue and M.N. Guddati, “Dispersion-reducing finite elements for transient acoustics,” *J. Acoust. Soc. Am.*, vol. 118, no. 4, 2005.

2. H.-O. Kreiss, N.A. Peterson, and J. Yström, “Difference approximations for the second order wave equation,” *SIAM J. Numer. Anal.*, vol. 40, no. 5, 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.

Model Library path: `Acoustics_Module/Tutorial_Models/gaussian_explosion`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **2D** from the **Space dimension** list, then go to the list of application modes and select **Acoustics Module>Pressure Acoustics>Transient analysis**.

2 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1 Shift-click the **Rectangle/Square** button on the Draw toolbar. In the dialog box that appears, change the following settings, then click **OK**.

Size	Width	10
	Height	4
Position	x	-5

2 Zoom out by clicking the **Zoom Extents** button on the Main toolbar.

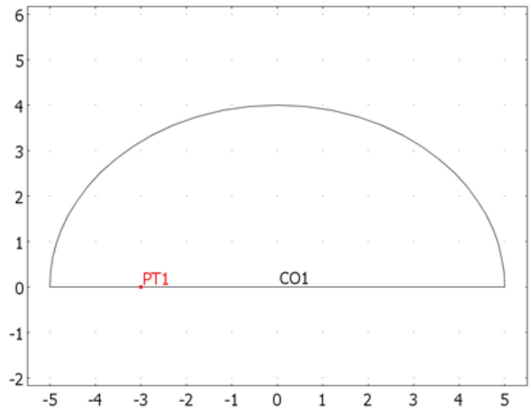
3 Shift-click the **Ellipse/Circle** button on the Draw toolbar. Modify the following entries, then click **OK**.

A-semiaxes	5
B-semiaxes	4

4 Press Ctrl+A to select both objects, then click the **Intersection** button.

5 Click the **Point** button, then draw a point at the coordinates (–3, 0).

The completed geometry in the drawing area should look like that in the figure below.



OPTIONS AND SETTINGS

Choose **Options>Constants** and define the following constants; when finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
c_air	343[m/s]	Speed of sound in air
h_max	0.15[m]	Typical element size
N	6	Points per wavelength
A	4[m^2/s]	Point-source amplitude
f0	c_air/(h_max*N)	Frequency bandwidth
t0	1/f0	Pulse half width
CFL	0.05	CFL number
t_step	CFL*h_max/c_air	Maximum time step

Note the value of roughly $2 \cdot 10^{-5}$ s for the maximum time step; you will use this result at the solving stage.

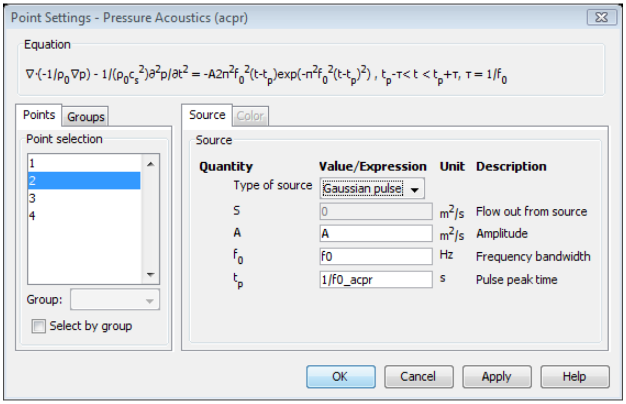
PHYSICS SETTINGS

Point Settings

1 Choose **Physics>Point Settings**.

2 Select Point 2, then select **Gaussian pulse** from the **Type of source** list.

3 In the **A** edit field type `A` and in the $\mathbf{f_0}$ edit field type `f0`.



4 Click **OK**.

GENERATING THE MESH

With the choice of a typical mesh size of $h = 0.15$, 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.

1 Choose **Mesh>Free Mesh Parameters**.

2 Click the **Custom mesh size** option button and type `0.15` in the **Maximum element size** edit field.

3 Click the **Remesh** button, then click **OK**.

COMPUTING THE SOLUTION

1 Choose **Solve>Solver Parameters** or click the corresponding button on the Main toolbar to open the **Solver Parameters** dialog box.

2 Specify the output **Times** in the **Time stepping** frame as `0:0.0005:0.035`.

3 Click the **Time Stepping** tab and set **Time step taken by solver** to **Manual**.

4 Set the time step to the value calculated from the chosen mesh density and the CFL number by typing `2e-5` in the **Time step** edit field. Click **OK** to close the dialog box.

5 Click the **Solve** button on the Main toolbar to start solving.

To compare the effect of different choices of CFL number and resolution parameter, N, do the following:

6 Open the **Constants** dialog box, change N or CFL to some other interesting value, then click **Apply**.

7 If you changed the CFL number, check the new value of `t_step` and transfer it to the **Time step** edit field on the **Time Stepping** page in the **Solver Parameters** dialog box.

8 Solve again by clicking the **Solve** button.

POSTPROCESSING AND VISUALIZATION

The default plot shows the pressure at the final time.

1 For a more attractive plot, click first the **3D Surface Plot** button and then the **Headlight** button on the Plot toolbar.

2 To see the solution close to the refocusing moment, click the **Plot Parameters** button on the Main toolbar, select `0.0315` from the **Solution at time** list, then click **OK**. The result should look like that in Figure 2-25 on page 72.

3 It is illustrative to animate transient problems in general and wave propagation in particular; you can do this by clicking the **Animate** button on the Plot toolbar.

Ultrasound Scattering Off a Cylinder

Introduction

This model, which is inspired by a benchmark problem discussed in Ref. 1, exemplifies ultrasound acoustics modeling with the ultraweak variational formulation (UWVF) in a simple setting. The main advantage of the UWVF for modeling high-frequency acoustic phenomena is its economy with regards to computational complexity compared to low-order finite element analysis. This advantage stems from the property of the UWVF's elements containing information about the solution to the Helmholtz scalar wave equation in free space.

Model Definition

The modeling domain, depicted in Figure 2-27, consists of a circle of radius 5 cm surrounded by a concentric annulus of outer radius 10 cm. In the circle domain, representing an obstacle, the speed of sound is $c_o = 3000$ m/s and the density $\rho_o = 2000$ kg/m^3. Outside the circular obstacle, the speed of sound is $c_s = 1500$ m/s and the density $\rho_o = 1000$ kg/m^3. From a location 1 cm outside the outer boundary of the annulus, a sound source emits cylindrical ultrasound waves of frequency $f_0 = 250$ kHz, corresponding to wavelengths in the two domains of $\lambda_o \equiv c_o/f_0 = 1.2$ cm and $\lambda_s \equiv c_s/f_0 = 0.6$ cm, respectively. On the outer annulus boundary, you impose a radiation boundary condition that absorbs outgoing cylindrical waves. This boundary condition also allows you to model the incoming waves.

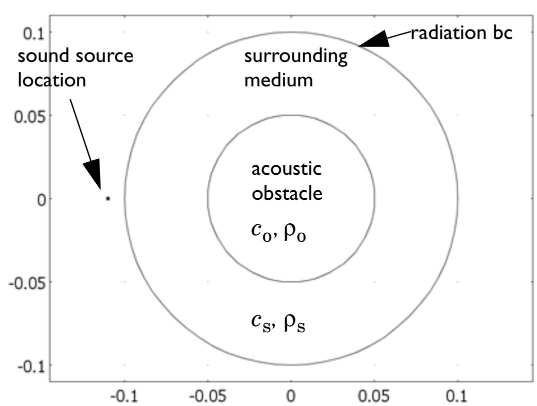


Figure 2-27: Modeling domain.

Results and Discussion

Figure 2-28 shows the acoustic pressure field, p, using the wave colormap. The difference in wavelength between the two subdomains is clearly visible, as is the interference pattern between the incident and scattered waves. Notice also how no artificial distortion of the waves leaving the modeling domain can be discerned.

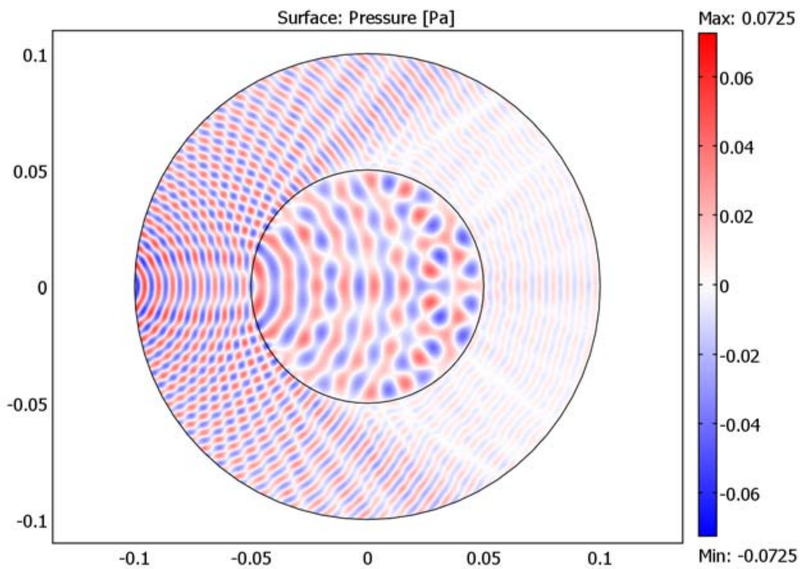


Figure 2-28: The acoustic pressure field. The wavelength inside the obstacle is twice that in the surrounding medium. A radiation boundary condition on the modeling domain's outer boundary both specifies the source and absorbs outgoing waves.

The incoming and scattered waves are most easily distinguishable in the grayscale plots in Figure 2-29 of the pressure field's real (left) and imaginary (right) parts. (Because COMSOL Multiphysics by default plots the real part of a time-harmonic variable such as $p(\mathbf{x}, t) = p(\mathbf{x})e^{i\omega t}$, the plots in Figure 2-28 and the left panel of Figure 2-29 depict the same quantity.

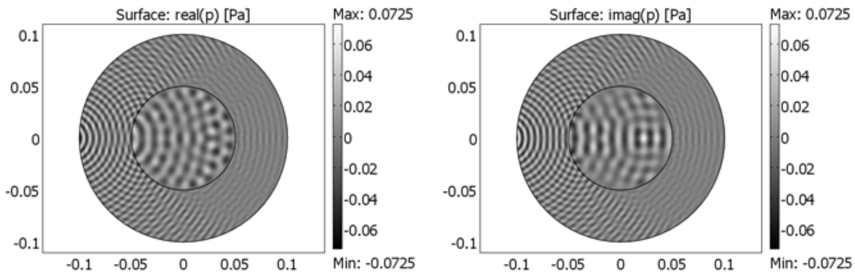


Figure 2-29: Real (left) and imaginary (right) parts of the acoustic pressure field.

You solve this problem using a mesh with 822 elements, shown in Figure 2-30. With the default Ultraweak Helmholtz elements of order 20, the resulting number of DOFs is 16,440. The solution time with the default UMFPACK solver is about 8.6 s with a peak memory usage of 481 MB on a machine running Windows Vista with a 2.6 GHz AMD Athlon X2 Dual Core 500 CPU and 2 GB of RAM. (Using the SPOOLES solver, the solution time is doubled but peak memory usage is reduced to 322 MB.)

Solving the same problem with the same resolution using 2nd-order Lagrange elements requires about 220,000 DOFs (the maximum element size set equal to 1/6th of a wavelength), and takes about five times longer and consumes roughly 75% more memory. In this simple comparison, the UWVF thus shows its strengths.

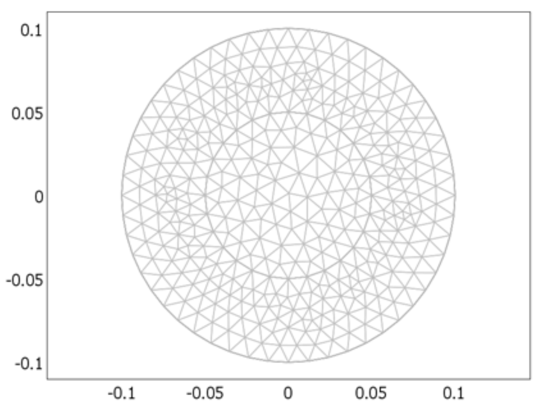


Figure 2-30: The mesh used in the model for computing the solution for the pressure field. The maximum element size in both domains is specified as twice the wavelength.

Reference

1. T. Huttunen, *The ultra weak variational formulation for ultrasound transmission problems*, doctoral dissertation, Kuopio University Publications C, Natural and Environmental Sciences 168, 2004.

Model Library path: `Acoustics_Module/Tutorial_Models/ultrasound_scattering`

Modeling Using the Graphical User Interface

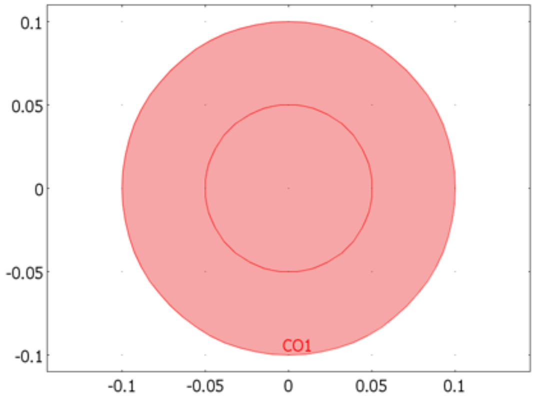
MODEL NAVIGATOR

1. In the **Model Navigator**, select **2D** from the **Space dimension** list.
2. From the list of application modes, select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis with UWVF**.
3. Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

1. Shift-click the **Ellipse/Circle (Centered)** button on the Draw toolbar.
2. Specify a **Radius** of `0.1`, then click **OK**.
3. Click the **Zoom Extents** button on the Main toolbar.
4. Repeat Steps 1–2 to add a second, concentric, circle, of radius 0.05.
5. Click in the drawing area and press Ctrl+A to select both circles.
6. Click the **Create Composite Object** button on the Draw toolbar. In the dialog box that appears, click **OK** to accept the default settings.

The finished geometry in the drawing area should look like that in the following figure.



OPTIONS

Constants

1 From the **Options** menu, select **Constants**.

2 Enter the following names, expressions, and descriptions (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
c_o	3000[m/s]	Speed of sound in obstacle
c_s	1500[m/s]	Speed of sound in surrounding medium
rho_o	2000[kg/m^3]	Density in obstacle
rho_s	1000[kg/m^3]	Density in surrounding medium
f0	250[kHz]	Sound frequency
lda_o	c_o/f0	Wavelength in obstacle
lda_s	c_s/f0	Wavelength in surrounding medium
k_s	2*pi[rad]/lda_s	Wave number in surrounding medium

Note the values of the wavelengths, $\lambda_o = 12$ mm and $\lambda_s = 6$ mm, which are important for determining an appropriate mesh element size.

Scalar Expressions

1 From the **Options** menu, point to **Expressions**, and then select **Scalar Expressions**.

2 Define the following constants; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`R`	`sqrt((x+0.11[m])^2+y^2)`	`Distance from source`
`p_in`	`(i/4)*(besselj(0,k_s*R)-i*bessely(0,k_s*R))`	`Incoming wave`

The point-source expression, `p_in`, representing the incoming wave is proportional to the 0th-order Hankel function of the 2nd kind:

$$p_{\mathrm{in}} = \frac{i}{4}H_0^{(2)}(k_s R) = \frac{i}{4}(J_0(k_s R) - iY_0(k_s R))$$

where $R \equiv \sqrt{(x-x_0)^2 + (y-y_0)^2}$ is the distance from the source location (x_0, y_0).

PHYSICS SETTINGS

Subdomain Settings

1 From the **Physics** menu, choose **Subdomain Settings**.

2 Enter settings according to the following table; when done, click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2
ρ_0	`rho_s`	`rho_o`
c_s	`c_s`	`c_o`

Boundary Conditions

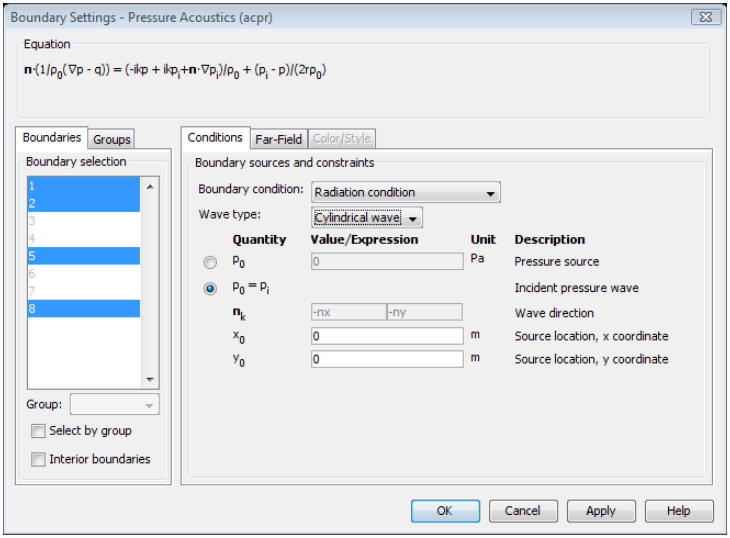
1 From the **Physics** menu, choose **Boundary Settings**.

2 Select Boundaries 1, 2, 5, and 8 (all segments of the outer boundary).

3 From the **Boundary condition** list, select **Radiation condition**.

4 For the outgoing wave, select **Cylindrical wave** from the **Wave type** list.

5 Click the **p_0 = p_i** option button to use the application mode variable **p_in_acpr** for the incoming wave.



6 Click **OK**.

Scalar Variables

1 From the **Physics** menu, choose **Scalar Variables**.

2 Set the excitation frequency by changing the expression for **freq_acpr** to `f0`.

3 Specify the incoming wave by changing the expression for **p_i_acpr** to `p_in`.

4 Click **OK**.

GENERATING THE MESH

To give acceptable accuracy, the Acoustics Module's ultraweak variational formulation needs a relatively uniform mesh, governed by the maximum element size parameter, $h_{\max}$, rather than by sharp geometry details. As a rule of thumb, when using the default UWVF settings, specifying an $h_{\max}$ equal to or slightly larger than 2λ gives a good balance between convergence and accuracy.

Because the wavelength differs between the two subdomains in this model, specify $h_{\max}$ at the subdomain level in the **Free Mesh Parameters** dialog box:

1 From the **Mesh** menu, select **Free Mesh Parameters**.

2 Click the **Subdomain** tab.

3 Specify the maximum element sizes in the two subdomains as follows:

SETTING	SUBDOMAIN 1	SUBDOMAIN 2
Maximum element size	0.012	0.024

4 Click the **Remesh** button.

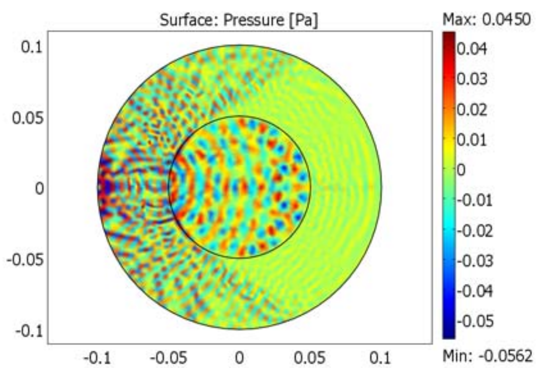
5 Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot, shown in the following figure, displays the acoustic pressure field, p—or, more precisely, the real part of the complex quantity $|p|e^{i\,\arg(p)}$. The coarse appearance is a consequence of the ultraweak variational formulation's high element order. Moreover, the jet colormap is not ideally suited for visualizing the wave pattern.



To modify the settings for element refinement and colormap, follow these steps:

1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page, clear the **Auto** check box for **Element refinement** and type 20 in the associated element field (the order of the default Ultraweak Helmholtz elements).

3 On the **Surface** page, find the **Surface color** area. From the **Colormap** list, select **wave**.

4 Click **Apply**.

The wave colormap is ideally suited for visualizing waves, because of its clear distinction between the low and high ends of the scale. For optimal effect, the range should be symmetric around zero, which then is rendered white. To ensure this feature for the current plot, you need to adjust the range slightly.

5 Click the **Range** button. Clear the **Auto** check box, then set **Min** to `-0.0725` and **Max** to `0.0725` (these limits are sufficient for all phases of the pressure field). Click **OK**.

6 Click **Apply** to generate the plot in Figure 2-28.

Proceed to generate the plots in Figure 2-29 with the following steps:

7 In the **Expression** edit field on the **Surface Data** page, type `real(p)`. Because COMSOL Multiphysics plots the real part of a complex quantity by default, this setting only affects the plot title.

8 Change the **Colormap** to **gray** (or **grayprint**, if you want to print the plot on paper).

9 Click **Apply** to generate the plot in the left panel of Figure 2-29.

10 Change the entry in the **Expression** edit field to `imag(p)`.

11 Click **Apply** to generate the plot in the right panel.

12 Change the **Expression** to `p` and the **Colormap** to **wave**, then click **OK** to close the **Plot Parameters** dialog box with the plot you see when you open the model in the drawing area.

3

Industrial Models

This chapter contains a selection of models from the automotive and aerospace industries as well as a model of a loudspeaker.

Absorptive Muffler

Introduction

This model describes the pressure-wave propagation in a muffler for an internal combustion engine. The approach is generally applicable to analyzing the damping of propagation of harmonic pressure waves.

The model's purpose is to show how to analyze both inductive and resistive damping in pressure acoustics. In addition to a 3D model in two versions—with and without resistive damping—this example includes a 2D analysis of propagation modes in the muffler chamber. It then calculates the transmission loss for the frequency range 50 Hz–1500 Hz.

Model Definition

The muffler—schematically depicted in Figure 3-1—consists of a 24-liter resonator chamber with a section of the centered exhaust pipe included at each end. In the first version of the model the chamber is empty. In the second version it is lined with 15 mm of absorbing glass wool.

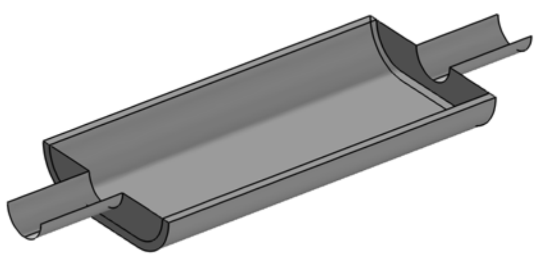


Figure 3-1: Geometry of the lined muffler with the upper half removed. The exhaust fumes enter through the left pipe and exit through the right pipe.

DOMAIN EQUATIONS

This model solves the problem in the frequency domain using the time-harmonic Pressure Acoustics application mode. The model equation is a slightly modified version of the Helmholtz equation for the acoustic pressure, p:

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

where ρ is the density, c_s equals the speed of sound, and ω gives the angular frequency.

In the absorbing glass wool, 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.

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 impedance of air, respectively. You can find flow resistivities in tables. 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 $\rho_{ap} = 12\ \mathrm{kg/m^3}$ and $d_{av} = 10\ \mu\mathrm{m}$.

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:

$$\left(-\frac{\nabla p}{\rho}\right) \cdot \mathbf{n} = 0$$

- The boundary condition at the inlet involves a combination of incoming and outgoing plane waves:

$$\mathbf{n}\cdot\frac{1}{\rho_0}\nabla p + ik\frac{p}{\rho_0} + \frac{i}{2k}\Delta_T p = \left(\frac{i}{2k}\Delta_T p_0 + (1-(\mathbf{k}\cdot\mathbf{n}))ik\frac{p_0}{\rho_0}\right)e^{-ik(\mathbf{k}\cdot\mathbf{r})}$$

In this equation, p_0 represents the applied outer pressure, Δ_T is the boundary tangential Laplace operator, and i equals the imaginary unit (see Ref. 3). This boundary condition is valid as long as the frequency is kept below the cutoff frequency for the second propagating mode in the tube.

- At the outlet boundary, the model specifies an outgoing plane wave:

$$\mathbf{n}\cdot\frac{1}{\rho_0}\nabla p + i\frac{k}{\rho_0}p + \frac{i}{2k}\Delta_T p = 0$$

The wave numbers and mode shapes through a cross section of the chamber are found as the solution of a related eigenvalue problem:

$$\nabla\cdot\left(-\frac{\nabla p(y,z)}{\rho_0}\right) - \left(\frac{\omega^2}{\rho_0 c^2} - \frac{\kappa_x^2}{\rho_0}\right)p(y,z) = 0$$

For a given angular frequency $\omega = 2\pi f$, only modes such that $\kappa_x{}^2$ is positive can propagate. The cutoff frequency of each mode is calculated as

$$f_j = \frac{\sqrt{\omega^2 - c^2\kappa_x^2}}{2\pi}$$

Results and Discussion

The following equation defines the attenuation (in dB) of the acoustic energy, d_w:

$$d_w = 10\log\left(\frac{w_\mathrm{o}}{w_\mathrm{i}}\right)$$

Here w_o and w_i denote the outgoing power at the outlet and the incoming power at the inlet, respectively. You can calculate each of these quantities as an integral over the corresponding surface:

$$w_\mathrm{o} = \int_{\partial\Omega}\frac{|p|^2}{2\rho c_s}dA$$

$$w_i = \int_{\partial\Omega} \frac{p_0^2}{2\rho c_s} dA$$

Figure 3-2 shows the result of a parametric frequency study for the case of an empty muffler without any absorbing material. The plot shows that the damping works rather well for most low frequencies with the exception of a few distinct dips where the muffler chamber displays resonances.

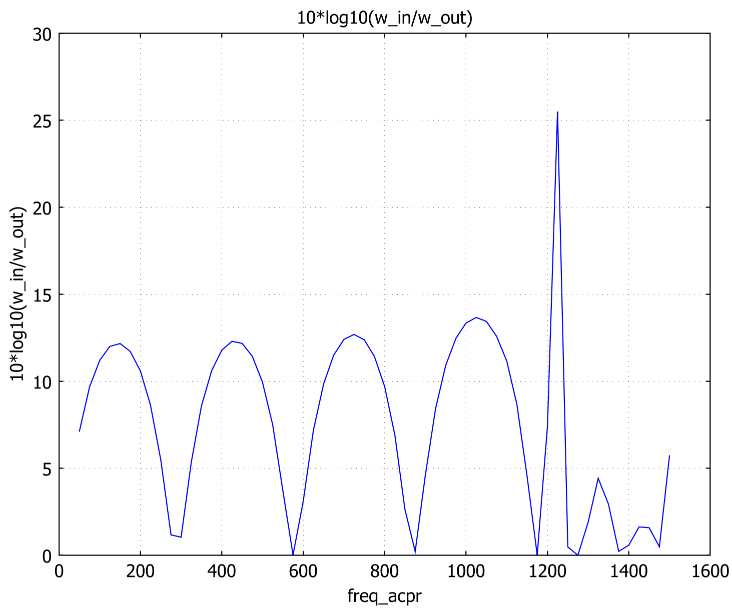


Figure 3-2: Attenuation (dB) in the empty muffler as a function of frequency. The first four dips are due to longitudinal resonances.

At frequencies higher than approximately 1250 Hz, the plot's 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. The first propagation mode that is excited is symmetric with respect to both the y-axis and the z-axis. Figure 3-3 shows this mode, which for an infinitely long chamber occurs at 1239 Hz. 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.

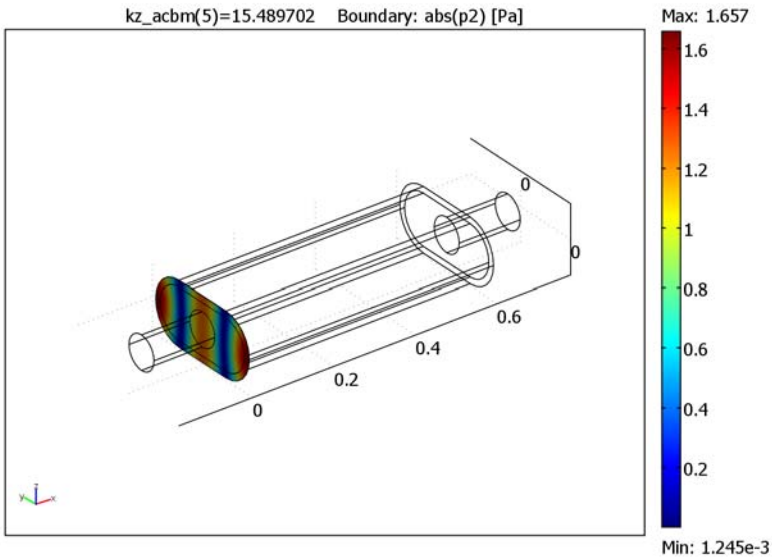


Figure 3-3: The chamber's first symmetric propagation mode. The plot shows the absolute value of the pressure.

The glass-wool lining improves attenuation at higher frequencies. Figure 3-4 shows the attenuation with a layer of lining on the chamber's upper and lower walls.

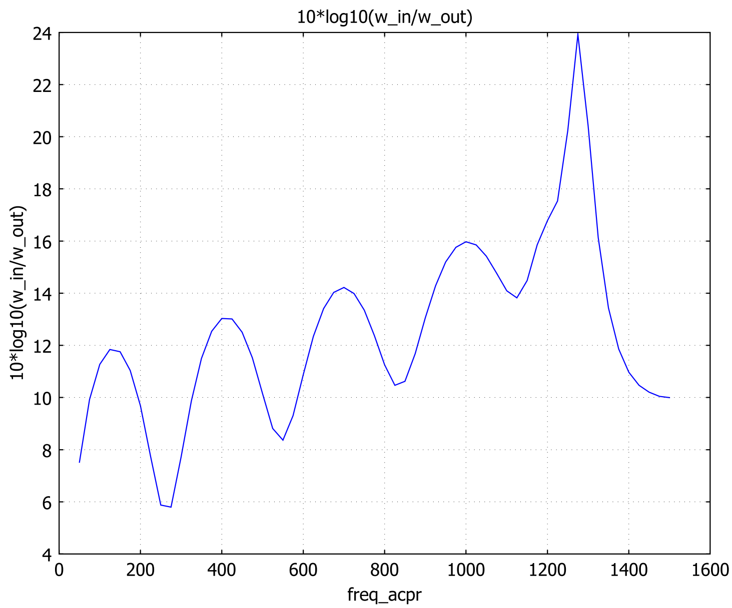


Figure 3-4: Attenuation (dB) in the absorbing muffler as a function of frequency (Hz). The dips are still present, but the general trend is that the higher the frequency, the better the damping.

Modeling in COMSOL Multiphysics

Setting up this model in COMSOL Multiphysics requires the Acoustics Module. You employ two application modes: Pressure Acoustics for the full 3D model, and Boundary Modal Analysis for the propagating mode analysis.

The software has predefined expressions for the Delany-Bazley coefficients, so the only damping parameter you must 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 variables, and it plots the resulting attenuation 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, “Acoustical properties of fibrous absorbent materials,” *Appl. Acoust.*, vol. 14, 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.

Model Library path: `Acoustics_Module/Industrial_Models/absorptive_muffler`

Modeling Using the Graphical User Interface—Rigid Walls

MODEL NAVIGATOR

1 In the **Model Navigator** find the **Space dimension** list and select **3D**.

2 In the list of application modes select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis**.

3 Click **OK** to close the **Model Navigator**.

OPTIONS AND SETTINGS

Choose **Options>Constants** and enter the data in this table (the descriptions are optional); when finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`c_air`	`343[m/s]`	`Speed of sound in air`
`p0`	`1[Pa]`	`Amplitude of incoming pressure wave`

GEOMETRY MODELING

1 Choose **Draw>Work-Plane Settings**. Go to the **Quick** page and select the **y-z** option button, then in the **x** edit field type `0`. Click **OK**.

2 Create four circles with the specifications in the following table. Use the **Circle** dialog box, which you open by shift-clicking the **Ellipse/Circle (Centered)** button on the Draw toolbar; after specifying each cylinder, click **OK**.

RADIUS	BASE CENTER	
	X	Y
0.075	-0.075	0
0.075	0.075	0
0.06	-0.075	0
0.06	0.075	0

3 Shift-click the **Rectangle/Square** button on the Draw toolbar. Create two rectangles with the following specifications; after each rectangle, click **OK**.

WIDTH	HEIGHT	BASE CENTER	
		X	Y
0.15	0.15	0	0
0.15	0.12	0	0

4 Choose **Draw>Create Composite Object**. In the dialog box that appears, enter the **Set formula** as `C1+C2+R1`. Clear the **Keep interior boundaries** check box, then click **Apply** to create a union of these objects.

5 In the same dialog box, enter the **Set formula** as `C3+C4+R2`. Click **OK** to create the union and close the dialog box.

6 Choose **Draw>Extrude**. Select both objects, type `0.6` in the **Distance** edit field, and click **OK**.

7 Shift-click the **Cylinder** button on the Draw toolbar. Specify properties according to the first line in the following table, then click **OK** to create the cylinder.

RADIUS	HEIGHT	AXIS BASE POINT			AXIS DIRECTION VECTOR		
		X	Y	Z	X	Y	Z
0.04	0.15	-0.15	0	0	1	0	0
0.04	0.15	0.6	0	0	1	0	0

8 Repeat the previous step but enter the properties on the second table row.

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

If you select all the objects, the geometry should now look like that in Figure 3-5.

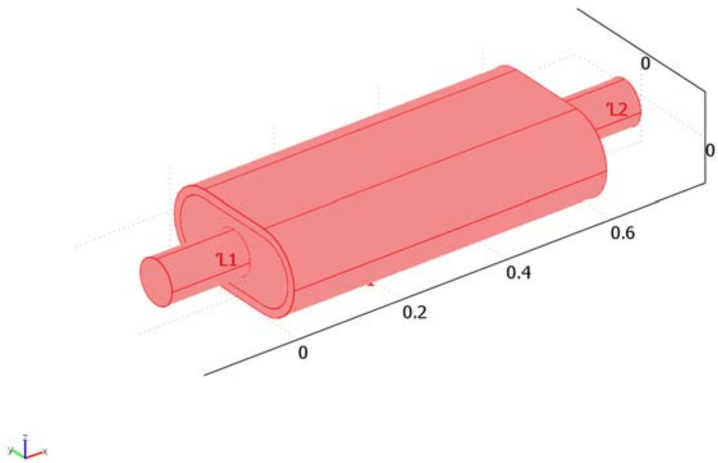


Figure 3-5: The muffler geometry.

PHYSICS SETTINGS

Subdomain Settings

In the second version of this model you insert a lining in Subdomain 2. For now, however, the muffler is completely hollow. Leave all subdomain settings at their default values, which represent air.

Boundary Conditions

1 Choose **Physics>Boundary Settings**.
2 Go to the **Boundary** list and Ctrl-click to select Boundaries 1 and 28.
3 From the **Boundary condition** list select **Radiation condition**. Leave the **Wave type** at the corresponding default selection, which is **Plane wave**.
4 Select Boundary 1 only, and in the $\mathbf{p_0}$ edit field type `p0`.

   This setting results in a pressure source value of p_0 on Boundary 1, and a setting of 0 on Boundary 28. On all other boundaries use the default condition, **Sound hard boundary (wall)**.
5 Click **OK** to confirm the settings and close the dialog box.

Coupling Variables

1 Choose **Options>Integration Coupling Variables>Boundary Variables**.

2 In the **Boundary Integration Variables** dialog box select Boundary 1, then create a boundary integration variable with the **Name** `w_in` and defined by the **Expression** `p0^2/(2*rho_acpr*cs_acpr)`.

3 Select Boundary 28 and create a second boundary integration variable. To do so, add it to the second row of the table with the **Name** `w_out` and define it by the **Expression** `abs(p)^2/(2*rho_acpr*cs_acpr)`.

4 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu open the **Free Mesh Parameters** dialog box.

2 From the **Predefined mesh sizes** list select **Coarse**.

3 Click the **Custom element size** button and type `1.2` in the **Resolution of narrow regions** edit field.

4 On the **Advanced** page go to the **x-direction scale factor** edit field and type `0.25`.

5 Click **OK**.

6 Click the **Initialize Mesh** button on the Main toolbar to generate the mesh.

COMPUTING THE SOLUTION

1 From the **Solve** menu open the **Solver Parameters** dialog box.

2 In the **Solver** list select **Parametric**.

3 In the **Parameter name** edit field type `freq_acpr`, and in the **Parameter values** edit field type `50:25:1500`. This setting computes the solution for frequencies from 50 Hz to 1500 Hz in steps of 25 Hz.

Note: The solution process as set up here takes approximately 10 minutes. If you want to run a faster but less detailed analysis, try a frequency range of 100 Hz to 1500 Hz with a step of 50 Hz; to do so, type `100:50:1500` in the **Parameter values** edit field.

---

4 Click **OK**.

5 Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

The default plot shows the pressure on five equidistant slices of the geometry at 1500 Hz, the last frequency in the frequency sweep. To get a better view of what goes on inside the muffler, you might want to look at other slices.

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box.
2 On the **Slice** page set the **Number of levels** to 1 along all three coordinate directions.
3 Click **Apply** to generate the plot.
4 On the **General** page go to the **Solution to use** area, where you can select which frequency to look at among the entries in the **Parameter value** list. Try, for example, 1250 Hz, which is just where the damping becomes less efficient.
5 Click the **Slice** tab and type `abs(p)` in the **Expression** edit field.
6 Click **Apply** to generate the new plot.

   As you can see, the pressure field varies primarily with the y-coordinate, while it is nearly constant in the z direction. The reason is that 1250 Hz is just higher than the cutoff frequency for the first symmetric propagating mode, which is excited by the incoming wave. You can get an indication of the mode's shape by looking at a cross section of the chamber:
7 Click the **Go to YZ View** button on the Camera toolbar to the left of the drawing area. Take a quick glance at Figure 3-3, which shows the true shape of the corresponding eigenmode for an infinite tube with the same cross section as the chamber; this should resemble what you have. In the last exercise for this model you reproduce the eigenmode calculation.

As an alternative way of visualizing the pressure, try an isosurface plot.

1 Click the **Go to Default 3D View** button on the Camera toolbar.
2 In the **Plot Parameters** dialog box, go to the **General** page. Clear the **Slice** check box and select the **Isosurface** check box.
3 On the **Isosurface** page, enter 15 for the **Number of levels**.
4 On the **General** page, select **1400** from the **Parameter value** list.
5 Click **OK** to close the dialog box and generate the plot.

6 Click the **Scene Light** button on the Camera toolbar to make the pressure field look even nicer.

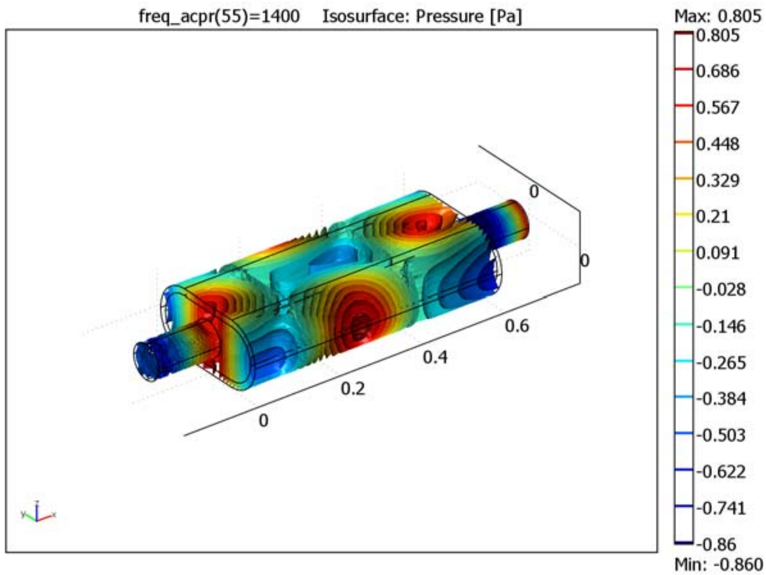


Figure 3-6: The pressure field in the muffler for a frequency of 1400 Hz.

To study the attenuation as a function of frequency, plot some global variables.

1 Choose **Postprocessing>Global Variables Plot**.

2 In the **Expression** edit field type `10*log10(w_in/w_out)`. Click the **>** button next to the edit field to copy your expression to the **Quantities to plot** list.

3 Click **OK** to see the plot, which should look like the one in Figure 3-2.

Absorptive Muffler—Absorbing Walls

In this, the second version of the model, you line the muffler with a layer of absorptive glass wool. Continue working from where you left off with the model developed thus far and make the following changes and additions.

OPTIONS AND SETTINGS

1 Choose **Options>Constants**. In the resulting dialog box, retain the existing constants and add new ones according to the following table; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
rho_ap	12	Apparent density of glass wool (kg/m^3)
d_av	10e-6[m]	Mean fiber diameter
R_f	3.18e-9[N*s/m^4]*rho_ap^1.53/d_av^2	Flow resistivity (Ns/m^4)

PHYSICS SETTINGS

Subdomain Settings

Choose **Physics>Subdomain Settings**. In the resulting dialog box select Subdomain 2 and change its settings according to the following table; when done, click **OK**.

SETTINGS	SUBDOMAIN 2
Type of damping	Delany-Bazley
R_f	R_f

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

Looking at the pressure field at a selection of frequencies, it is clear that the damping is now better than before, especially at higher frequencies. To see the attenuation and reproduce Figure 3-4, return to the **Global Variables Plot** dialog box and again plot `10*log10(w_in/w_out)`.

Absorptive Muffler—Propagating Mode Analysis

As an additional exercise, use a 2D perpendicular boundary mode analysis to calculate the chamber's propagating eigenmodes.

Model Library path: `Acoustics_Module/Industrial_Models/muffler_eigenmodes`

MODEL SETTINGS

1 Choose **Multiphysics>Model Navigator**.
2 From the list of application modes select **Acoustics Module>Pressure Acoustics>Boundary modal analysis**, then click **Add**.
3 From the **Ruling application mode** list select **Pressure Acoustics, Boundary Modal Analysis (acbm)**.
4 Click **OK**.

PHYSICS SETTINGS

Boundary Conditions

The model performs a boundary-mode analysis on a cross section of the muffler, so you should deactivate it elsewhere. Keep the default boundary conditions for air and the wall-edge conditions.

1 Choose **Physics>Boundary Settings**.
2 Select all boundaries and clear the **Active in this domain** check box.
3 Select Boundaries 6, 9, and 16, then select the **Active in this domain** check box.
4 Click **OK** to confirm and close the dialog box.

Scalar Variables

1 Choose **Physics>Scalar Variables** and set the value of **freq_acbm** to `1500`. This means that the software looks for propagating modes with cutoff frequencies as high as 1500 Hz.
2 Click **OK** to close the **Application Scalar Variables** dialog box.

COMPUTING THE SOLUTION

1 Open the **Solver Manager** dialog box either by clicking the corresponding button on the Main toolbar or from the **Solve** menu.
2 On the **Solve For** page instruct the software to solve only for **p2**, then click **OK**.
3 Open the **Solver Parameters** dialog box.
4 From the **Solver** list select **Eigenvalue**.
5 On the **General** page go to the **Propagation constant** area. In the **Desired number of propagation constants** edit field type `8`, and in the **Search for propagation constants around** edit field type `20`.
6 Click **OK**.
7 Click the **Solve** button on the Main toolbar.

You should expect to find the free-space propagation mode with a propagation constant equal to $\omega/c_s = 27.5$ rad/m, along with all other propagating modes—the higher the mode, the lower the propagation constant. In this case there are five propagating modes all together. However, the solver does not stop at zero but instead moves on to find three damped modes with imaginary propagation constants.

POSTPROCESSING AND VISUALIZATION

Visualize the boundary eigenmodes as follows:

1 Open the **Plot Parameters** dialog box.
2 On the **General** page clear the **Isosurface** check box and select the **Boundary** check box.
3 Click the **Boundary** tab and type `p2` in the **Expression** edit field.
4 Return to the **General** page and study the propagating modes by selecting, in turn, the real-valued entries in the **Propagation constant** list, then clicking **Apply** to generate each plot.

Next calculate the cutoff frequencies for the five propagating modes.

1 Choose **Postprocessing>Data Display>Global**. In the **Expression** edit field type `sqrt(omega_acbm^2-kz_acbm^2*c_air^2)/(2*pi)`.
2 Select a real-valued entry in the **Propagation constant** list, then click **Apply** to evaluate the corresponding cutoff frequency.

   The following is a list of the cutoff frequencies that you should obtain when repeating the last step for all five propagating modes, together with their symmetry properties:

CUTOFF FREQUENCY (HZ)	CHARACTERISTICS
0	Plane wave
635	Antisymmetric with respect to y, symmetric with respect to z
1209	Symmetric with respect to y, antisymmetric with respect to z
1239	Symmetric with respect to y and z
1466	Antisymmetric with respect to y and z

Finally, to reproduce the plot in Figure 3-3 for the symmetric propagating mode, follow these steps:

1 In the **Plot Parameters** dialog box go to the **General** page and select the symmetric mode (propagation constant equal to $15.5\ \mathrm{m}^{-1}$).
2 Click the **Boundary** tab and type `abs(p2)` in the **Expression** edit field.
3 Click **OK** to generate the plot.

The mode shape you see should resemble the profile of the propagating-wave solution at 1250 Hz derived in the time-harmonic 3D analysis of the model.

Car Interior

Introduction

In this model, a point source generates a pressure wave in the Sound Brick, a test bench car interior (Ref. 1). The sound 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 level is studied using a couple of different resolutions.

Model Definition

The geometrical and material parameters in this model come from Ref. 2. For the dimensions of the Sound Brick see Figure 3-7.

For the harmonic sound waves of acoustic pressure, $p(\mathbf{x}, t) = p(\mathbf{x})e^{i\omega t}$, that you study in this model, the following frequency-domain Helmholtz equation applies for $p(\mathbf{x})$:

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

Here ρ_0 is the density (kg/m^3), $\omega = 2\pi f$ denotes the angular frequency (rad/s), c_s refers to the speed of sound (m/s), and Q (1/s^2) is a monopole source.

A point source flow of strength $S = 10^{-5}$ m^3/s located at the point $\mathbf{R}_0 = (0.21,\ 0,\ 1.28)$ drives the system, so that

$$Q = \omega S\,\delta^{(3)}(\mathbf{R} - \mathbf{R}_0)$$

where $\delta^{(3)}(\mathbf{R})$ is the 3D Dirac delta function. Assume, furthermore, that the walls of the Sound Brick are perfectly reflecting.

The sound 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.

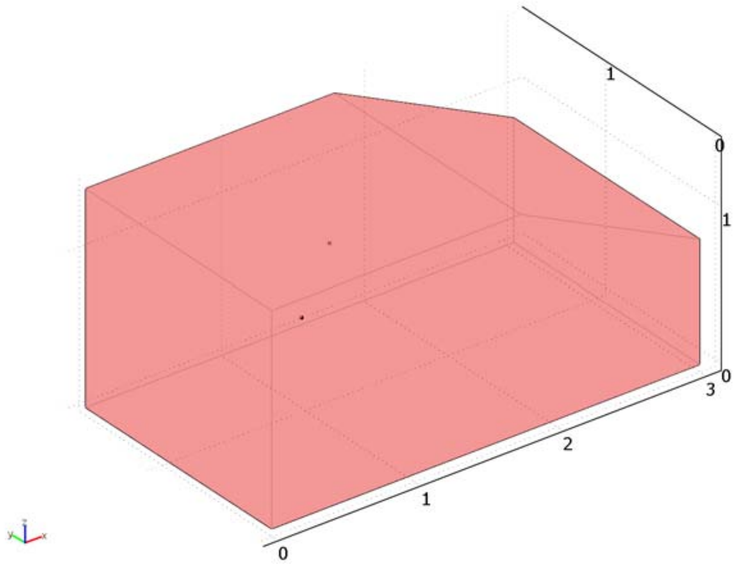


Figure 3-7: The geometry of the car interior. The dimensions are length × height × depth = 3.0 m × 1.4 m × 1.7 m. The windshield has its lower end 0.8 m above the floor and with an inclination such that the entire volume of the geometry is 6.5 m^3.

Results and Discussion

Figure 3-8 shows the sound pressure distribution in the car interior at 500 Hz.

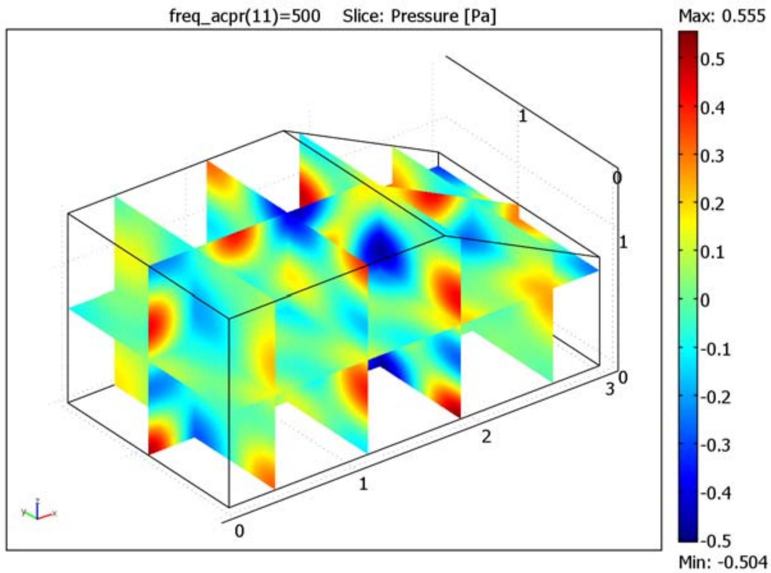


Figure 3-8: Slice plot of the sound pressure distribution at 500 Hz.

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.

Table 3-1 shows how the number of wavelengths per element (first column) and the resulting number of degrees of freedom (second column) affect the memory usage (third column) and the time needed to solve for each frequency (fourth column). The fifth column indicates the line style used in Figure 3-9. The numbers were measured while solving on a workstation running Windows Vista with a 2.6 GHz AMD Athlon X2 Dual Core 500 CPU and 2 GB of RAM.

TABLE 3-1: MEMORY AND TIME USAGE FOR DIFFERENT MESH RESOLUTIONS

ELEMENTS PER WAVELENGTH	DOFS	MEMORY (MB)	SOLUTION TIME PER FREQUENCY (S)	LINE STYLE
5.6	85,733	300	5	Dotted
6.4	125,413	360	8	Dash-dot
7.2	179,335	520	12	Dashed
8.0	248,791	720	20	Solid

To provide an idea of the accuracy of the various solutions, Figure 3-9 shows the measured sound level between 490 Hz and 500 Hz, computed at 0.1 Hz intervals for the listed mesh resolutions. 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.

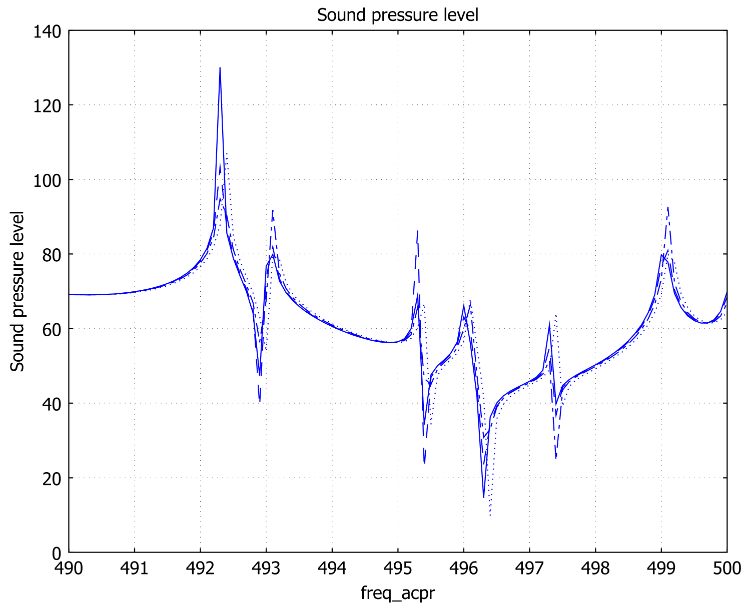


Figure 3-9: Sound pressure level (dB) at the point of measurement as a function of frequency for a few different mesh densities. For a legend see Table 3-1.

Modeling in COMSOL Multiphysics

You set up the Sound Brick model using the Acoustics Module's Pressure Acoustics application mode. The GMRES solver with the Geometric Multigrid preconditioner ensures low memory consumption at a high mesh resolution.

The preconditioner operates on two meshes:

- An initial mesh that you specify, on which a coarse approximation to the solution is computed.
- An automatically generated mesh, finer than the initial one, on which the final solution is obtained.

In the case treated in the step-by-step instructions, you generate the initial mesh by specifying a maximum element size of $L/3.2$, where $L = c_s/f$ is the free-space wavelength of the sound waves at 500 Hz. The maximum element size that the preconditioner can handle for 2nd-order Lagrange elements is just less than $L/2$, as given by the Nyquist criterion. However, using such a coarse mesh would give an ill-conditioned problem, resulting in higher memory usage and a longer solution time.

During the solution procedure, COMSOL Multiphysics automatically generates a finer mesh from the coarse one using a regular refinement method where every element is split into eight identical smaller elements. The maximum element size is hence smaller by a factor of two compared to the original mesh. It follows that the rule-of-thumb minimum of ten to twelve degrees of freedom per wavelength for the solution to be reliable is fulfilled.

For a detailed description of the geometric multigrid technique and its use in modeling, see the section "The Geometric Multigrid Solver/Preconditioner" on page 555 of the *COMSOL Multiphysics Reference Guide*.

References

1. The Sound Brick is located in Acoustic Competence Centre, Inffeldgasse 25, A-8010 Graz, Austria. Web site: http://www.accgraz.com

2. Hepberger and others, "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 2002, 887-884.

Model Library path: `Acoustics_Module/Industrial_Models/car_interior`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator** select **3D** from the **Space dimension** list.
2. From the list of application modes select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis**.
3. Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

4. Choose **Draw>Work-Plane Settings**. On the **Quick** tab select **z-x y:** `0`, then click **OK**.

5 Choose **Draw>Specify Objects>Line**. In the dialog box that appears, make the following entries; when done, click **OK**.

Coordinates x	`0 1.4 1.4 0.8 0`
Coordinates y	`0 0 1.7451 3 3`
Style	Closed polyline (solid)

6 Choose **Draw>Extrude**. Make sure the object you just created (CO1) is selected, then type `1.7` in the **Distance** edit field. Click **OK** to extrude CO1 and close the dialog box.

7 Choose **Draw>Point**. In the dialog box that appears, make the following entries:

Coordinates x	`0.21 1.34`
Coordinates y	`0 1.22`
Coordinates z	`1.28 0.8`

8 Click **OK** to close the dialog box.

OPTIONS AND SETTINGS

Choose **Options>Constants** and define the following constants (the descriptions are optional). When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`rho`	`1.2[kg/m^3]`	`Air density`
`cs`	`343.8[m/s]`	`Speed of sound`
`S`	`1e-5[m^3/s]`	`Flow source strength`

PHYSICS SETTINGS

Subdomain Settings

1 Choose **Physics>Subdomain Settings**.

2 Select Subdomain 1, then in the **Fluid density** edit field type `rho` and in the **Speed of sound** edit field type `cs`.

3 Click **OK** to close the dialog box.

Boundary Conditions

Use the default **Sound hard boundary (wall)** condition for all boundaries.

Point Settings

1 Choose **Physics>Point Settings**.

2 Select Point 5, then type `S` in the **Flow out from source** edit field.

3 Click **OK** to close the dialog box.

GENERATING THE MESH

1 From the **Mesh** menu open the **Free Mesh Parameters** dialog box.

2 On the **Global** page, select **Custom mesh size** and type `343.8/500/3.2` in the **Maximum element size** edit field.

3 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

1 Choose **Solve>Solver Parameters** or click the corresponding button on the Main toolbar to open the **Solver Parameters** dialog box.

2 From the **Solver** list, select **Parametric**.

3 In the **Parameter name** edit field type `freq_acpr`, and in the **Parameter values** edit field type `linspace(490,500,11)`.

4 Select **GMRES** from the **Linear system solver** list, then select **Geometric multigrid** from the **Preconditioner** list.

5 Click the **Settings** button. In the dialog box that appears select **Preconditioner** from the **Linear system solver** tree, and then select **Refine mesh** from the **Hierarchy generation method** list.

   These settings implement the solution procedure described in the subsection "Modeling in COMSOL Multiphysics" on page 108. In particular, the choice of hierarchy-generation method ensures that the automatically generated mesh satisfies the Nyquist criterion.

6 Click **OK** twice to close first the **Linear System Solver Settings** dialog box and then the **Solver Parameters** dialog box.

7 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the acoustic pressure on five equidistant slices along the x-axis. As a first postprocessing step, generate a couple of more informative plots.

1 Choose **Postprocessing>Plot Parameters** or click the corresponding button on the Main toolbar.

2 On the **Slice** page find the **Number of levels** edit fields. Enter `4`, `1`, and `1` for the **x levels**, **y levels**, and **z levels**, respectively.

3 Click **Apply** to see the plot, which should resemble the one in Figure 3-8.

To get an idea of the location of the resonances, try an isosurface plot of the sound pressure level.

4 Return to the **Plot Parameters** dialog box and click the **General** tab.

5 Clear the **Slice** check box and select the **Isosurface** check box.

6 Clear the **Element refinement: Auto** check box and type `1` in the corresponding edit field. At the cost of a less refined plot, this setting is done to avoid running out of graphics memory.

7 On the **Isosurface** page select **Sound pressure level** from the **Predefined quantities** list.

8 Select the **Vector with isolevels** option button and type `50 60 70 80` in the corresponding edit field.

9 Click **OK** to generate the plot, then click the **Headlight** button on the Camera toolbar to get a clearer view. The plot should resemble that in Figure 3-10.

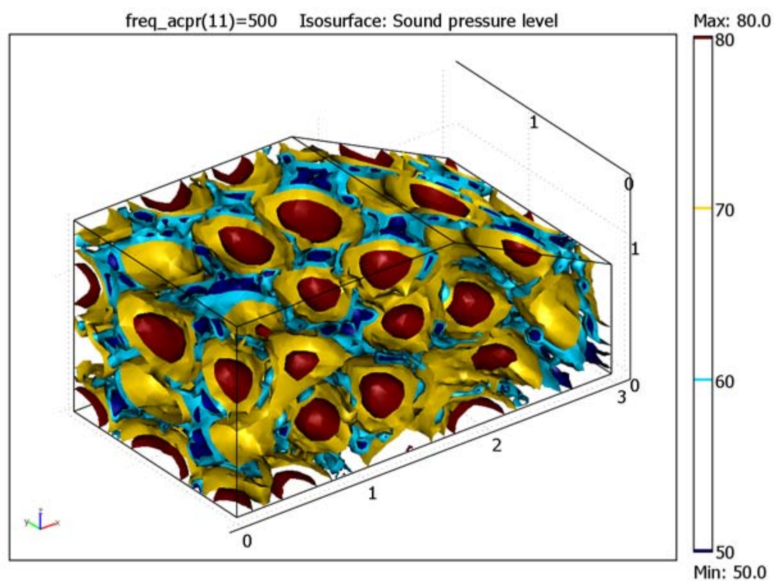


Figure 3-10: Isosurface plot of the sound pressure level in dB at 500 Hz.

Finally, display the sound pressure level at the point of measurement as a function of frequency:

10 Choose **Postprocessing>Domain Plot Parameters**.

11 On the **General** page, select the **Keep current plot** check box.

12 On the **Point** page, select Point 6. In the **Expression** field, type `Lp_acpr`.

13 Click **OK** to close the dialog box and generate the plot.

If you want to compare the results from this solution with a finer mesh resolution, take the following steps:

1 Choose **Mesh>Mesh Cases**.

2 From the **Mesh cases** list select **1**, then click the **Delete** button.

3 Click **OK** to close the dialog box.

4 Choose **Mesh>Free Mesh Parameters**.

5 Enter a different value for the **Maximum element size**. For example, `343.8/500/3.6` gives 3.6 elements per wavelength in the coarse mesh and 7.2 elements per wavelength in the solution mesh. Make sure that you check with Table 3-1 for an estimate of the time and memory consumption before solving.

6 Click the **Remesh** button, then click **OK**.

7 In the **Solver Parameters** dialog box, click the **Settings** button.

8 In the **Linear System Solver Settings** dialog box, select **Preconditioner** from the **Linear system solver** tree to the left. From the **Hierarchy generation method** list, select **Refine mesh**. Click **OK** twice to close the dialog boxes.

9 Click the **Solve** button.

10 Open the **Domain Plot Parameters** dialog box.

11 On the **Point** page, click the **Line Settings** button. Select a different **Line color**, **Line style**, or **Line marker** to separate the new solution from the previous one. Click **OK** to close the **Line Settings** dialog box.

12 Click **OK**.

A new plot should now appear in the same figure as the old one.

13 Start over with another mesh resolution if desired.

Figure 3-11 shows the measured sound pressure level as a function of frequency for a four different meshes and with a frequency pitch of 1 Hz. To reproduce Figure 3-9 with a pitch of 0.1 Hz, open the **Solver Parameters** dialog box and edit the entry in the **Parameter values** edit field to read `linspace(490,500,101)` before going through the solution procedure again. Note, however, that such a detailed frequency sweep takes a few hours of computing time even on a comparatively powerful workstation.

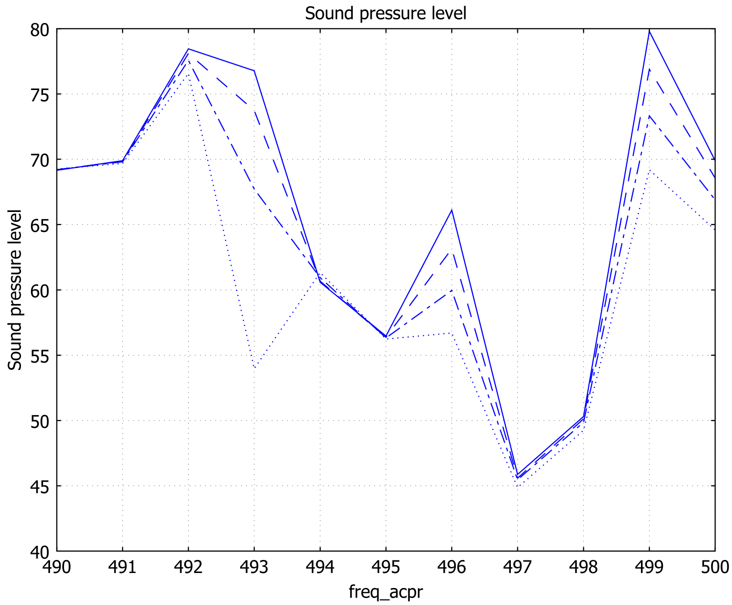


Figure 3-11: The measured sound pressure level plotted against the frequency with a 1 Hz pitch and for a few different mesh resolutions. With this pitch, the resonances are not sufficiently resolved for the plot to give a fair representation of the convergence. However, it is clear that the difference between the solutions decreases as you increase the resolution.

LINE STYLE	ELEMENTS PER WAVELENGTH
Dotted	5.6
Dash-dot	6.4
Dashed	7.2
Solid	8

Flow Duct

Introduction

The modeling of aircraft-engine noise is a central problem in the field of computational aeroacoustics. 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 terms of variables made dimensionless by division by suitable combinations of a reference duct radius, R_∞, a reference speed of sound, c_∞, and a reference density, ρ_∞, it is described by

$$\frac{\partial\tilde{\rho}}{\partial t} + \nabla\cdot(\tilde{\rho}\tilde{\mathbf{v}}) = 0$$

$$\tilde{\rho}\left(\frac{\partial\tilde{\mathbf{v}}}{\partial t} + \tilde{\mathbf{v}}\cdot\nabla\tilde{\mathbf{v}}\right) + \nabla\tilde{p} = 0$$

$$\gamma\tilde{p} = \tilde{\rho}^{\gamma} \qquad\qquad \tilde{c}^2 = \frac{d\tilde{p}}{d\tilde{\rho}} = \tilde{\rho}^{\gamma-1}$$

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.

Because the flow is 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:

$$\tilde{\phi} = \Phi + \phi e^{i\omega t} \qquad \tilde{\mathbf{v}} = \mathbf{V} + \mathbf{v}e^{i\omega t} \qquad \tilde{\rho} = \rho + \rho_a e^{i\omega t}$$

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, and it extends the mean-flow + acoustic split to the pressure:

$$\tilde{p} = P + pe^{i\omega t}$$

In particular, the linearized equations for the acoustic variables are

$$i\omega\rho_a + \nabla \cdot (-\rho\nabla\phi + \rho_a V) = 0$$
$$\rho(i\omega\phi + \mathbf{V} \cdot \nabla\phi) = p$$
$$p = C^2\rho_a$$

The duct geometry used in this model, depicted in Figure 3-12, is taken from Ref. 1. It is an approximate model of the inlet section of a turbofan engine in the very common CFM56 series.

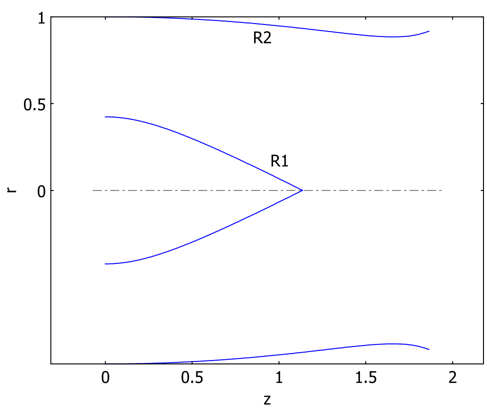


Figure 3-12: The duct geometry.

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 non-reflecting 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.

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

$$i\omega(\mathbf{v}\cdot\mathbf{n}) = [i\omega + \mathbf{V}\cdot\nabla - (\mathbf{n}\cdot(\mathbf{n}\cdot\nabla\mathbf{V}))]\left(\frac{p}{Z}\right)$$

  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 Aeroacoustics application mode of the COMSOL Multiphysics Acoustics Module. 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 3-13. Note, in particular, 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.

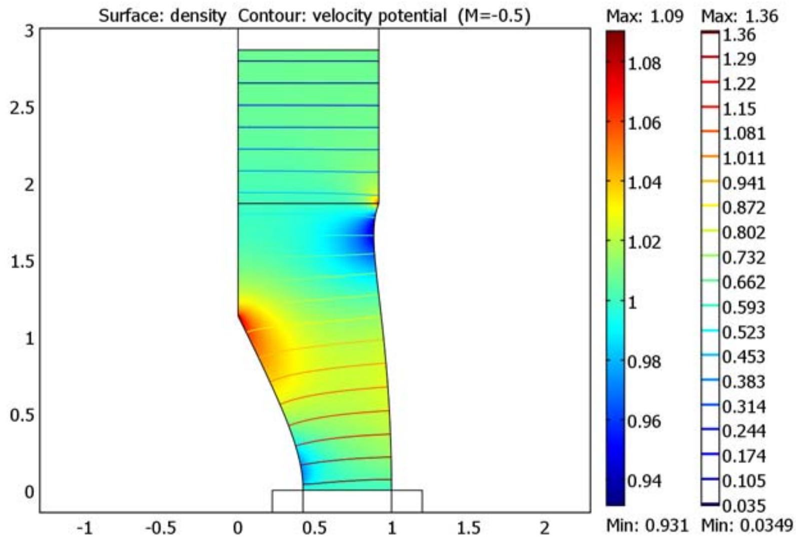


Figure 3-13: Mean-flow velocity potential and density for source-plane Mach number $M = -0.5$*.*

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 3-14.

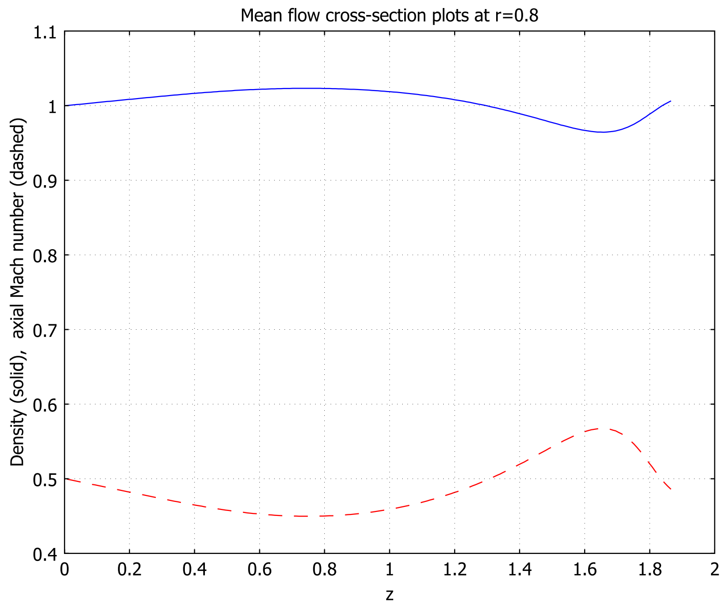


Figure 3-14: Mean-flow cross-section plot at a sample radius of 0.8.

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 3-15 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 $M = -0.5$.

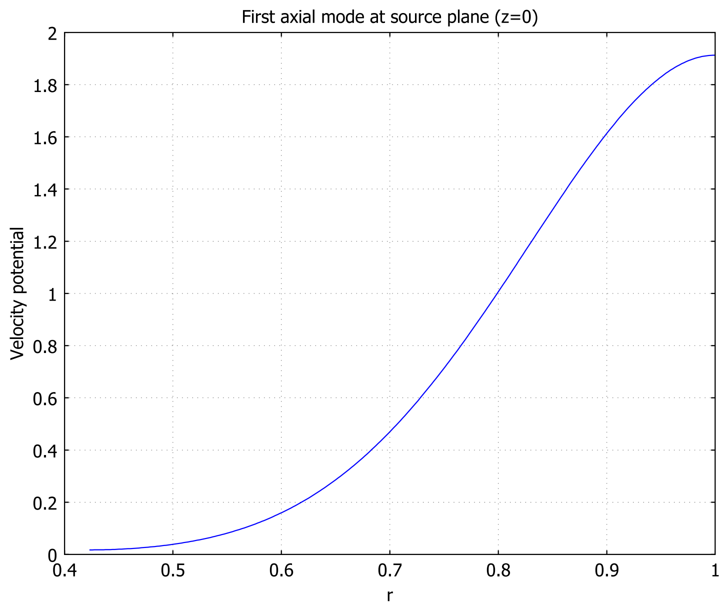


Figure 3-15: The first axial boundary mode at the source plane (z = 0) for the case of a background flow with Mach number $M = -0.5$.

THE AEROACOUSTIC FIELD

The pressure fields for the case without a background mean flow, depicted in Figure 3-16, very closely match those for the corresponding 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: 50.3 dB for the COMSOL Multiphysics solution versus 51.6 dB for the FEM solution in Ref. 1.

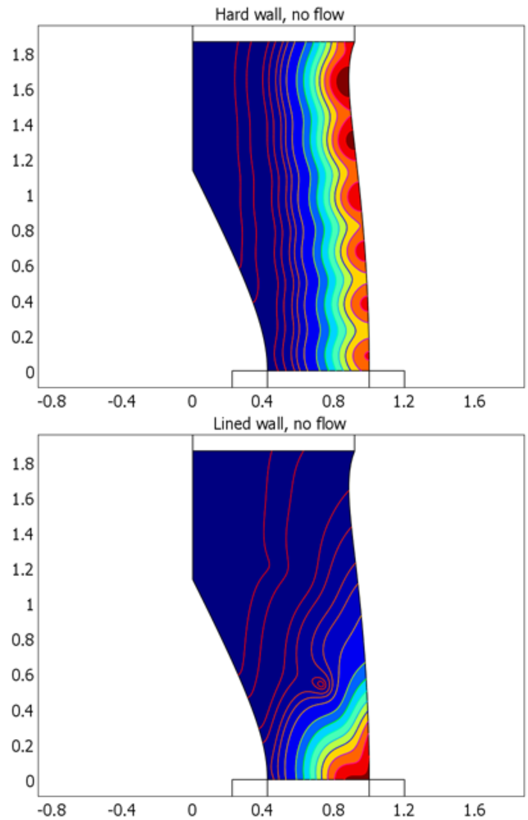


Figure 3-16: Acoustic pressure field for the cases of hard (top) and lined (bottom) duct wall with no mean flow and at circumferential mode number m = 10 and angular frequency $\omega = 16$.

Turning to the case with a mean flow, the pressure field for the hard-wall case in the upper image of Figure 3-17 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 22.0 dB differs significantly from the value of 27.2 dB obtained in Ref. 1.

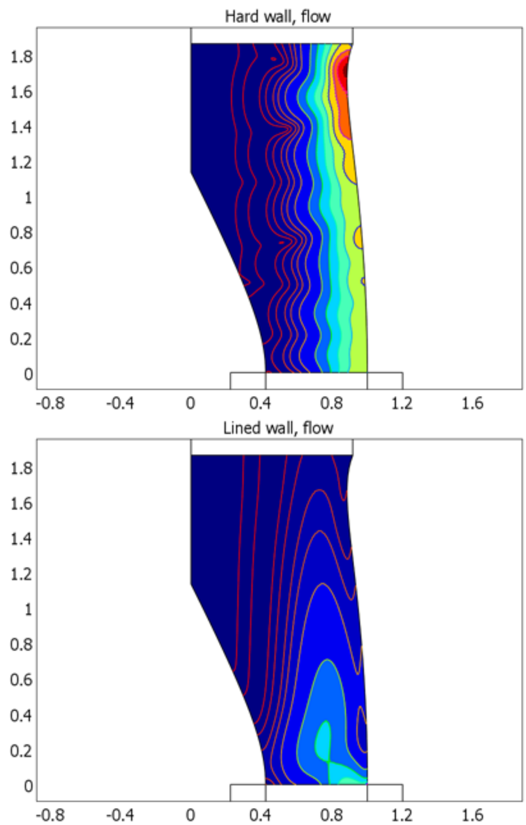


Figure 3-17: Acoustic pressure distribution for the cases of hard (top) and lined (bottom) duct wall with mean flow ($M = -0.5$) and at circumferential mode number $m = 10$ and angular frequency $\omega = 16$.

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 in the lower half of Figure 3-17 is not optimally adapted to the duct, and it is consequently not maximally attenuated.

Modeling in COMSOL Multiphysics

The model involves three application modes, the last of which is used twice:

- *Compressible Potential Flow* (acpf)—for modeling the mean-flow velocity field
- *Aeroacoustics, Boundary modal analysis* (acab)—for calculating the boundary eigenmode to be used as the source of the acoustic noise in the mean-flow background
- *Aeroacoustics, Time-harmonic analysis* (acae, acae2)—for modeling the acoustic field above and below the source plane

After an initial modeling stage—consisting of creating the geometry and the mesh, then defining expressions and variables—you proceed by solving the model of the aero-engine duct in a mean-flow background in three steps. Thereafter you repeat part of the procedure in order to derive the corresponding solution in the absence of a background flow.

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.

Model Library path: `Acoustics_Module/Industrial_Models/flow_duct`

Initial Stage—Geometry, Mesh, and Common Settings

MODEL NAVIGATOR

1. In the **Model Navigator** go to the **Space dimension** list and select **Axial symmetry (2D)**, then click **Multiphysics**.
2. From list of application modes select **Acoustics Module>Aeroacoustics>Boundary modal analysis**. In the **Dependent variables** edit field type `phi_b`, then click **Add**.
3. From the list of application modes select **Acoustics Module>Aeroacoustics with flow**, then click **Add**.
4. From the list of application modes select **Acoustics Module>Aeroacoustics>Time-harmonic analysis**. In the **Dependent variables** edit field type `phi2`, then click **Add**.
5. Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

First import the duct geometry, which is supplied in the form of an `mphbin` file:

1. Choose **File>Import>CAD Data From File**, browse to the folder `Acoustics Module/ Industrial Models`, select the file `flow_duct.mphbin`, and click **Import**.

   Next attach rectangles at the inlet and outlet. These are not part of the duct but allow you to implement the appropriate boundary conditions using perfectly matched layers (PMLs).
2. If the **SNAP** field on the Status bar at the bottom of the workspace is not already selected, double-click to select it.
3. Choose **Draw>Draw Objects>Rectangle/Square** or click the corresponding button at the top of the Draw toolbar. In the drawing area click at the top right corner of the duct and drag to the point ($r = 0$, $z = 3$).
4. With the rectangle (R1) that you just created still selected, choose **Draw>Object Properties** to open the **Rectangle** dialog box. In the **Height** edit field type `0.2`, then click **OK**.
5. Click the **Rectangle/Square** button on the Draw toolbar. In the drawing area click at the bottom left corner of the duct and drag to the point ($r = 1$, $z = -0.5$).

6 With the rectangle (R2) that you just created still selected, choose **Draw>Object Properties**. In the **Height** edit field type `0.2`, and in the **z** edit field type `-0.2`. Click **OK**.

7 Click the **Rectangle/Square** button on the Draw toolbar. In the drawing area click at the bottom left corner of rectangle R2 and drag to the origin ($r = 0$, $z = 0$).

8 With the rectangle (R3) that you just created still selected choose **Draw>Object Properties**. In the **Width** edit field type `0.2`, and in the **r** edit field type `0.223557`. Click **OK**.

9 Click the **Rectangle/Square** button on the Draw toolbar. In the drawing area click at the bottom right corner of rectangle R2 and drag to the point ($r = 1.5$, $z = 0$).

10 With the rectangle (R4) that you just created still selected, choose **Draw>Object Properties**. In the **Width** edit field type `0.2`. Click **OK**.

11 Click the **Zoom Extents** button on the Main toolbar to get a proper overview of the duct geometry, which should look like the one in Figure 3-18.

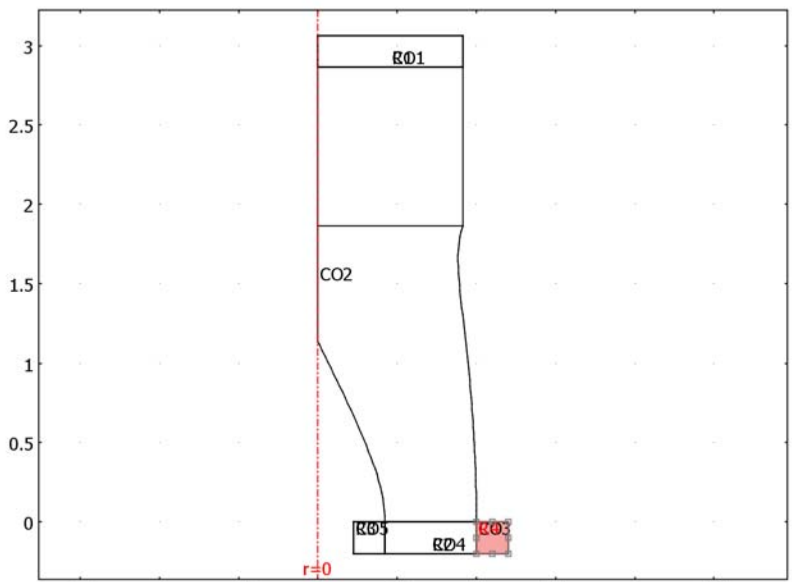


Figure 3-18: The axisymmetric duct geometry with auxiliary PML domains (R1–R4).

MESH GENERATION

1 Choose **Mesh>Mapped Mesh Parameters**, then in the resulting dialog box go to the **Boundary** page.

2 Select Boundary 7. Select the **Constrained edge element distribution** check box, and in the **Number of edge elements** edit field type 40.

3 Repeat the procedure in Step 2 using the data in the following table:

BOUNDARIES	NUMBER OF EDGE ELEMENTS
22, 61, 98, 99	18
60	60
1	39
2	1
44–59, 62–65	2
8–19, 23–39, 66–94	3
40, 95	5

4 Click **OK** to close the dialog box.

5 Click the **Mesh All (Mapped)** button on the Mesh toolbar to generate the mesh. The result should look like that in Figure 3-19.

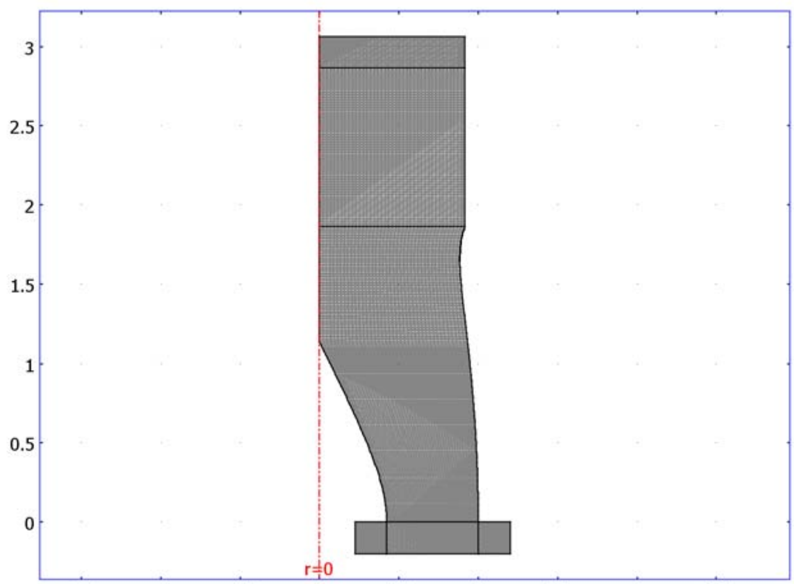


Figure 3-19:The meshed geometry.

Note that the mesh you just created is significantly denser than required to accurately model the background flow. However, in order to resolve the acoustic perturbations—which appear on a smaller scale—it is necessary to use a fine mesh.

OPTIONS AND SETTINGS

Model Settings

As explained in the introduction, 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. To reflect these nondimensionalizations, change the unit system from the default SI units to none:

1 Choose **Physics>Model Settings**.

2 From the **Base unit system** list choose **None**, then click **OK**.

Constants

1 Choose **Options>Constants**.

2 In the dialog box that opens, enter the data given in the following table (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
gamma	1.4	Ratio of specific heats
M	-0.5	Mean flow Mach number
m	10	Circumferential mode number
omega	16	Angular frequency
A	0.01	Acoustic source strength
Z	2-i	Duct wall impedance
b	0.01	Impedance onset rate
zi	1.86393	Axial coordinate, inlet plane
zt	2.86393	Axial coordinate, terminal plane

Global Expressions

1 Choose **Options>Expressions>Global Expressions**.

2 In the dialog box that opens define an expression for the mean-flow speed of sound with the following data; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
C	sqrt(gamma*p0_acpf*rho^(gamma-1)/ rho0_acpf^gamma)	Mean flow speed of sound

Subdomain Expressions

1 Choose **Options>Expressions>Subdomain Expressions**.

2 Define variables with the scope, name, and expressions from the following table; when done, click **OK**.

SUBDOMAINS	NAME	EXPRESSION
4–6	rho	rho0_acpf
5	V	M

Boundary Expressions

1 Choose **Options>Expressions>Boundary Expressions**.

2 Select Boundary 43, then define an expression for the source mode's intensity component normal to the source boundary according to the following table:

NAME	EXPRESSION
Iz_source	A^2*0.5*real((p_acab/rho0_acpf+Vr_acab*vr_acab-Vz_acab *i*kz*phi_b)*conj(rho*Vz_acab-rho0_acpf*i*kz*phi_b))

This expression is the acab application mode's equivalent of I_z in Table 6-3 on page 121 of the *Acoustics Module User's Guide*. The expression involves a constant `kz` for the out-of-plane wave number that you specify later. It is convenient to introduce this constant here instead of using the application scalar variable `ikz_acab` set equal to `i*kz`. This is because `ikz_acab` must take its default value `-lambda` when you solve for the boundary mode, and you would therefore have to change its value back and forth repeatedly between the various modeling stages.

3 Click **OK** to close the **Boundary Expressions** dialog box.

Because the definition of the boundary mode's acoustic pressure variable, `p_acab`, involves `ikz_acab`, you must additionally modify its expression as follows:

4 From the **Physics** menu, choose **Equation System>Boundary Settings**.

5 In the **Boundary Settings - Equation System** dialog box, click the **Variables** tab.

6 With Boundary 43 still selected, edit the **Expression** for `p_acab` by replacing `ikz_acab` with `i*kz`.

7 Click **OK** to close the **Equation System>Boundary Settings** dialog box.

Note that `p_acab` is just a postprocessing variable, so changing its expression does not affect the solution.

Boundary Variables

Define some expressions that you later use to extend the mean-flow density and velocity variables to the upper PML domain by means of extrusion from the terminal plane at $z = 2.86393$. Do so by executing the following instructions:

1 Choose **Options>Extrusion Coupling Variables>Boundary Variables**.

2 Go to the **Source** page. Select Boundary 6, then define variables with the following data:

NAME	EXPRESSION	TRANSFORMATION
`V`	`vz_acpf`	General
`rho`	`rho`	General

3 Click the **Destination** tab.

4 In the **Level** list select **Subdomain**, and in the **Variable** list select **V**.

5 In the **Subdomain selection** list select the check box next to Subdomain 3. The **Use selected subdomains as destination** check box is then automatically selected.

6 From the **Variable** list select **rho**, then select the **Use selected subdomains as destination** check box.

7 Click **OK**.

Next define two integration coupling variables that evaluate to the input acoustic power produced by the noise source and the power through the inlet plane.

1 Choose **Options>Integration Coupling Variables>Boundary Variables**.

2 Enter the data given in the following table below, leaving the **Global destination** check box selected (the default) for both variables; when done, click **OK**.

BOUNDARY	NAME	EXPRESSION
43	`w_source`	`Iz_source`
4	`w_inlet`	`Iz_acae`

Scalar Expressions

1 Choose **Options>Expressions>Scalar Expressions**.

2 In the dialog box that opens define an expression for the acoustic attenuation with the following data; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`dw`	`10*log10(w_source/w_inlet)`	`Acoustic attenuation (dB)`

PHYSICS SETTINGS

Application Mode Properties

1 Choose **Physics>Scalar Variables** to open the **Application Scalar Variables** dialog box.

2 With the **Synchronize equivalent variables** check box selected, modify the variables according to the following table; when done, click **OK**.

NAME	EXPRESSION
`freq_acab`	`omega/(2*pi)`
`m_acab`	`10`
`p0_acpf`	`rho0_acpf^gamma/gamma`
`rho0_acpf`	`1`
`v0_acpf`	`M`

Stage I—The Background Flow

In this modeling stage you derive the stationary background on which propagate the time-harmonic acoustic perturbations that you model in Stage III. You calculate this flow field using the Compressible Potential Flow application mode defined on the duct geometry proper (Subdomain 1) and on the auxiliary region (Subdomain 2) appended at the inlet plane ($z = 1.86393$), and subsequently extend the flow to the PML domains by continuity. A mass-flow boundary condition is imposed at the source plane, and a Dirichlet condition for the velocity potential is applied at the terminal plane ($z = 2.86393$). The duct wall and the spinner are both impervious to the flow.

PHYSICS SETTINGS

Subdomain Settings

1 In the **Multiphysics** menu select **Compressible Potential Flow (acpf)**.

2 Choose **Physics>Subdomain Settings**.

3 In the **Subdomain selection** list select Subdomains 1–2, then type `gamma` in the γ edit field.

4 In the **Subdomain selection** list select Subdomains 3–6, then clear the **Active in this domain** check box.

5 Click **OK**.

Boundary Conditions

1 Choose **Physics>Boundary Settings**.

2 In the **Boundary selection** list select Boundaries 1 and 3.

3 From the **Boundary condition** list choose **Axial symmetry**.

4 Select Boundary 6 (the terminal plane), and as the **Boundary condition** select **Normal flow**.

5 Select Boundary 43 (the source plane), and as the **Boundary condition** select **Mass flow**. Leave the **Normal velocity** and **Fluid density** edit fields at their default values.

6 Click **OK**.

COMPUTING THE SOLUTION

1 Choose **Solve>Solver Manager** or click the corresponding button on the Main toolbar. Either action opens the **Solver Manager** dialog box.

2 Click the **Solve For** tab.

3 In the **Solve for variables** list select **Compressible Potential Flow (acpf)**, then click **Apply**.

4 Choose **Solve>Solver Parameters** (or click the corresponding button on the Main toolbar) to open the **Solver Parameters** dialog box.

5 From the **Solver** list select **Stationary**, then click **OK**.

6 Return to the **Solver Manager** dialog box and click **Solve**.

7 On the **Initial Value** page, click the **Store Solution** button.

8 Click **OK**.

POSTPROCESSING AND VISUALIZATION

First zoom in on the interesting part of the geometry.

1 Choose **Options>Suppress>Suppress Subdomains**.

2 From the **Subdomain selection** list select Subdomains 3–6, then click **OK**.

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

Next visualize the mean-flow field.

1 Choose **Postprocessing>Plot Parameters** or click the corresponding button on the Main toolbar.

2 On the **General** page, go to the **Plot type** area and select only the **Surface**, **Contour**, and **Geometry edges** check boxes.

3 On the **Surface** page go to the **Predefined quantities** list and select **Compressible Potential Flow (acpf)>Density** (alternatively, type `rho` in the **Expression** edit field).

4 On the **Contour** page go to the **Predefined quantities** list and select **Compressible Potential Flow (acpf)>Velocity potential**.

5 On the **General** page click the **Title** button.

6 Click the option button next to the edit field and enter the title `Surface: density    Contour: velocity potential (M=-0.5)`.

7 Click **OK** to close the **Title** dialog box, then click **OK** to close the **Plot Parameters** dialog box and generate the plot presented in Figure 3-13 on page 119.

To get a detailed view of the density and velocity profiles along the length of the duct, proceed as follows:

1 Choose **Postprocessing>Cross-Section Plot Parameters**.
2 On the **General** page select the **Keep current plot** check box and clear the **Display cross-section in main axes** check box
3 Click the **Line/Extrusion** tab.
4 In the **y-axis data** area, find the **Expression** edit field and enter `rho`.
5 In the **x-axis data** area, select **z** from the list next to the upper option button.
6 In the **Cross-section line data** area, modify the coordinate values according to the following table; when done, click **Apply**.

r0, r1	0.8
z1	1.86393

7 In the **y-axis data** area, go to the **Expression** edit field and type `-vz_acpf`.
8 Click the **Line Settings** button.
9 From the **Line color** list, select **Color**, then in the **Line style** list select **Dashed line**. Click **OK** to close the **Line Settings** dialog box.
10 In the **Cross-Section Plot Parameters** dialog box, click the **General** tab.
11 Click the **Title/Axis** button to open the **Title/Axis Settings** dialog box.
12 Click the option button next to the **Title** edit field and enter the text `Mean flow cross-section plots at r=0.8`.
13 Click the option button next to the **Second axis label** edit field and enter the text `Density (solid)    axial Mach number (dashed)`.
14 Click **OK** to close the **Title/Axis Settings** dialog box, then click **OK** to close the **Cross-Section Plot Parameters** dialog box. Doing so generates the graphs in Figure 3-14 on page 120.

Stage II—The Boundary Source Mode

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.

PHYSICS SETTINGS

Boundary Conditions

1 In the **Multiphysics** menu select **Aeroacoustics, Boundary Modal Analysis (acab)**.
2 Choose **Physics>Boundary Settings**.
3 In the **Boundary selection** list select Boundary 43.
4 In the left and right parts of the **V** edit field type `vr_acpf` and `vz_acpf`, respectively.
5 In the $\mathbf{c_s}$ edit field type `C`, and in the ρ edit field type `rho`.
6 Select Boundary 40, then select the **Select by group** check box and clear the **Active in this domain** check box.
7 Click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Manager** button on the Main toolbar to open the **Solver Manager** dialog box.
2 On the **Initial Value** page select the **Stored solution** option button in both the **Initial value** area and the **Values of variables not solved for and linearization point** area.
3 Click the **Solve For** tab.
4 In the **Solve for variables** list select **Aeroacoustics, Boundary Modal Analysis (acab)**, then click **Apply**.
5 Click the **Solver Parameters** button on the Main toolbar to open the **Solver Parameters** dialog box.
6 From the **Solver** list select **Eigenvalue**.
7 In the **Desired number of propagation constants** edit field type `8`, and in the **Search for propagation constants around** edit field type `10`.
8 Click **OK** to close the **Solver Parameters** dialog box.
9 Return to the **Solver Manager** dialog box and click **Solve**.
10 Go to the **Initial Value** page and click the **Store Solution** button to open the **Store Solution** dialog box.

Inspect the **Propagation constant** list and notice that there are four solutions with a purely real propagation constant, 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 propagation constants, which in the absence of a mean flow would be symmetrically distributed around zero.

In the subsequent modeling stage, you will use the value of the largest positive propagation constant (≈27.12) and the absolute value of the single negative propagation constant (≈5.778); these propagation constants correspond to the lowest forward-propagating and backward-propagating modes, respectively.

11 Select the last entry from the **Propagation constant** list to store the solution corresponding to the lowest forward-propagating mode. Click **OK**.

12 Choose **Options>Constants** and add the following entries to the list of constants:

NAME	EXPRESSION	DESCRIPTION
`kz`	`27.123744`	`Axial propagation constant`
`kzr`	`5.778166`	`Axial propagation constant, reflected mode`

(In the expression column, preferably use the exact values you obtained for the propagation constants, although they should be quite close to those displayed here.)

13 Click **OK** to close the **Solver Manager** dialog box.

POSTPROCESSING AND VISUALIZATION

To verify that you have found the lowest axial mode, plot the boundary mode velocity potential as follows:

1 Choose **Postprocessing>Domain Plot Parameters**.

2 On the **General** page select the last entry in the **Solutions to use** list.

3 From the **Plot in** list select **New figure**, then select the **Keep current plot** check box.

4 Click the **Title/Axis** button, then select the option button next to the **Title** edit field. Enter the text `First axial mode at source plane (z=0)`. Click **OK**.

5 On the **Line/Extrusion** page go to the **Boundary selection** list and select Boundary 43.

6 In the **x-axis data** area select **r** from the list next to the upper option button. In the **y-axis data** area keep the default variable choice, **phi_b**.

7 Click **OK** to close the dialog box and generate the plot in Figure 3-15 on page 121.

Stage III—The Acoustic Field

Equipped with the solution derived in the two previous stages, you can now go on to simulate the acoustic field. You model the noise source through the judicious choices of boundary conditions at the source plane ($z = 0$) for the two time-harmonic Aeroacoustics application modes. 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.

OPTIONS AND SETTINGS

Constants

1 Choose **Options>Constants**.

2 Add a constant with the name `pmax` defined by the expression `1.500211`.

3 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings—Aeroacoustics (acae)

1 In the **Multiphysics** menu select **Aeroacoustics (acae)**.

2 Choose **Physics>Subdomain Settings**.

3 In the **Subdomain selection** list select Subdomains 4–6, then clear the **Active in this domain** check box.

4 Select Subdomains 1–3, then in the $\mathbf{c_s}$ edit field type `C`.

5 Select Subdomain 3 only.

6 In the left part of the **V** edit field type `0`, and in the right part type `V`.

7 Click the **PML** tab.

8 In the **Type of PML** list select **Cylindrical**.

9 Select the **Absorbing in z direction** check box. Specify the PML width in this direction by typing `0.2` in the corresponding edit field.

10 In the $\mathbf{L_z}$ edit field type `2*pi/kz` to specify the scaled PML length.

11 Click **OK**.

Subdomain Settings—Aeroacoustics (acae2)

1 In the **Multiphysics** menu select **Aeroacoustics (acae2)**.

2 Choose **Physics>Subdomain Settings**.

3 Select Subdomains 1–3, then clear the **Active in this domain** check box.

4 Select Subdomains 4–6.

5 In the $\mathbf{c_s}$ edit field type `C`, and in the ρ edit field type `rho`.

6 Select Subdomain 5 by itself.

7 In the right-hand part of the **V** edit field type `V`, and in the left-hand part leave the default value, `0`.

8 Click the **PML** tab.

9 In the **Type of PML** list select **Cylindrical**.

10 Select the **Absorbing in z direction** check box and type `0.2` in the corresponding edit field.

11 In the $\mathbf{L_z}$ edit field type `2*pi/kzr`.

12 Select the two Subdomains 4 and 6.

13 From the **Type of PML** list select **Cylindrical**.

14 Select both the **Absorbing in r direction** and the **Absorbing in z direction** check boxes, then type `0.2` in the corresponding edit fields.

15 In both the $\mathbf{L_r}$ and $\mathbf{L_z}$ edit fields enter `2*pi/kzr`.

16 Select Subdomain 4 only, then in the $\mathbf{R_0}$ edit field type `0.223557`.

17 Click **OK**.

Boundary Conditions

1 Choose **Physics>Boundary Settings**.

2 Select Boundary 43.

3 In the **Boundary condition** list select **Velocity potential**.

4 In the ϕ_0 edit field type `phi-A*phi_b`, then click **OK**.

5 Choose **Physics>Properties** to open the **Application Mode Properties** dialog box.

6 In the **Weak constraints** list select **On**, then click **OK**.

7 In the **Multiphysics** menu select **Aeroacoustics (acae)**.

8 Choose **Physics>Boundary Settings**.

9 With Boundary 43 still selected, go to the **Boundary condition** list and select **Normal mass flow**. Then in the $\mathbf{m_n}$ edit field type `rho*(-i*kz*A*phi_b)`.

10 Select Boundaries 1, 3, and 5, then in the **Boundary condition** list select **Axial symmetry**.

11 Click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Manager** button on the Main toolbar.

2 Click the **Solve For** tab.

3 In the **Solve for variables** list select both **Aeroacoustics (acae)** and **Aeroacoustics (acae2)**, then click **OK**.
4 Click the **Solver Parameters** button on the Main toolbar.
5 From the **Solver** list select **Stationary**, then click **OK**.
6 Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

Zoom in on the duct geometry proper, that is, Subdomain 1.

1 Choose **Options>Suppress>Suppress Subdomains**.
2 Select Subdomains 2–6, then click **OK**.
3 Click the **Zoom Extents** button on the Main toolbar.

Next visualize the aeroacoustic field.

1 Click the **Plot Parameters** button on the Main toolbar.
2 On the **General** page clear the **Surface** check box.
3 Click the **Contour** tab.
4 In the **Expression** edit field type `p_acae/pmax`.
5 Select the **Vector with isolevels** option button, then type the following vector in the corresponding edit field: `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 In the **Colormap** list select **hsv**, then click **Apply**.
7 Click the **General** tab.
8 Select the **Keep current plot** check box, and then return to the **Contour** page.
9 In the **Colormap** list select **jet**.
10 Select the **Filled** check box, then click **Apply**.
11 Go to the **General** page and click the **Title** button.
12 Select the check box next to the edit field and enter the title `Hard wall, flow`.
13 Click **OK** to close the **Title** dialog box, then click **OK** to close the **Plot Parameters** dialog box.

In the drawing area of the COMSOL Multiphysics user interface you should now see the upper image in Figure 3-17 on page 123, which shows the acoustic field in a mean-flow field of axial Mach number –0.5 for the case of a hard duct wall.

Next change the model to that of a duct with a lined wall by implementing an impedance boundary condition. Begin by adjusting the value of p_{max} so as to obtain a normalized plot.

OPTIONS AND SETTINGS

Constants

1 Choose **Options>Constants**.

2 Change the value of `pmax` to `1.718862`, then click **OK**.

PHYSICS SETTINGS

Boundary Conditions

1 Choose **Physics>Boundary Settings**.

2 Select Boundaries 44–59 and 62–95.

3 From the **Boundary condition** list select **Impedance boundary condition**.

4 In the **Z** edit field type `Z/flc2hs(z/zi,b)`.

5 Select Boundary 60 by itself, then from the **Boundary condition** list select **Impedance boundary condition**.

6 In the **Z** edit field type `Z/flc2hs((zt-z)/(zt-zi),b)`, then click **OK**.

The reason behind using the smoothed Heaviside function `flc2hs` in Steps 4 and 6 above 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).

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page clear the **Keep current plot** check box.

3 Click the **Contour** tab.

4 In the **Colormap** list select **hsv**.

5 Clear the **Filled** check box, then click **Apply**.

6 Click the **General** tab.

7 Select the **Keep current plot** check box, then return to the **Contour** page.

8 In the **Colormap** list select **jet**.

9 Select the **Filled** check box, then click **Apply**.

10 Go to the **General** page and click the **Title** button.

11 Change the text in the title edit field to `Lined wall, flow`, then click **OK** to close the **Title** dialog box.

12 Click **OK** to close the **Plot Parameters** dialog box.

The plot in the main window should now closely resemble the lower image in Figure 3-17.

With a lined duct wall, the acoustic perturbations emanating from the noise source no longer propagate without loss. To display the acoustic attenuation between the source plane and the inlet plane at $z = 1.86393$, execute the following commands:

1 Choose **Postprocessing>Data Display>Global**.

2 In the **Expression** edit field type `dw`, then click **OK**.

The value for the current solution of this previously defined expression—approximately 22.0 dB—now appears in the Message log at the bottom of the COMSOL Multiphysics user interface.

Note: 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. This observation is borne out by a comparison of the calculated attenuation of 22.0 dB with the corresponding quantity calculated in Ref. 1, where a value of approximately 27 dB was obtained.

The Case Without a Background Flow

Follow these instructions to repeat the calculations for the case when no background mean-flow field is present.

OPTIONS AND SETTINGS

Constants

Choose **Options>Constants**. Change the value of `M` to `0`, then click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Manager** button on the Main toolbar.

2 On the **Solve For** page go to the **Solve for variables** list and select **Compressible Potential Flow (acpf)**.

3 Click the **Initial Value** tab.

4 In the **Initial value** area select the **Initial value expression** option button.

5 In the **Values of variables not solved for and linearization point** area, select the **Use selection from Initial value frame** option button.

6 Click the **Solve** button, then click the **Store Solution** button.

7 Select the **Stored solution** option button in both the **Initial value** area and the **Values of variables not solved for and linearization point** area.

8 Go back to the **Solve For** page and select **Aeroacoustics, Boundary Modal Analysis (acab)**, then click **Apply**.

9 Click the **Solver Parameters** button, select **Eigenvalue** from the **Solver** list, then click **OK** to close the **Solver Parameters** dialog box.

10 Return to the **Solver Manager** dialog box and click the **Solve** button.

11 Go to the **Initial Value** page and click the **Store Solution** button.

12 Return to the main user interface and from the **Options** menu open the **Constants** dialog box.

13 Change the expressions for both `kz` and `kzr` to the value of the last entry in the **Propagation constant** list, which should be approximately 10.83. Click **OK**.

14 Return to the **Store Solution** dialog box and from the **Propagation constant** list select the last entry (with a value of the propagation constant approximately equal to 10.83). Click **OK**.

15 Click **OK** to close the **Solver Manager** dialog box.

Having computed and saved the mean-flow-free background and boundary mode solution, turn next to the calculation of the acoustic field.

OPTIONS AND SETTINGS

Constants

1 Choose **Options>Constants**.

2 Change the value of `pmax` to `0.422433`, then click **OK**.

PHYSICS SETTINGS

Boundary Conditions

1 Choose **Physics>Boundary Settings**.

2 From the **Boundary selection** list select Boundary 44.

3 Select the **Select by group** check box, then in the **Boundary condition** list select **Sound hard boundary (wall)**.

4 Select Boundary 60, then in the **Boundary condition** list select **Sound hard boundary (wall)**.

5 Click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Manager** button on the Main toolbar.

2 On the **Solve For** page go to the **Solve for variables** list and select both **Aeroacoustics (acae)** and **Aeroacoustics (acae2)**. Click **OK**.

3 Click the **Solver Parameters** button on the Main toolbar.

4 From the **Solver** list select **Stationary**, then click **OK**.

5 Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page clear the **Surface** check box.

3 Click the **Contour** tab.

4 In the **Colormap** list select **hsv**.

5 Clear the **Filled** check box, then click **Apply**.

6 Click the **General** tab.

7 Select the **Keep current plot** check box, then return to the **Contour** page.

8 In the **Colormap** list select **jet**.

9 Select the **Filled** check box, then click **Apply**.

10 Go to the **General** page and click the **Title** button.

11 Select the check box next to the edit field and enter the title `Hard wall, no flow`.

12 Click **OK** to close the **Title** dialog box, then click **OK** to close the **Plot Parameters** dialog box.

A plot similar to the one in the upper half of Figure 3-16 on page 122 should now appear in the drawing area. It depicts the acoustic field in the duct in the absence of a background mean flow and for the case of a hard duct wall.

As the fourth and final case of study, consider a lined duct wall in the absence of a background flow.

OPTIONS AND SETTINGS

Constants

1. Choose **Options>Constants**.
2. Change the value of `pmax` to `0.264331`, then click **OK**.

PHYSICS SETTINGS

Boundary Conditions

1. Choose **Physics>Boundary Settings**.
2. Select Boundaries 44–59 and 62–95, then in the **Boundary condition** list select **Impedance boundary condition**.

   The expression `Z/flc2hs(z/zi,b)` should appear in the **Z** edit field.
3. Select Boundary 60, then in the **Boundary condition** list select **Impedance boundary condition**.

   Now the expression in the **Z** edit field should read `Z/flc2hs((zt-z)/(zt-zi),b)`.
4. Click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1. Click the **Plot Parameters** button on the Main toolbar.
2. On the **General** page clear the **Keep current plot** check box.
3. Click the **Contour** tab.
4. In the **Colormap** list select **hsv**.

5 Clear the **Filled** check box, then click **Apply**.

6 Click the **General** tab.

7 Select the **Keep current plot** check box, then return to the **Contour** page.

8 In the **Colormap** list select **jet**.

9 Select the **Filled** check box, then click **Apply**.

10 Go to the **General** page and click the **Title** button.

11 Change the text in the edit field to `Lined wall, flow`, then click **OK** to close the **Title** dialog box.

12 Click **OK**.

The plot in the main window should now closely resemble the lower one in Figure 3-16.

Finally display the value of the acoustic attenuation. To do so, choose **Postprocessing>Data Display>Global**, then click **OK**.

The value of dw—for this case approximately 50.3 dB—should appear in the Message log at the bottom of the user interface.

Note: In contrast to the attenuation calculated for the case of a mean-flow background in the first part of this exercise, the latest value 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.

Loudspeaker Driver

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 on-axis sound pressure level at a nominal driving voltage, as functions of the frequency.

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” on page 175, uses the electromechanical properties modeled here and adds a vented enclosure.

The model starts out with the Small-Signal analysis from the AC/DC Module to compute the driving force and the blocked voice coil impedance. It then uses these results and adds application modes from the Acoustics Module for the acoustic-structure analysis. As an optional final modeling step, you can choose to extract and export the data needed for the 3D model.

Model Definition

Figure 3-20 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 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 basked 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.

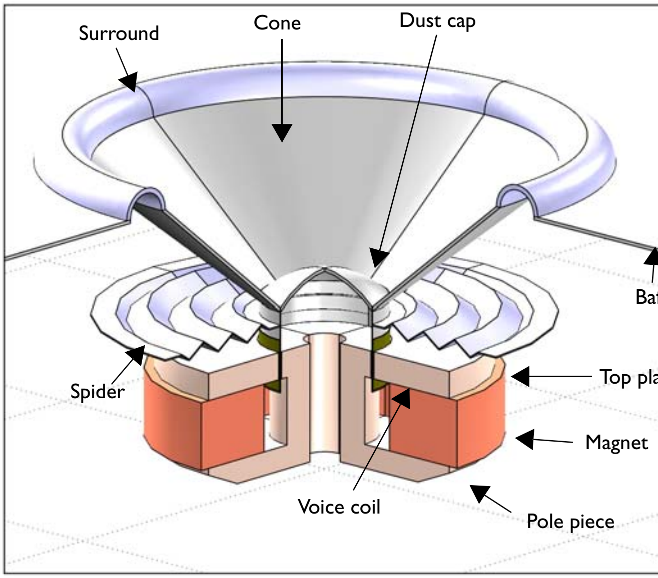


Figure 3-20: Geometry of the modeled loudspeaker driver.

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. Once the relation between the driving voltage and the force is established, the force is applied in an acoustic-structure interaction analysis set up to compute the sound production.

The force on a wire of length L and with the current I in a magnetic flux density B perpendicular to the wire is given by $F = IBL$. The voice coil consists of $N = 100$ turns of thin copper wire and occupies a cross-sectional area A in the *rz*-plane. The total force F_e from the current on the coil hence becomes

$$F_e = -I\frac{2\pi N}{A}\int rB_r\mathrm{dA} \qquad (3\text{-}1)$$

where the integral is taken over the coil domain.

The current through the voice coil relates to the applied voltage as

$$I = (V_0 + V_{\mathrm{be}})/Z_{\mathrm{b}}, \tag{3-2}$$

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 $-V_{\mathrm{be}}$ denotes the *back EMF* (the voltage induced in the coil due to its motion through the permanent magnetic field in the gap).

To evaluate the back EMF, consider the same wire of length L in the magnetic flux density B, but now traveling at a velocity v perpendicular to the wire. The wire gets an induced back EMF equal to vBL. The total back EMF in the coil becomes

$$-V_{\mathrm{be}} = -v\frac{2\pi N}{A}\int rB_r \mathrm{dA} \tag{3-3}$$

The common factor in F_{e} and V_{be} is known in the loudspeaker community as the force factor, "BL":

$$BL = -\frac{2\pi N}{A}\int rB_r \mathrm{dA}. \tag{3-4}$$

To find the blocked coil impedance, an external current density corresponding to a time-harmonic loop voltage of $(V_0 + V_i)/N$ is applied to the domain of the still stationary coil. The induced voltage V_i is computed as the calculated induced azimuthal electric field in the coil times the length of the wire. The blocked coil impedance, finally, is obtained as $Z_{\mathrm{b}} = V_0/I_{\mathrm{b}}$, where I_{b} is the current through the blocked coil.

With the constant BL and the frequency-dependent Z_{b} now both known, Equations 3-1 to 3-4 can be rearranged to form the sought relationship between V_0 and F_{e}:

$$F_{\mathrm{e}} = BL\frac{V_0 - BLv}{Z_{\mathrm{b}}} \tag{3-5}$$

Note the dependence of the driving force on the velocity of the moving coil, which is unknown prior to the acoustic-structure interaction computation.

In computing the acoustic-structure interaction, F_{e} is applied to the voice coil domain. A structural equation is solved in the moving parts of the driver, and a pressure acoustics equation in the surrounding air. The pressure acoustics equation uses a normal acceleration boundary condition for the structural vibrations to propagate into the air. Conversely, the acoustic pressure is applied as a boundary load on the structure.

The air domains and the baffle should ideally extend to infinity. To avoid unphysical reflections where you truncate the geometry, you will use Perfectly Matched Layers (PMLs), as seen in Figure 3-21. For more information, see “Perfectly Matched Layers (PMLs)” on page 37 in the *Acoustics Module User’s Guide*.

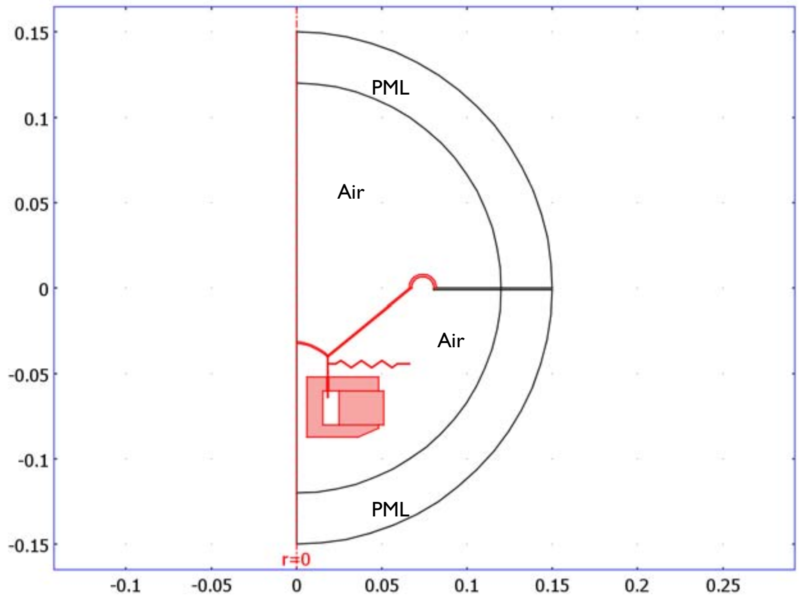


Figure 3-21: Overview of the model geometry with the structures in pink.

Although the modeled air domain has a radius of only 0.12 m, the local acoustic pressure can be extracted anywhere thanks to COMSOL’s functionality for far-field pressure computation. The sensitivity is calculated as the sound pressure level on the axis at 1 m, for the applied voltage $V_0 = 4$ V.

As a final modeling step, results are extracted from this model for use as lumped parameters in the Vented Loudspeaker Enclosure. The documentation to that example contains a circuit representation of the force balance on the cone.

Results and Discussion

The magnetic flux density in and around the voice coil appears in Figure 3-22. The maximum flux density in the air arises in the gap between the pole piece and the top plate. Performing the integral in Equation 3-4 over the voice coil domain gives a force factor $BL = 7.55$ N/A.

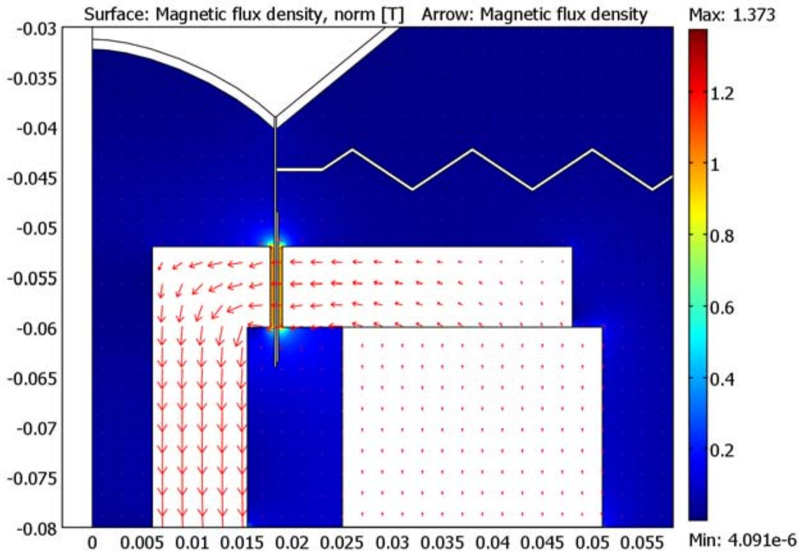


Figure 3-22: Magnetic flux density near the voice coil.

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 3-23 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.

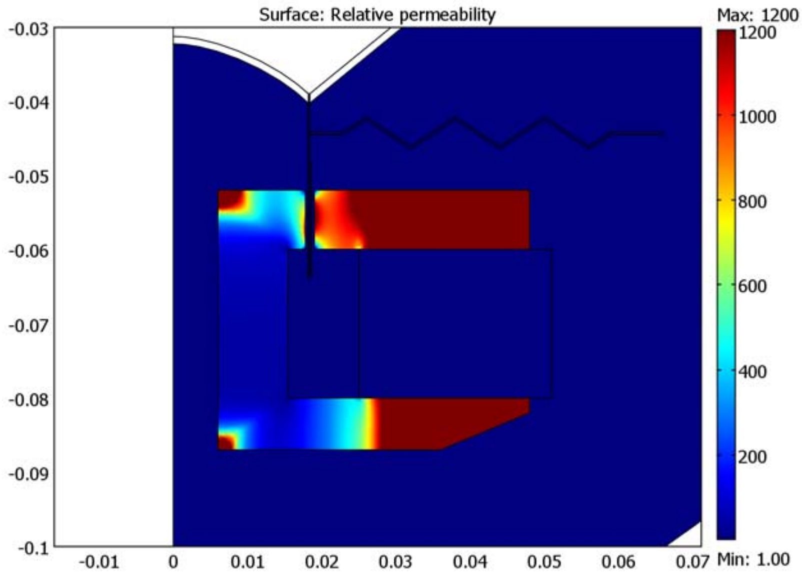


Figure 3-23: The local relative permeability in the pole piece and top plate, when subjected to the field from the magnet.

In computing the blocked coil impedance, the AC equation is linearized around the local permeability resulting from the static solution. Figure 3-24 shows the induced currents at two different frequencies.

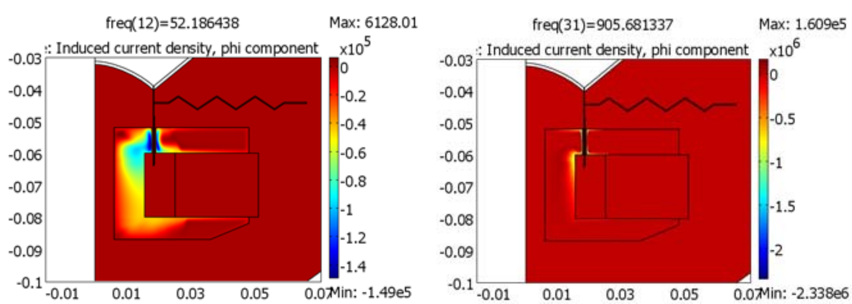


Figure 3-24: Induced currents in the pole piece and top plate, at 52 Hz (left) and 905 Hz (right).

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 3-25 shows a plot of the blocked coil inductance versus frequency.

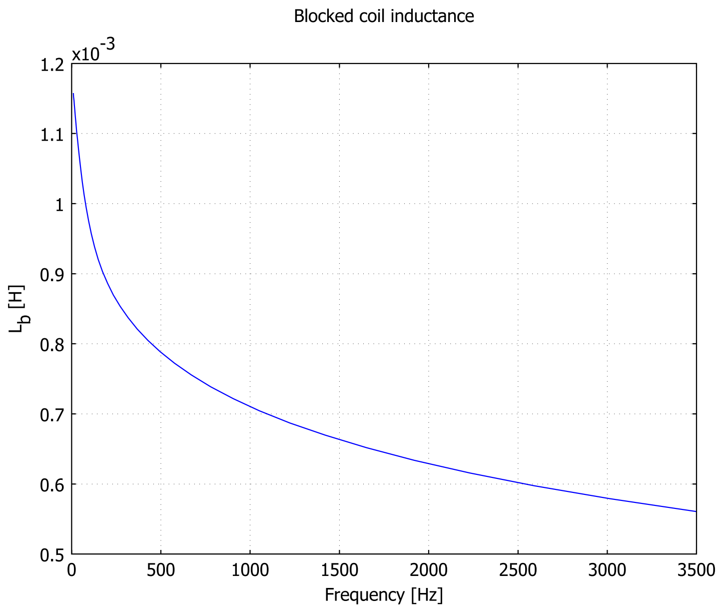


Figure 3-25: The inductance of the blocked coil as a function of frequency.

From the acoustic-structure interaction analysis, Figure 3-26 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.

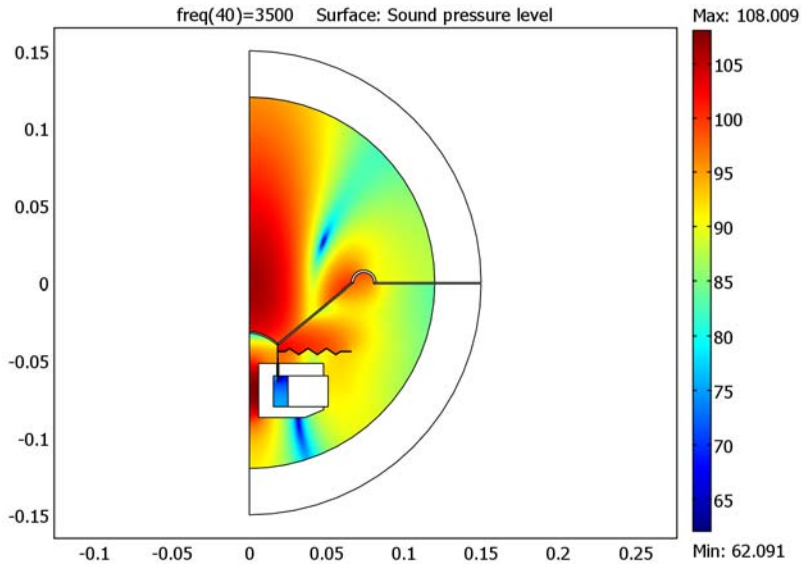


Figure 3-26: Sound pressure level distribution in dB at 3500 Hz.

Figure 3-27 presents the loudspeaker's sensitivity. 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 Vented Loudspeaker Enclosure model.

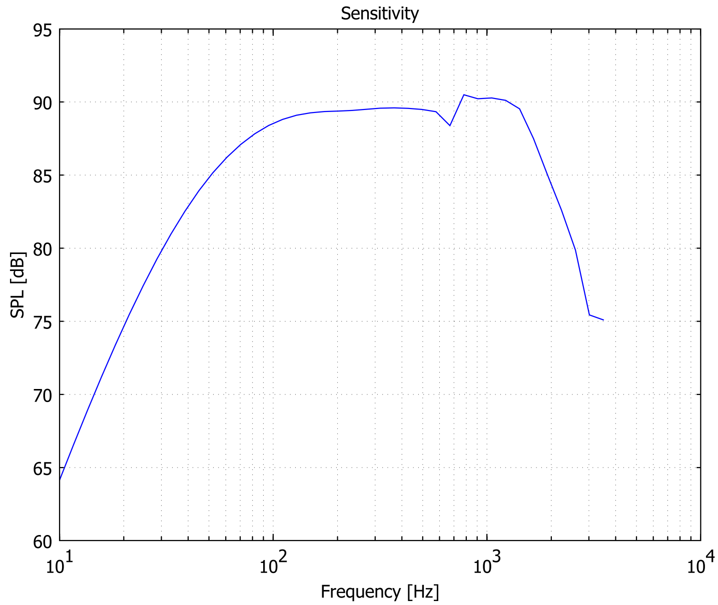


Figure 3-27: Loudspeaker sensitivity, measured as the on-axis sound pressure level (dB) at a distance of 1 m from the unit. The pressure is evaluated using an input signal of 4 V, or 2.83 V RMS, which corresponds to a power of 1 W at an 8 Ω. nominal impedance. Note the logarithmic frequency scale.

The total electrical impedance, defined as $Z = V_0/I$, appears in Figure 3-28. 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.

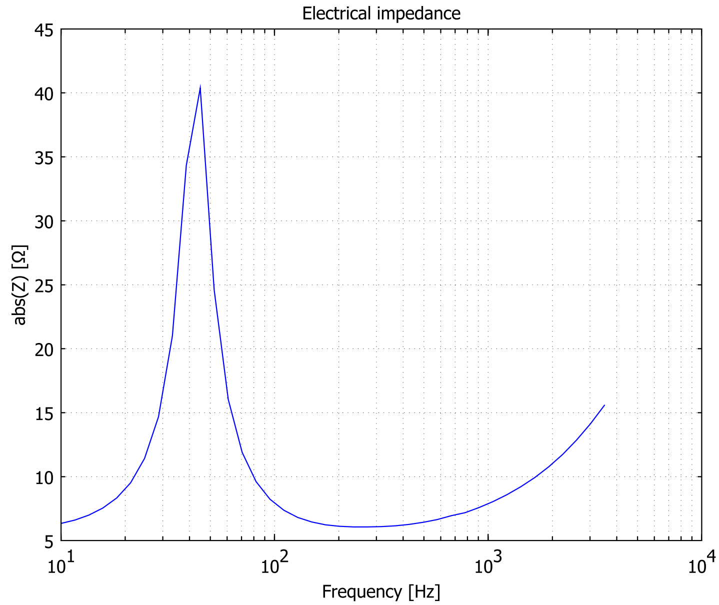


Figure 3-28: Electrical impedance (Ω) of the loudspeaker as a function of frequency (Hz).

Modeling in COMSOL Multiphysics

The step-by-step instructions takes you through the following steps:

- Perform a static analysis of the magnetic field distribution to evaluate the force factor, BL.
- Sweep the frequency in an AC analysis where you apply a time-harmonic voltage to the voice coil, to compute the blocked coil impedance, Z_b.
- Use BL and Z_b in an acoustic-structure interaction analysis computing the sound pressure level and total electrical impedance of the driver as functions of the frequency.
- (Optional): Prepare for the Vented Loudspeaker Enclosure model by performing a separate mechanical analysis of the suspension, exporting its frequency-dependent resistance and compliance as well as Z_b.

The static magnetic field computation and the AC analysis use the Small-Signal Analysis multiphysics node available in COMSOL's AC/DC Module. This node contains two application modes. First, the field from the magnet is computed in a static application mode. 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. After solving this step of the model, you can evaluate BL by integrating the magnetic flux density over the voice coil domain.

The time-harmonic application mode in the small-signal analysis inherits the permeability distribution from the static application mode. For the time-harmonic 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 hardly the situation here, linearizing around a local biased permeability should still be a better approximation than assuming a constant permeability. The very 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 node features a structural Stress-Strain application mode for the moving structures and a Pressure Acoustics application mode for the air. It also provides the boundary conditions for the two-way acoustic coupling between the air and the structures.

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.

Model Library path: `Acoustics_Module/Industrial_Models/loudspeaker_driver`

Model Library path: `Acoustics_Module/Industrial_Models/loudspeaker_suspension`

MODEL NAVIGATOR

1. In the **Model Navigator** go to the **Space dimension** list and select **Axial symmetry (2D)**.
2. In the list of application modes select **AC/DC Module>Quasi-Statics, Magnetic>Azimuthal Induction Currents, Vector Potential>Small-signal analysis**.
3. Click **OK**.

Selecting the Small-signal analysis makes two **Azimuthal Induction Currents, Vector Potential** application modes appear in the **Multiphysics** menu. They are named emqa and emqa2. The emqa application mode is for computing the static magnetic field. The time-harmonic emqa2 application mode uses the static field solution in evaluating the permeability in the partially saturated iron core.

OPTIONS AND SETTINGS

Choose **Options>Constants**. Enter the data given in this table (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`N`	`100`	`Number of turns in coil`
`B0`	`0.4[T]`	`Remanent flux density in magnet`

GEOMETRY MODELING

The geometry in this model can be created with COMSOL's drawing tools, but to save time, you will import it from a prepared file.

1. Choose **File>Import>CAD Data From File**.
2. Browse to the file `loudspeaker_driver.mphbin`, which is located in your COMSOL Multiphysics installation folder under `models/Acoustics_Module/Industrial_Models`.

PHYSICS SETTINGS

Subdomain Settings

In order to evaluate the loudspeaker's force factor you need to compute the magnetic flux density in the voice coil's air gap. This simulation is made with the emqa application mode.

1. In the **Multiphysics** menu, select the first one of the two application modes, named **emqa**.

The field from the static magnet is mostly contained in the magnetic motor of the driver, and decreases the further out you get from it. To minimize the computational effort, it makes sense to deactivate those subdomains where you can expect the field to be negligible.

2 Choose **Physics>Subdomain Settings**.

3 Select Subdomains 1, 3–5, 10, and 13. Clear the **Active in this domain** check box.

Next, apply a magnetization to the magnet.

4 Select Subdomain 12. For the **Constitutive relation** pick $\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$, then type `B0` in the right-most of the two edit fields for remanent flux density. This gives a remanent flux density equal to 0.4 T in the z direction.

The pole piece and top plate will take their nonlinear B-H relation from the Material Library. The electric conductivity is also assigned and will be used in the subsequent impedance computation.

5 Select Subdomains 6 and 11. Click the **Load** button, then in the **Electric (AC/DC) Material Properties** list select **Soft Iron (with losses)** and click **OK**.

The cone apex, the voice coil, and the spider are not magnetic. Therefore, as for the surrounding air, these parts will use the default value for the relative permittivity, $\mu_r = 1$.

6 Click **OK** to close the dialog box.

Boundary Conditions

1 Choose **Physics>Boundary Settings**.

2 Select Boundary 2, then in the **Boundary condition** list select **Axial symmetry**.

Of the boundaries along the z-axis, Boundary 2 is the only one that is active in the magnetic model. All other exterior boundaries use the default **Magnetic insulation**, which is reasonable where the field is small.

3 Click **OK** to close the dialog box.

Coupling Variables

1 Choose **Options>Integration Coupling Variables>Subdomain Variables**.

2 In the **Subdomain Integration Variables** dialog box select the voice coil (Subdomain 8) and create the following subdomain integration variables:

NAME	EXPRESSION
A	1
BL	-N*Br_emqa*2*pi*r/A

(Use the default settings for **Integration order** and **Global destination**.) The **Expression** field contains the integrand to be integrated over the voice coil subdomain. A will hence evaluate to the cross-sectional area of the voice coil, making the force factor BL roughly equal to the number of windings times the mean flux density in the voice coil.

MESH GENERATION

To optimize the accuracy of the force factor and prepare for the impedance computation, you need a dense mesh throughout the magnetic motor. The voice coil, where your integration coupling variable evaluates the force factor, is so thin that the default settings automatically will give a sufficient resolution.

1. Choose **Mesh>Free Mesh Parameters** and click the **Subdomain** tab.
2. Press Ctrl+A to select all subdomains, then set the **Method** to **Triangle**.
3. Select only Subdomains 6, 11, and 12, then set the **Maximum element size** to `1e-3`.
4. Click the **Remesh** button.
5. When the mesher has finished, click **OK** to close the **Free Mesh Parameters** dialog box.

COMPUTING THE SOLUTION

1. Choose **Solve>Solver Manager**.
2. On the **Solve For** page, select only the first application mode, solving for **Aphi**.
3. Click **OK**, then the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

The default plot shows the magnetic flux density. You can use the **Zoom Window** button to zoom in on the magnetic motor. If you keep only the air subdomain visible and suppress all others from the view, it will become apparent that the gap acts to focus the flux.

1. Choose **Options>Suppress>Suppress Subdomains**.
2. Select Subdomain 2, then click **Apply**.
3. Click the **Invert Suppression** button, then click **OK**.
4. Click the **Postprocessing Mode** button on the Main toolbar.

Proceed to visualize the direction and relative magnitude of the flux with an arrow plot.

1. Choose **Postprocessing>Plot Parameters** and click the **Arrow** tab.
2. On the **Arrow** page, select the **Arrow** plot check box.

3 From the list of **Predefined quantities**, select **Azimuthal Induction Currents, Vector Potential (emqa)>Magnetic flux density**.

4 Under **Number of points**, enter `60` both for **r points** and **z points**.

5 Click **OK** to see the plot and reproduce Figure 3-22.

The force factor should ideally be independent of the momentary location of the voice coil in the air gap. For this to happen, the flux density should be nearly constant in the air gap and close to zero outside of it. Proceed to visualize the flux density as a function of the z-coordinate along the voice coil.

1 Choose **Postprocessing>Domain Plot Parameters**.

2 On the **Line/Extrusion** page select Boundary 29 and type `normB_emqa` in the **Expression** edit field. Click **OK** to generate the new plot, which should resemble that in Figure 3-29.

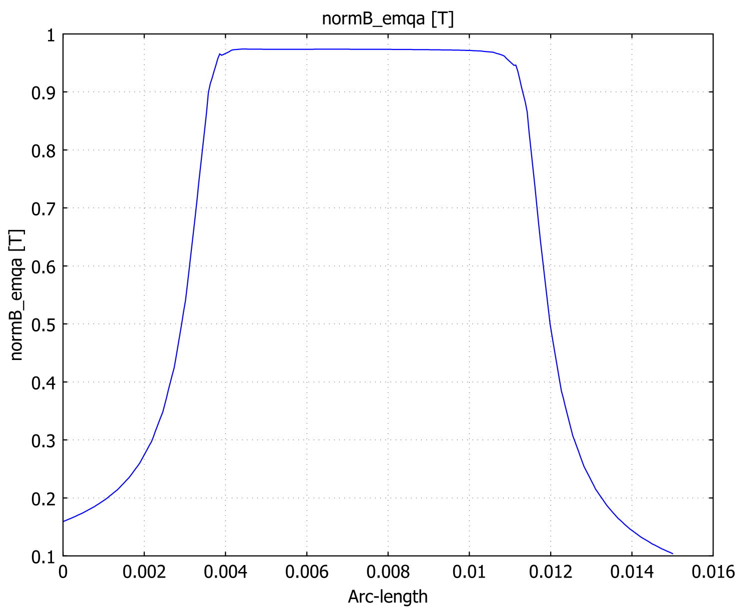


Figure 3-29: Magnetic flux density along the voice coil.

Next, find out the value of the force factor.

1 Choose **Postprocessing>Data Display>Global**.

2 In the **Expression** edit field type `BL`. Click **OK**.

The force factor evaluates to 7.55 (N/A). As a final postprocessing step on the static solution, make a plot of the relative permeability.

1 Choose **Options>Suppress>Suppress Subdomains**.
2 Click the **Suppress None** button, then **OK**.
3 In the **Plot Parameters** dialog box, go to the **Arrow** page and clear the **Arrow plot** check box.
4 Go to the **Surface** page. From the list of **Predefined quantities**, select **Azimuthal Induction Currents, Vector Potential (emqa)>Relative permeability**.
5 Click **OK** to see the plot and reproduce Figure 3-23.

Modeling Using the Graphical User Interface—Blocked Impedance

OPTIONS AND SETTINGS

Choose **Options>Constants** and add the following constants to the list. The descriptions are optional.

NAME	EXPRESSION	DESCRIPTION
`V0`	`4[V]`	`Peak driving voltage`
`R0`	`5.6[ohm]`	`DC resistance of coil`
`freq`	`100[Hz]`	`Sample driving frequency`

The peak driving voltage of 4 V corresponds to an RMS level of 2.83 V. Defining `freq`, which you will use as the solver parameter, as a constant is not necessary However, it gives you the option of verifying that your model setup is complete by solving for a single frequency with the stationary solver before doing the full frequency sweep.

Coupling Variables

1 Choose **Options>Integration Coupling Variables>Subdomain Variables**.
2 In the **Subdomain Integration Variables** dialog box select the voice coil (Subdomain 8) and add the following subdomain integration variables:

NAME	EXPRESSION
`Vi`	`N*Ephi_emqa2*2*pi*r/A`
`Ib`	`Jphi_emqa2/N`

You have now defined V_i, the induced voltage in the blocked voice coil, as the mean induced azimuthal electric field in the coil subdomain times the total wire length.

I_b, the current through the blocked coil, is the total current through the subdomain divided by the number of turns.

3 Choose **Options>Expressions>Global Expressions**. In the resulting dialog box enter the following expressions and descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`Zb`	`V0/Ib`	`Blocked coil impedance (ohm)`
`Lb`	`imag(Zb)/(2*pi*freq)`	`Blocked coil inductance (H)`

The blocked coil impedance and inductance are now defined in terms of the applied voltage and the yet-to-be-computed current through the blocked coil. Note that because they are functions of coupling variables, which do not support units, these expressions have undefined units in the COMSOL user interface.

PHYSICS SETTINGS

Subdomain Settings

1 In the **Multiphysics** menu, select the second one of the two application modes, named **emqa2**.

2 Choose **Physics>Subdomain Settings**.

3 Select the same subdomains that you deactivated in the emqa application mode (Subdomains 1, 3–5, 10, and 13). Clear the **Active in this domain** check box.

4 Select Subdomain 8 and set the **External current density** to `N*(V0+Vi)/R0/A`.

The current through each of the N turns of the coil is now equal to the sum of the applied voltage and the negative induced voltage, divided by the resistance of the coil. The emqa2 application mode inherits the conductivity and the computed relative permeability from the emqa application mode, so there is no need to reapply any material settings.

Note: Normally, the **Loop potential** edit field is the most convenient way to apply a voltage. The reason you used an equivalent external current density instead is that with a loop potential, you would also need to supply a nonzero **Electric conductivity** to describe the material in the coil. This would result in the induced current being inhomogeneously distributed over the coil domain, as if it were a continuous metal cylinder. In contrast, setting the conductivity to zero and letting the coil resistance appear in the external current density means that the induced current density is forced to be constant over the coil domain. Since the current in each turn of the coil wire by necessity is the same, this is the desired homogenization of the coil.

Boundary Conditions

1 Choose **Physics>Boundary Settings**.
2 Select Boundary 2, then in the **Boundary condition** list select **Axial symmetry**.
3 Click **OK** to close the dialog box.

MESH GENERATION

The mesh used in computing the impedance needs to resolve the induced eddy currents in the pole piece and 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 $1.17 \cdot 10^7$ 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 0.1 mm along the iron surfaces that are closest to the voice coil. To apply this fine mesh mostly where it is needed, you will at the same time increase the global mesh growth rate.

1 Choose **Mesh>Free Mesh Parameters**.
2 On the **Global** page click the **Custom mesh size** button and set the **Element growth rate** to `2`.
3 On the **Boundary** page select Boundaries 8, 9, 11, 12, 30–32, and 37. Set the **Maximum element size** to `1e-4`.
4 Click the **Remesh** button.
5 When the mesher has finished, click **OK**.

COMPUTING THE SOLUTION

1 Choose **Solve>Solver Manager**.
2 On the **Initial Value** page, click the **Store Solution** button and set **Value of variables not solved for and linearization point** to **Stored solution**.
3 On the **Solve For** page, select only the second application mode, solving for **Aphi2**.

4 Click **OK** to close the dialog box.

By setting the **Value of variables not solved for** to **Stored solution**, you have now told the software to use the static magnetic field solution when doing the impedance computation.

5 Choose **Physics>Scalar Variables**. In the dialog box that appears, find the frequency `nu_emqa2` and type `freq` in the **Expression** edit field; click **OK** to close the dialog box.

6 From the **Solve** menu open the **Solver Parameters** dialog box.

7 From the **Solver** list, select **Parametric**.

8 In the **Parameter names** edit field, type `freq`.

9 In the **Parameter values** edit field type `logspace(1,log10(3500),40)`. This will generate 40 logarithmically-spaced frequencies between 10 Hz and 3.5 kHz.

10 Click **OK**, then click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

The plot still shows the relative permeability. Zoom in on the iron core and study the induced currents for a couple of frequencies.

1 From the **Postprocessing** menu select the **Plot Parameters** dialog box. Go to the **Surface** page and from the list of **Predefined quantities** select **Azimuthal Induction Currents, Vector Potential (emqa2)>Induced current density, phi component**.

2 On the **General** page, set the **Element refinement** to 3 and try a few different frequencies from the **Parameter value** list. Verify that the current extends further into the core at lower frequencies.

You can also plot the blocked coil impedance and inductance as functions of the frequency.

3 Choose **Postprocessing>Global Variables Plot**.

4 From the **Predefined quantities** list, select the **Blocked coil inductance**.

5 Click first the **>** button and then **OK** to see a plot of the blocked coil inductance versus frequency. It decreases from near 1.2 mH at 10 Hz to just below 0.6 mH at 3.5 kHz.

Modeling Using the Graphical User Interface—Acoustics

Now knowing the constant force factor and the blocked impedance versus the frequency, it is time to set up the mechanical and the acoustic parts of the loudspeaker model.

1 Choose **Multiphysics>Model Navigator**.
2 In the list of application modes select **Acoustics Module>Acoustic-Structure Interaction>Frequency response analysis**. Click **Add**.
3 Click **OK**.

Selecting **Acoustic-Structure Interaction** gives you an Axial Symmetry, Stress-Strain application mode for the moving parts in the driver and a Pressure Acoustics application mode for the acoustic pressure wave in the air.

OPTIONS AND SETTINGS

Constants

Choose **Options>Constants** and add the following constants to the existing list; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`f0`	`40[Hz]`	`Frequency where loss factor is given`
`omega0`	`2*pi*f0`	`Angular frequency where loss factor is given`

Coupling Variables

1 Choose **Options>Integration Coupling Variables>Subdomain Variables**.
2 In the **Subdomain Integration Variables** dialog box select the voice coil (Subdomain 8) and add the following subdomain integration variable:

NAME	EXPRESSION
`v`	`w_t_acaxi/A`
`Vol`	`2*pi*r`

The expression `w_t_acaxi` is the local z-velocity of the voice coil. You can find the definition of this and other application mode-specific variables from **Physics>Equation System>Subdomain Settings**, on the **Variables** page. Integrating over Subdomain 8 and dividing by its area means that the variable v will now contain the average z-velocity of the voice coil. The expression `Vol` will evaluate to the volume of the coil.

Global Expressions

Choose **Options>Expressions>Global Expressions**. Add the following scalar expressions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`I`	`(V0-BL*v)/Zb`	`Total current (A)`
`Z`	`V0/I`	`Total electrical impedance (ohm)`
`Fe`	`BL*I`	`Total electric force (N)`

You have now defined the current and the electric impedance of the voice coil in motion, as well as the electric force driving the coil. Note that in the case of zero velocity, both the current and the impedance reduce to their blocked coil counterparts, I_b and Z_b.

Application Scalar Variables

1 Choose **Physics>Scalar Variables**.

2 Make sure that the **Synchronize equivalent variables** check box is checked, then in the **Expression** field for **freq_acaxi** enter `freq`.

   Note that because the **Synchronize equivalent variables** check box is selected, **freq_acpr** will automatically change and become `freq`.

3 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings—Stress-Strain

1 Choose **Multiphysics>Axial Symmetry, Stress-Strain (acaxi)**.

2 Choose **Physics>Subdomain Settings**.

3 Select all subdomains that represent air: 1, 2, 4, and 5.

4 From the **Group** list (not the **Groups** tab), select **Fluid domain**.

   The subdomains that you have made fluid domains will be deactivated from the solid equation. The magnet assembly will not move or deform in this simulation, and should therefore also be deactivated.

5 Select Subdomains 6, 11, and 12, then clear the **Active in this domain** check box.

6 The active subdomains need material properties. Assign properties according to the table below. The damping parameters are located on the **Damping** page.

SETTINGS	SUBDOMAINS 3, 10	SUBDOMAIN 7	SUBDOMAIN 8	SUBDOMAIN 9	SUBDOMAIN 13
E	`140e9`	`70e9`	`110e9`	`0.48e9`	`1.6e6`
ν	`0.42`	`0.33`	`0.35`	`0.3`	`0.4`
ρ	`720`	`2e3`	`8.7e3`	`650`	`67`
Damping: loss factor	`0.04`	`0.04`	-	-	-
Damping: Rayleigh, α_{dM}	-	-	-	`0`	`0`
Damping: Rayleigh, β_{dK}	-	-	-	`0.14/omega0`	`0.46/omega0`

The material properties used in this model are partly made up, but aim to 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 fibre materials. The spider, acting as a spring, is made of a phenolic cloth with a much lower stiffness. The surround, finally, is a light resistive foam.

7 Select Subdomain 8 and click the **Load** tab. In the $\mathbf{F_z}$ edit field, type `Fe/Vol` to apply the electric force to the voice coil.

8 Click **OK**.

Boundary Conditions—Stress-Strain

Except for the symmetry boundary and those where the spider and the surround are attached to the case, all boundaries experience the pressure load from the surrounding air.

1 Choose **Physics>Boundary Settings**.

2 Press Ctrl+A to select all active boundaries.

3 In the **Group** list, select **Fluid load**.

4 Select Boundaries 58 and 62.

5 Set the **Constraint condition** to **Fixed**, then on the **Load** page change the load to `0` in both directions.

6 Select Boundary 3 and change the load to `0` in both directions.

7 Click **OK**.

Subdomain Settings—Pressure Acoustics

In the Pressure Acoustics application mode, only the air subdomains should be active. There is no need to set any material properties, since the defaults are those for air.

1. Choose **Multiphysics>Pressure Acoustics (acpr)**.
2. Choose **Physics>Subdomain Settings**.
3. Select all the solid subdomains: 3 and 6–13.
4. From the **Group** list (not the **Groups** page), select **Solid domain**.

   The subdomains that you have turned into solid domains will be deactivated from the pressure acoustics equation.
5. Select Subdomains 1 and 5. On the **PML** page, set the **Type of PML** to **Spherical** and select the **Absorbing in radial direction** check box.
6. Click **OK**.

Boundary Conditions—Pressure Acoustics

The same boundaries as experienced a pressure load in the Stress-Strain application mode should create a pressure wave due to the structural vibrations. The easiest way to select those boundaries is to briefly return to the Stress-Strain application mode and select them as a group.

1. Choose **Multiphysics>Axial Symmetry, Stress-Strain (acaxi)**.
2. Choose **Physics>Boundary Settings**.
3. Click the **Groups** tab and select the **Fluid** load group.
4. Click **Cancel** to leave the dialog box.
5. Without clicking in the geometry, choose **Multiphysics>Pressure Acoustics**.
6. Choose **Physics>Boundary Settings**.
7. With the same boundaries still selected, set the **Group** drop-list to **Structural acceleration**.
8. Select the active boundaries along the symmetry axis: 1, 2, 4, and 5, then in the **Boundary condition** list select **Axial symmetry**.

   All remaining exterior boundaries will use the default condition, **Sound hard boundary (wall)**. In order to compute the far-field sound pressure, you will also need a source boundary.
9. Select the **Interior boundaries** check box.
10. Select Boundary 70 and click the **Far-Field** tab.
11. Enter a new far-field variable with the **Name** `pfar`.

12 Click in the **Field** edit field, and the **Field** and **Normal derivative** entries will automatically be filled in.

13 Select the **z=0: Symmetric pressure** check box and click the **Full integral** radio button.

14 Click **OK**.

You have now supplied a source boundary encompassing all local sound sources and applied a symmetry plane to account for the infinite baffle. After computing the solution, you can evaluate the pressure in a point (r, z) by entering `pfar(r,z)`. To facilitate extraction of the sensitivity, define a new expression for the on-axis sound pressure level at a 1 m distance:

15 Choose **Options>Expressions>Scalar Expressions**.

16 Define a new variable with the **Name** `SPL` and the **Expression** `10*log10(0.5*pfar(0,1)*conj(pfar(0,1))/p_ref_acpr^2)`.

MESH GENERATION

In this computation, the air domain and the thin moving structures need to be well resolved.

1 Choose **Mesh>Free Mesh Parameters**.

2 On the **Global** page, click the **Reset to Defaults** button, then in the **Predefined mesh sizes** list select **Extra fine**.

3 Click the **Custom mesh size** button. Set both the **Element growth rate** and the **Resolution of narrow regions** to 2.

4 Click the **Subdomain** tab. Press Ctrl+A to select all subdomains, then set the **Method** to **Triangle**.

5 Click the **Remesh** button.

6 When the mesher has finished, click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu select the **Solver Manager**.

2 On the **Initial Value** page, click the **Store Solution** button.

3 In the dialog box that appears, all parameter values should be selected. Click **OK** to store all solutions.

4 In the **Values of variables not solved for and linearization point** area, select **All** from the **Parameter value** list.

5 On the **Solve For** page select **Axial Symmetry, Stress-Strain (acaxi)** and **Pressure Acoustics (acpr)**.

6 Click **OK** to close the **Solver Manager**.

7 Click the **Solve** button on the Main menu to compute the solution.

POSTPROCESSING AND VISUALIZATION

The plot still shows the induced currents, but now on a mesh that is not optimized for this analysis. Proceed to visualize the acoustic pressure field in the air.

1 Choose **Options>Suppress>Suppress Subdomains**. Select the PML subdomains (number 1 and number 5), then click **OK**.

2 Choose **Postprocessing>Plot Parameters**.

3 Click the **Surface** tab, and in the **Predefined quantities** list select **Pressure Acoustics (acpr)>Pressure**, or **Pressure Acoustics (acpr)>Sound pressure level** if you want to see the sound pressure measured in dB.

4 On the **General** page, use the **Parameter value** list to plot the pressure field for a few different frequencies; click **Apply** after making each selection to generate the plot. Figure 3-26 shows the sound pressure level at 3500 Hz.

5 When finished, click **OK**.

To view the sensitivity as a function of the pressure, execute the following instructions:

1 Choose **Postprocessing>Domain Plot Parameters**. On the **General** page click the **Title/Axis** button and select to use a **Log scale** for the **First axis**.

2 On the **Point** page select an arbitrary point, then in the **Expression** edit field enter `SPL`.

   Being defined as a scalar expression, the sensitivity needs to be evaluated somewhere in the geometry. No matter which point you choose, the result will still be that at a 1 m on-axis distance.

3 Click **Apply** to see the plot. You should see something similar to the image in Figure 3-27.

4 To see the total electric impedance, type `abs(Z)` in the **Expression** edit field and then click **Apply** again to reproduce Figure 3-28.

The impedance could also have been shown as a global plot.

Preparing for the Vented Loudspeaker Enclosure Model

The model "Vented Loudspeaker Enclosure" is an extension of the model that you are currently working on. It investigates how the sensitivity of this driver is affected by placing it in a bass-reflex enclosure. Because this by necessity becomes a large 3D model, it is important to consider the time and memory required to solve it, and, if needed, reduce them. The Vented Loudspeaker Enclosure model does this by modeling only the pressure acoustics. Its electromagnetic and mechanical properties are modeled as lumped parameters exported from this model. To extract these data, proceed as follows:

BLOCKED COIL IMPEDANCE

1 Choose **Postprocessing>Global Variables Plot**.

2 Select all entries in the **Quantities to plot** list, then click the **<** button.

3 From the **Predefined quantities** list, select **Blocked coil impedance (ohm)**.

4 Click the **>** button, then click **Apply**.

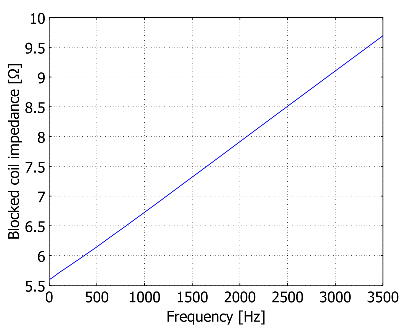
The plot that appears shows the real (resistive) part of the blocked coil impedance.

5 In the figure window, click the **ASCII** button to export the plot data.

6 Use the **Browse** button, select a suitable folder, and save the plot as `Rb.txt`.

7 Repeat the procedure to save the **Blocked coil inductance (H)** as `Lb.txt`.

Figure 3-30 shows the results.



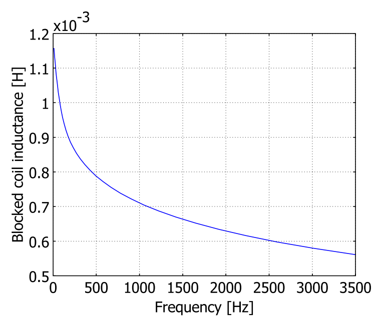


Figure 3-30: Blocked coil resistance (Ω)and blocked coil inductance vs. frequency (Hz).

MECHANICAL SUSPENSION IMPEDANCE

The mechanical suspension impedance is an idealized way of describing the suspension's mechanical behavior. It is defined as the complex-valued ratio between a force applied where the suspension would otherwise connect to the cone and the velocity at the same place. To isolate the mechanical suspension from the rest of the model, run a structural simulation of only the spider and the surround.

OPTIONS AND SETTINGS

1 Choose **Options>Constants**.

2 Add the following constant to the existing list; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
v0	1[m/s]	Velocity for suspension impedance simulation

PHYSICS SETTINGS

Subdomain Settings

1 Choose **Multiphysics>Axial Symmetry, Stress-Strain (acaxi)**.

2 Choose **Physics>Subdomain Settings** and select the solid subdomains that now should be excluded from the simulation: 3, 7, 8, and 10.

3 Clear the **Active in this domain** check box, then click **Apply**.

4 In the list of subdomains, verify that 9 and 13 are the only ones not grayed out, then click **OK** to close the dialog box.

Boundary Settings

1 Choose **Physics>Boundary Settings** and select the boundaries where the suspension connects to the cone, Boundaries 22 and 59.

2 In the **Constraint condition** list, select **Prescribed velocity**.

3 Select the $\mathbf{V_r}$ and $\mathbf{V_z}$ check boxes. For **Velocity r dir.**, keep the default 0. For **Velocity z dir.**, enter v0.

4 Click **OK** to close the dialog box.

From the earlier simulation, the outer perimeter of the suspension is still fixed.

COMPUTING THE SOLUTION

1 Choose **Solve>Solver Manager**.

2 On the **Solve For** page, select to solve only for **Axial Symmetry, Stress-Strain (acaxi)**.

3 On the **Initial Value** page, set **Values of variables not solved for and linearization point** to **Zero**.

Setting the value of the variables not solved for to zero means that you can keep the fluid load boundary condition and still not have any forces from the previous pressure acoustics solution acting on the suspension.

4 Click **OK** to close the dialog box.

5 Click the **Solve** button on the Main menu to compute the solution.

POSTPROCESSING AND VISUALIZATION

The plot now shows zero acoustic pressure everywhere. Follow the instructions below to show the suspension's deformation.

1 Choose **Postprocessing>Plot Parameters**, then click the **Surface** tab.

2 From the **Predefined quantities** list on the **Surface Data** page, choose **Axial Symmetry, Stress-Strain (acaxi)>Total displacement**. From the **Unit** list, choose **µm**.

3 Click the **Deform** tab, then select the **Deformed shape plot** check box.

4 From the **Predefined quantities** list, choose **Axial Symmetry, Stress-Strain (acaxi)>Displacement**.

5 On the **General** page, type 90 in the **Solution at angle (phase)** edit field.

6 Click **OK** to see the plot, which now shows the shape of the suspension at the time of maximal displacement (90° out of phase with the applied velocity). The plot should look like that in the picture below.

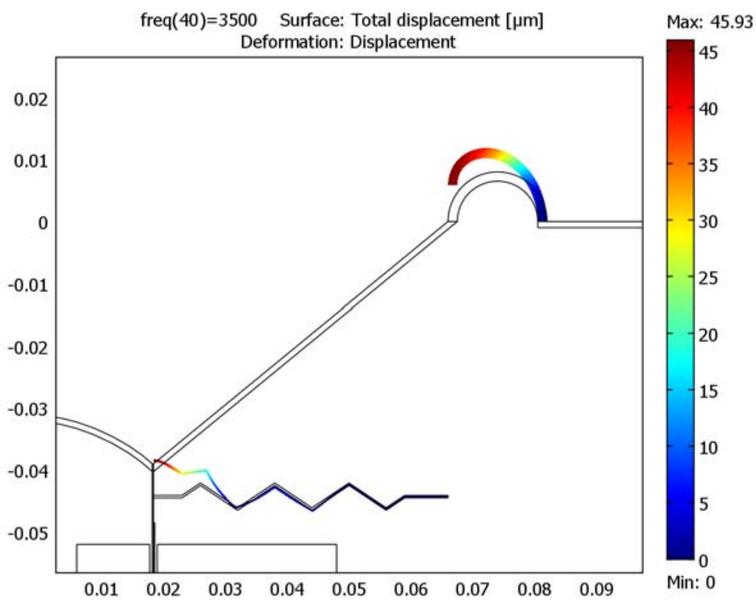


Figure 3-31: The deformed suspension at 3500 Hz.

Proceed to view and store the suspension resistance and compliance.

7 Choose **Postprocessing>Boundary Integration**.

8 In the **Expression to integrate** area, select **Axial Symmetry, Stress-Strain (acaxi)>Reaction Force, z-dir.** from the **Predefined quantities** list; the expression `RFz_acaxi` then appears in the **Expression** edit field with the unit **N** for newton displayed in the **Unit of integral** list.

The predefined boundary variables `RFr_acaxi` and `RFz_acaxi` provide a convenient way to compute accurate reaction forces. They are only defined in the node points of the finite element mesh and specifically designed to be used for postprocessing in the **Boundary Integration** dialog box.

9 Edit the **Expression** to read `real(RFz_acaxi/v0)`. This is the suspension's resistance. Verify that its unit is kg/s, which is the same as Ns/m.

10 Select the **Compute surface integral (for axisymmetric modes)** check box.

11 Click **Plot** to plot the resistance vs. frequency.

12 To obtain a logarithmic frequency scale, click the **X log** button in the figure window's toolbar.

Figure 3-32 shows the resulting plot. The title and axis labels in this figure have been modified in the **Edit Plot** dialog box, which you open by clicking the figure-window toolbar button with the same name.

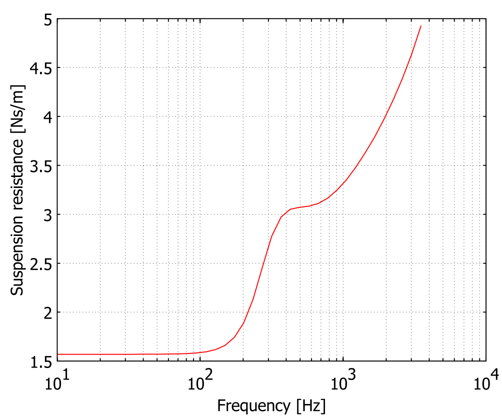


Figure 3-32: Suspension resistance (Ns/m) vs. frequency (Hz).

13 In the figure window, click the **ASCII** button to export the plot data.

14 In the **Save Current Plot in ASCII File** dialog box, click the **Browse** button.

15 In the **Save Data As** dialog box, select a suitable location, enter the **File name** `Rs.txt`, and then click **Save**.

16 Back in the **Save Current Plot in ASCII File** dialog box, the full file path now appears in the **Export to file** edit field. Click **OK** to export the plot in the default format, which gives a text file with a header followed by frequencies and the corresponding resistance values in two separate columns.

17 Return to the **Boundary Integration** dialog box, and change the **Expression** to `-1/(2*pi*freq*imag(RFz_acaxi/v0))`. This is the suspension's compliance with unit kg^2/m or, equivalently, m/N.

18 Click **Plot** to plot the compliance vs. frequency. Compare your result to Figure 3-33 (again, a logarithmic frequency scale, has been used).

19 Repeat Steps 13 through 16 to store the plot data in a text file, but this time use the file name `Cs.txt`.

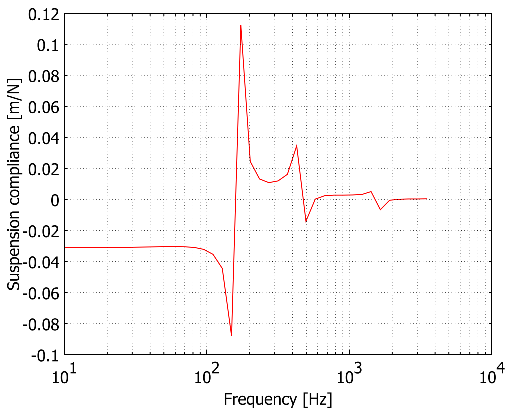


Figure 3-33: Suspension compliance (m/N) vs. frequency (Hz).

Loudspeaker Driver in a Vented Enclosure

Introduction

This example models the acoustic behavior of a loudspeaker driver mounted in a bass reflex enclosure.

One of the most important design parameters for a loudspeaker driver is its sensitivity as a function of the frequency. 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. 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 model in the Acoustics Module Model Library, called Loudspeaker Driver. This model borrows the electromechanical parameters from that example and shows how the enclosure affects the sensitivity.

The model uses the time-harmonic Pressure Acoustics application mode, available in the Acoustics Module, and a Global Equation connecting the driving voltage with the motion of the cone.

Model Definition

Figure 3-34 shows the geometry of the considered driver 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 3-35). The defining feature of this enclosure type is the vent, which in a properly designed enclosure acts to boost the sound at low frequencies.

As seen in Figure 3-36, the infinite baffle is now flush with the front wall of the enclosure. Figure 3-37 shows 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.

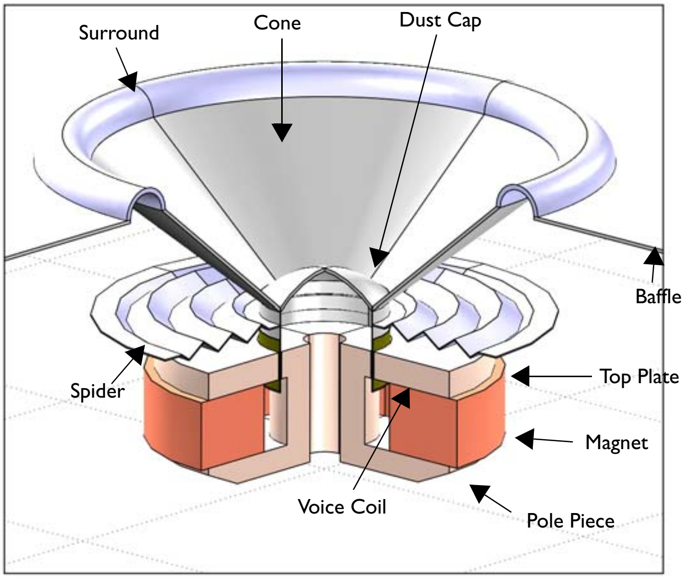


Figure 3-34: The driver, here set in an infinite baffle as in the Loudspeaker Driver model.

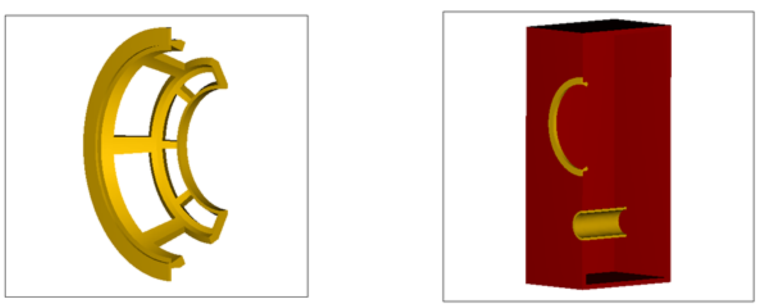


Figure 3-35: The frame and the vented enclosure.

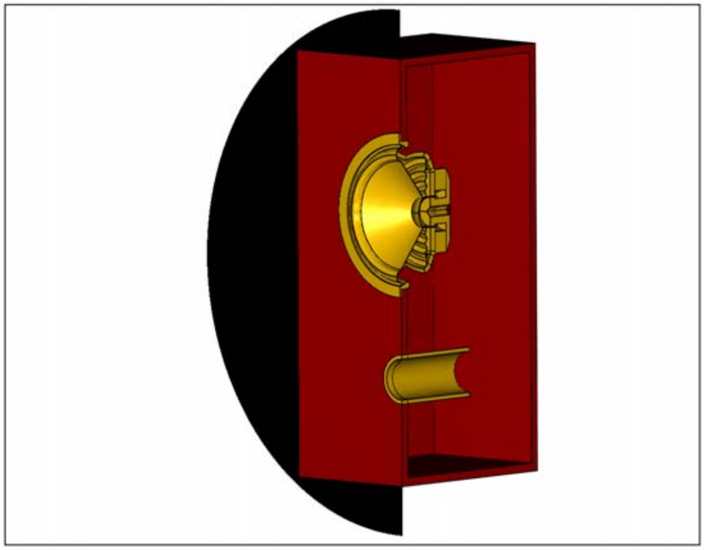


Figure 3-36: The geometry of the loudspeaker. The black half circle represents the baffle, which, in the model, forms a full half-plane extending to infinity.

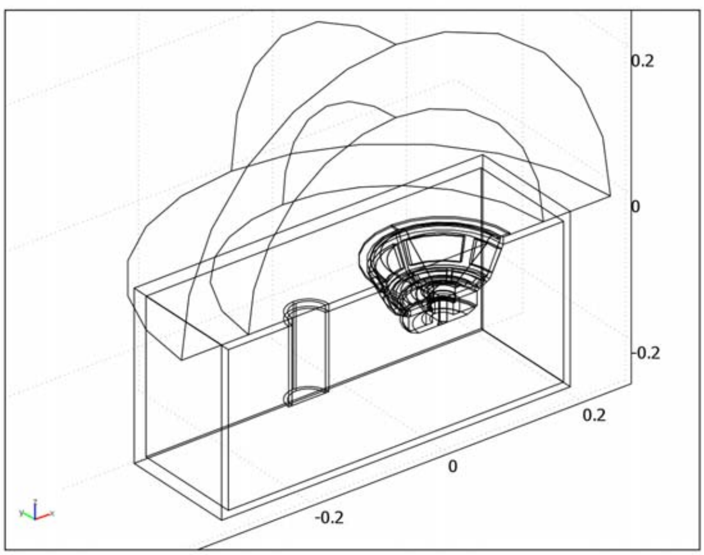


Figure 3-37: The complete model geometry. Thanks to the symmetry with respect to the xz-plane, the model consists of only one half of the speaker and the outside air.

The motion and deformation of the cone and the suspension as they interact with the sound pressure waves in the air should ideally be set up as a complete acoustic-structure interaction analysis, with the structural deformations computed locally. This approach is demonstrated in a number of models in the Model Library of COMSOL's Acoustics Module, including the Loudspeaker Driver.

This model however makes the assumption that the cone operates in piston-mode (that is as a perfectly rigid object). Furthermore, it approximates the motion of the spider and the surround by linear functions decreasing with the distance from where they attach to the cone, to where they are fixed to the immobile frame. These simplifications mean that the motion of all deforming parts can be described by just one scalar, here chosen to be the z-directed acceleration applied to the cone, a.

The equation of motion for the cone then reads:

$$ma = F_{\mathrm{e}} + F_{\mathrm{s}} + F_{\mathrm{a}}, \tag{3-6}$$

where m is the mass of the cone, F_{e} is the electric driving force, F_{s} is the reacting mechanical force from the suspension and spider, and F_{a} is the acoustic pressure force from the air. The forces are defined as follows:

$$\begin{aligned} F_{\mathrm{e}} &= \frac{\mathrm{BL}V_0}{Z_{\mathrm{b}}} - v\frac{(\mathrm{BL})^2}{Z_{\mathrm{b}}} \\ F_{\mathrm{s}} &= -vZ_{\mathrm{s}} = -v\left(R_{\mathrm{s}} + \frac{1}{j\omega C_{\mathrm{s}}}\right) \\ F_{\mathrm{a}} &= 2\oint_S p n_z ds \end{aligned} \tag{3-7}$$

Here, ω is the angular velocity and $v = a/(j\omega)$ is the cone velocity. BL is known as the 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. These parameters, as well as the mechanical resistance R_{s} and compliance C_{s}, forming the impedance Z_{s}, are imported from the Loudspeaker Driver model. While the impedances have been computed as complex-valued functions of the frequency, BL is a constant equal to 7.55 N/A. The documentation to the Loudspeaker Driver model ("Loudspeaker Driver" on page 145) contains a discussion of the terms in the expression for the electric force.

In the definition of F_{a}, the surface integral of the pressure, p, times the z-component of the normal (pointing outwards from the acoustics domain), is taken over both sides of the surface of the cone and suspension. The reason the suspension is included in the domain is that the suspension impedance was modeled in vacuum. The factor 2 accounts for the fact that only half of the total geometry is included in the model.

V_0, finally, is the 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 $V = V_0 \exp(i\omega t)$ with the amplitude $V_0 = 4$ V.

The model defines a in terms of the forces and V_0 as a function of F_e. It then solves a scalar equation adapting a so that $V_0 = 4$ V is fulfilled.

For an alternative view of the force balance on the cone, see the circuit model in Figure 3-38.

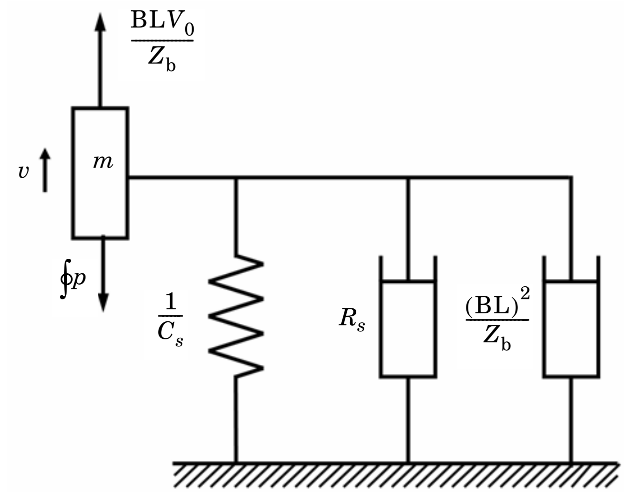


Figure 3-38: Circuit model of equations 3-6 and 3-7.

Results and Discussion

The model produces a sound pressure distribution inside and outside the enclosure for all frequencies solved for. As one example of how you can visualize the solution, see Figure 3-39, which shows the pressure at 1052 Hz as an isosurface plot. An alternative option is shown in Figure 3-40, where the sound pressure level in dB is plotted on a slice near the symmetry plane.

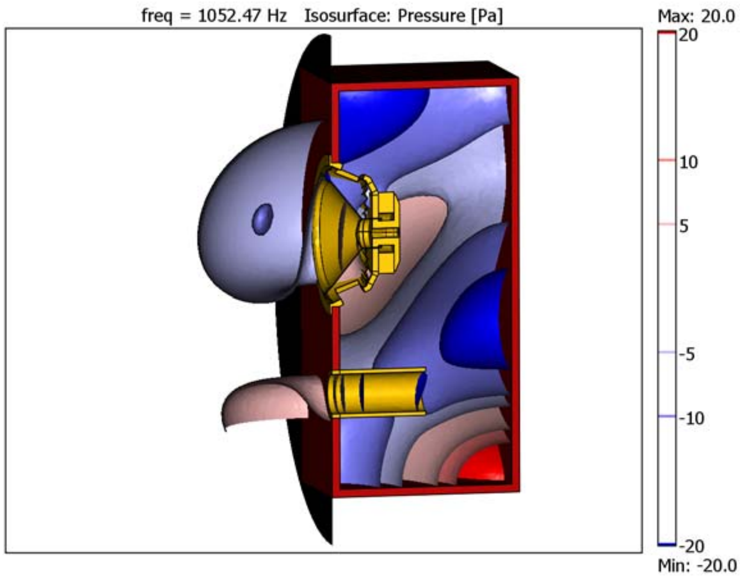


Figure 3-39: Isosurface plot of the sound pressure at 1052 Hz.

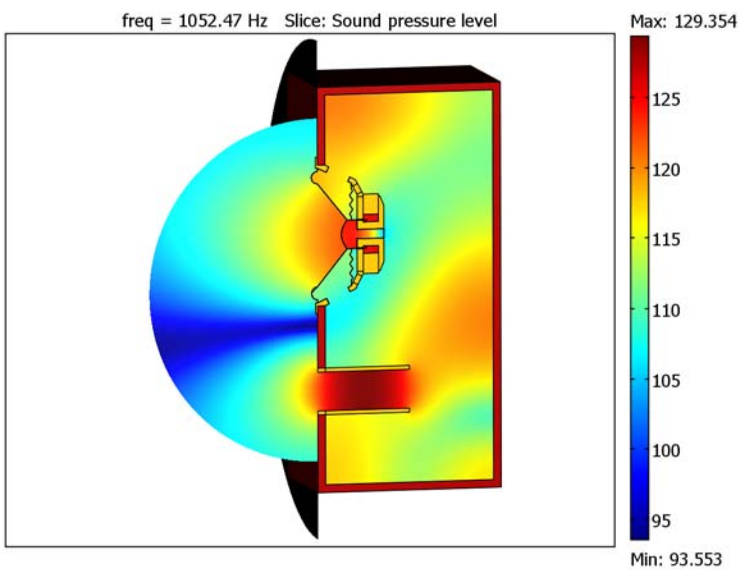


Figure 3-40: Slice plot of the sound pressure level at 1052 Hz.

Thanks to COMSOL's full integral far-field evaluation, you can evaluate the pressure not only inside the computational domain, but also anywhere outside the domain. This allows you to for instance plot the sound pressure level versus the elevation angle, or evaluate the directivity. The step-by-step instructions to this model show you how to plot the sensitivity (Figure).

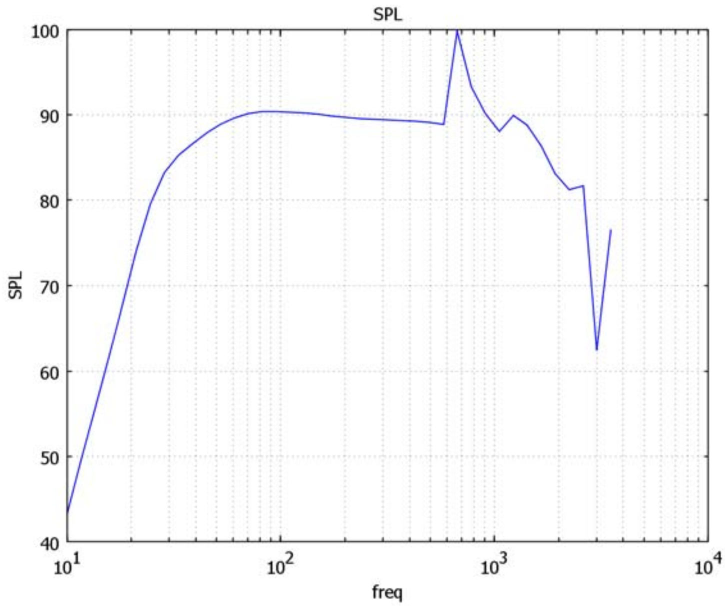


Figure 3-41: Loudspeaker sensitivity measured as the on-axis sound pressure level (dB) at a distance of 1 m from the unit. The pressure is evaluated using an RMS input signal of 2.83 V, corresponding to a power of 1 W at 8 Ω. Note the logarithmic scale for the frequency.

Compared to the sensitivity of the baffled driver in the Loudspeaker Driver model (Figure 3-27 on page 153), adding the enclosure clearly results in a "boost" for the lower frequencies, roughly the range between 30 and 100 Hz. On the flip side, it also causes an undesired peak at approximately 700 Hz, due to a resonance forming in the box. While you cannot completely avoid such resonances, loudspeaker manufacturers typically make them less prominent by careful positioning of the driver and the vent, as well as shaping of the vent and the enclosure. It is also common to line or partly stuff the enclosure with damping materials.

Modeling in COMSOL Multiphysics

This model requires the Acoustics Module and uses the time-harmonic version of the Pressure Acoustics application mode to solve for the acoustic pressure distribution. The frequency-dependent data from the Loudspeaker Driver model is imported by interpolation functions reading from text files.

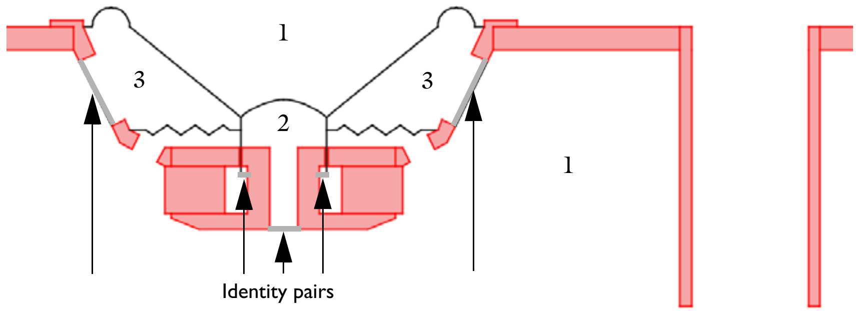


Figure 3-42: Cross-sectional view of the driver and vent. The parts, shown by numbers, are separated by black boundaries, representing pressure discontinuities, and gray identity pairs, across which the pressure is continuous.

In order to simplify the geometry, the cone and suspension are represented as boundaries. To allow for a discontinuous pressure across these boundaries, each boundary needs to constitute a border between two *parts* of an *assembly* (see "Using Assemblies" on page 413 in the *COMSOL Multiphysics Modeling Guide*). As shown in Figure 3-42, this implies that a total of 3 parts is required. The parts are joined with *identity boundaries* at the locations marked with arrows in the same figure, to make the pressure continuous in the air domains.

The model uses the parametric solver to sweep the frequency through a logarithmically spaced range of values from 10 Hz to 3.5 kHz.

Model Library path: `Acoustics_Module/Industrial_Models/vented_loudspeaker_enclosure`

MODEL NAVIGATOR

1 In the **Model Navigator** go to the **Space dimension** list and select **3D**.

2 In the list of application modes select **Acoustics Module>Pressure Acoustics**.

3 Click **OK**.

GEOMETRY MODELING

The geometry in this model can be created with COMSOL's drawing tools, but to save time, you will import it from a prepared file.

1 Choose **File>Import>CAD Data From File**.

2 Browse to the file `vented_loudspeaker_enclosure.mphbin`, which is located in your COMSOL Multiphysics installation folder under `models/Acoustics_Module/Industrial_Models`.

3 Click **Import** to import the geometry.

If you zoom in on the driver and click in the geometry, you will find that as depicted in Figure 3-42, it consists of three objects: one beneath the dustcap (CO2), one between the cone and the spider (CO3), and one for the rest of the geometry (CO1). The solid domains are part of CO1, but do not participate in the simulation and could in principle be excluded. In this model, they are kept in order to enable visualization of all parts of the speaker.

4 Choose **Draw>Use Assembly**.

You have now told the software to disconnect the solution between the objects. Proceed to create pairs reconnecting the objects only where the boundaries between them are not meant to represent physical barriers. Start with the boundaries between CO1 and CO2 (Pair 1 in Figure 3-43):

5 Choose **Physics>Identity Pairs>Identity Boundary Pairs**.

6 Click the **New** button. In the **Source boundaries** list, select the check boxes for Boundaries 99 and 118.

7 In the **Destination boundaries** list, select the check boxes for Boundaries 238 and 252.

Define Pair 2 between CO1 and CO2:

8 Click the **New** button. In the **Source boundaries** list, select the check boxes for Boundaries 108 and 110.

9 In the **Destination boundaries** list, select the check boxes for Boundaries 246 and 247.

Finally, create Pair 3 between CO1 and CO3:

10 Click the **New** button. In the **Source boundaries** list, select the check boxes for Boundaries 42, 53, 162, and 178.

11 In the **Destination boundaries** list, select the check boxes for Boundaries 186, 196, 227, and 233.

12 Click **OK** to close the dialog box.

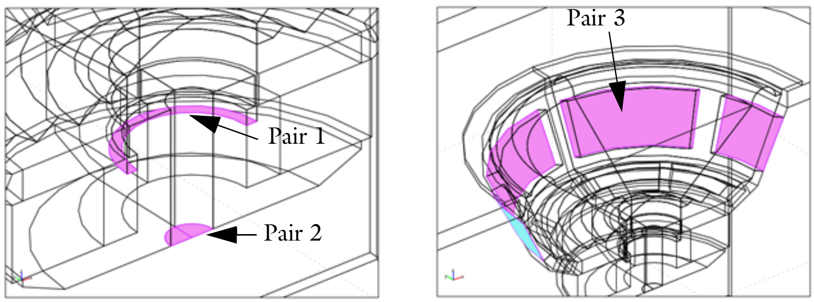


Figure 3-43: The pair boundaries.

OPTIONS AND SETTINGS

1 Choose **Options>Constants**. Enter the data given in this table (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
BL	7.55[N/A]	Force factor
m	0.017[kg]	Cone mass
su_a	66[mm]	Inner radius of surround
su_b	82[mm]	Outer radius of surround
sp_a	18.4[mm]	Inner radius of spider
sp_b	66[mm]	Outer radius of spider

The force factor is the one computed in the Loudspeaker Driver model. You will use the measures of the spider and surround in defining the linear decrease of the acceleration from their inner to their outer radii.

2 Choose **Physics>Scalar Variables**.

3 For the **Excitation frequency**, type `freq` in the **Expression** edit field; click **OK** to close the dialog box.

The following steps take you through importing the electric impedance of the blocked voice coil and the mechanical impedance of the suspension, both of them as computed for a range of frequencies in the Loudspeaker Driver model. Since you cannot interpolate from complex-valued data, you will import the real and imaginary constituents separately.

4 Choose **Options>Functions**. Click the **New** button.

5 In the **Function name** edit field, type `Rb`.

6 Click the **Interpolation** button and select to **Use data from File**.

7 Click the **Browse** button, browse to `Rb.txt` (in your COMSOL installation folder under `models/Acoustics_Module/Industrial_Models`) and click **Open**.

8 Click **OK**, then the **New** button; repeat the procedure for `Lb`, `Rs`, and `Cs`.

9 Click **OK** to close the dialog box.

10 Choose **Options>Expressions>Global Expressions**. Enter the data given in this table (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
omega	2*pi*freq	Angular frequency (rad/s)
Zb	Rb(freq)+j*omega*Lb(freq)	Blocked coil impedance (ohm)
Zs	Rs(freq)+1/(j*omega*Cs(freq))	Suspension impedance (Ns/m)
v	a/(j*omega)	Cone velocity (m/s)
Fs	-v*Zs	Force from suspension (N)
Fe	m*a-(Fs+Fa)	Driving electric force (N)
V0	Fe*Zb/BL+v*BL	Driving voltage (V)
r	sqrt(x^2+y^2)	Radial coordinate for scaling (m)
surround	(r-su_b)/(su_a-su_b)	Acceleration scaling on surround
spider	(r-sp_b)/(sp_a-sp_b)	Acceleration scaling on spider

The expressions for Z_b and Z_s reconstruct the blocked coil and suspension impedances by calling the interpolation functions containing your imported data. The velocity, force, and voltage expressions define the relation between the cone acceleration a and the driving voltage V_0, as described in the Model Definition. Note that F_a, the acoustic pressure force on the cone and suspension, is yet to be defined. The acceleration scaling expressions evaluate to 1 on the inner perimeters of the surround and spider, and decrease linearly to 0 on their outer perimeters.

PHYSICS SETTINGS

Global Equations

1 Choose **Physics>Global Equations**.

2 Create a new equation with the **Name** `a` and the **Expression** `V0-4`.

The equation you just entered means “adapt a so that $V_0 - 4$ V = 0 can be fulfilled.”

Subdomain Settings

1 Choose **Physics>Subdomain Settings**.

2 Select the solid subdomains, that is Subdomains 2 and 4–9. Clear the **Active in this domain** check box.

3 Select Subdomain 1 and click the **PML** tab.

4 From the **Type of PML** list, select **Spherical**.

5 Select the **Absorbing in r direction** check box.

6 For the **PML center point**, set $\mathbf{x_0}$ to `-0.07`.

7 Set the **PML scaling exponent** to `2`.

8 Click **OK** to close the dialog box.

The PML center point is set to coincide with the geometric center point of the PML domain. A scaling exponent of 2 means that the damping increases the further into the PML you go. This lowers the risk that any internal reflections due to insufficient mesh resolution substantially affect the solution.

Boundary Conditions

The first boundary setting you will apply is the acceleration condition on the spider. Start by selecting the boundaries constituting the spider:

1 Click the **Boundary Mode** button on the Main toolbar.

1 Click the **Go to YZ View** button on the Camera toolbar.

2 Click to deselect the **Orbit/Pan/Zoom** button on the Camera toolbar.

3 Use the **Zoom Window** button on the Main toolbar to zoom in on the driver; then draw a box to select the spider boundaries, as in Figure 3-44, which also shows an isometric view of the same selected boundaries.

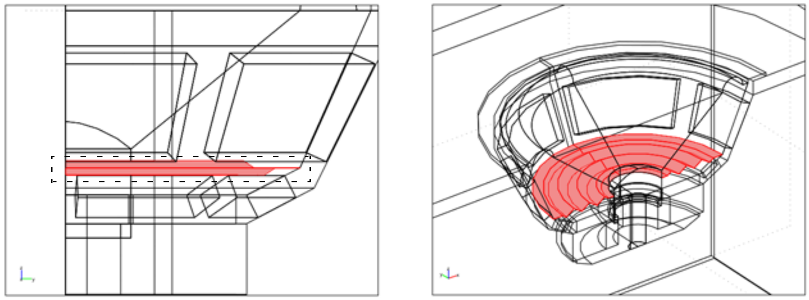


Figure 3-44: The spider boundaries, in the yz plane and in the default 3D view.

4 Choose **Physics>Boundary Settings**.

If you want to verify that you selected the right boundaries, you can scroll through the **Boundary selection** list. The selected boundaries should be 59, 66, 67, 75, 85, 88, 91, 92, 96, 122, 127, 131, 132, 134, 135, 141, 145, 146, 197, 200–207, 211, and 213–220. Note that because the spider constitutes the border between two parts of an assembly, it consists of two layers of boundaries, one for each part.

5 From the **Boundary condition** list, select **Normal acceleration**.

6 In the **Inward acceleration** text field, type `-a*nz*spider`.

The term "inward" here refers to the direction of a normal vector pointing from the boundary into its assembly part. The acceleration you applied is equal to the factor governing the linear decrease multiplied by $-\mathbf{a}\cdot\mathbf{n}$, where $\mathbf{a} = ae_z$. and $\mathbf{n}$ is the normal vector pointing outward from each part of the assembly.

7 By clicking in the geometry or picking from the **Boundary selection** list, select the boundaries on both sides of the surround: Boundaries 46, 48, 148, 150, 190, 192, 222, and 223. See Figure 3-45.

8 From the **Boundary condition** list, select **Normal acceleration**.

9 In the **Inward acceleration** text field, type `-a*nz*surround`.

10 By clicking in the geometry or picking from the **Boundary selection** list, select the boundaries on both sides of the cone and dust cap: Boundaries 61, 100, 111, 123, 198, 212, 240, and 248. See Figure 3-45.

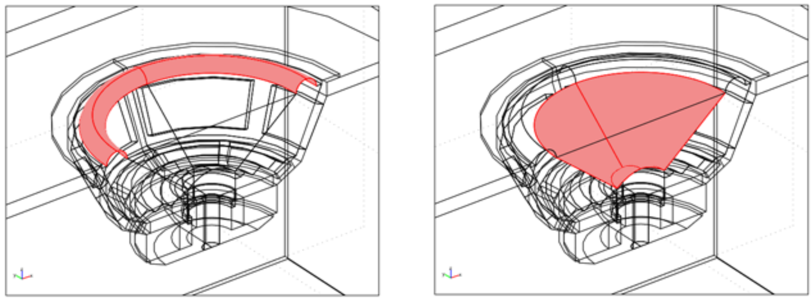


Figure 3-45: The boundaries of the surround (left) and cone and dust cap (right).

11 From the **Boundary condition** list, select **Normal acceleration**.

12 In the **Inward acceleration** edit field, type `-a*nz`.

The remaining boundaries will use the default **Sound hard boundary (wall)** condition. Proceed to specify the source boundaries for the far-field pressure computation.

13 Select the **Interior boundaries** check box.

14 Select Boundaries 16 and 55, then click the **Far-Field** tab.

15 Enter a new far-field variable with the **Name** `pfar`.

16 Click in the **Field** edit field, and the **Field** and **Normal derivative** entries will automatically be filled in.

17 Click the **Full integral** button.

18 Select the **y=0: Symmetric pressure** and **z=0: Symmetric pressure** check boxes.

19 Click **OK**.

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. After computing the solution, you can evaluate the pressure in a point (x, y, z) by entering `pfar(x,y,z)`. To facilitate extraction of the sensitivity, define a new expression for the on-axis sound pressure level at a 1 m distance:

20 Choose **Options>Expressions>Scalar Expressions**.

Define a new variable with the **Name** `SPL` and the **Expression** `10*log10(0.5*pfar(0,0,1)*conj(pfar(0,0,1))/p_ref_acpr^2)`, then click **OK**.

The variable p_ref_acpr is a reference level, defined from **Physics>Scalar Variables**. Per default, it is set to 20 μPa, which is the most commonly used value for air.

The z-component of the pressure force on the moving parts is given by an integral on all the moving boundaries. Now that you have applied the acceleration boundary conditions, it is easy to select these boundaries.

1 Open the **Boundary Settings** dialog box again, from **Physics>Boundary Settings**.
2 Click the **Groups** tab and select the groups where you have applied an acceleration; these groups are **unnamed3**, **unnamed4**, and **unnamed5**.
3 Click **Cancel** to close the dialog box without applying any further settings.
4 With the boundaries still selected, choose **Options>Integration Coupling Variables>Boundary Variables**.
5 Create a new boundary integration variable with the **Name** Fa and the **Expression** 2*p*nz.
6 Click **OK**.

MESH GENERATION

For the solution to properly connect across the identity pairs, the mesh on the destination boundary cannot be coarser than that on the source boundary. The optimal way to mesh an identity pair is to use the Copy mesh feature to get an identical mesh on both boundaries. This can only be done sequentially, two boundaries at a time. The effort pays off however, as the result is a mesh that allows an iterative solver to converge. Compared to a direct solver, the iterative solver operates slightly faster and uses much less memory.

The other thing to keep in mind when you mesh this model is that you need to resolve the wave at all frequencies, with at least 5 to 6 elements per wavelength to obtain a good accuracy. The highest frequency you will solve for is 3.5 kHz.

1 Choose **Mesh>Free Mesh Parameters**.
2 In the **Predefined mesh sizes** list select **Coarse**, then click the **Custom mesh size** button.
3 In the **Maximum element size** edit field, type 343/3500/4.5. The next step in the instruction will guide you through creating the boundary mesh, so make sure to keep the dialog box open and do not generate the mesh just yet.

   This setting will give you a mesh size less than or equal to the wavelength at 3.5 kHz divided by 4.5. Due to the exact definition of mesh size and the way the mesh elements are distributed, you will in practice get around 6 mesh elements per wavelength at this frequency.

Note: With the maximum element size that you just applied, the solution process will use roughly 1 GB of memory.

4 On the **Boundary** page, select all source boundaries in the **Boundary selection** list: Boundaries 42, 53, 99, 108, 110, 118, 162, and 178.

5 Click the **Mesh Selected** button.

6 In the **Boundary selection** list, select only Boundaries 42 and 186.

7 With the **Free Mesh Parameters** dialog box still open, click the **Copy Mesh** button on the Mesh toolbar.

8 Repeat the procedure for the following pairs of boundaries: 53 and 196, 99 and 238, 108 and 246, 110 and 247, 118 and 252, 162 and 227, 178 and 233.

9 Click **OK**, then click the **Mesh remaining (free)** button on the Mesh toolbar to generate the mesh.

COMPUTING THE SOLUTION

1 From the **Solve** menu choose **Solver Parameters**.

2 In the **Solver** list, select **Parametric**.

3 In the **Parameter names** edit field, type `freq`.

4 In the **Parameter values** field, type `logspace(1,log10(3500),40)`.

This gives you a logarithmically spaced list of 40 frequencies between 10 Hz and 3.5 kHz.

Note: Depending on the specifications of your computer, the solution may take up to a few minutes per frequency. If you want to save time, you can choose to solve for fewer frequencies by entering a number less than 40.

5 From the **Linear system solver** list, select **GMRES**.

6 From the **Preconditioner** list, select **Geometric multigrid**.

7 Click the **Settings** button.

8 In the tree structure, select **Coarse solver**.

9 From the **Coarse solver** list, select **PARDISO**.

10 Click **OK** twice to close both open dialog boxes.

11 Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

The default plot shows the pressure distribution on five equidistant slices parallel with the *yz*-plane, at 3500 Hz. To get a better view, try excluding the PML and plotting the sound pressure level on a slice near the symmetry boundary:

1. Choose **Options>Suppress>Suppress Subdomains**.
2. Select Subdomain 1 and click **OK**.
3. Choose **Postprocessing>Plot Parameters** and click the **Slice** tab.
4. From the list of **Predefined quantities** select **Sound pressure level**.
5. Under **Slice positioning**, set **Number of x levels** to `0`.
6. For **y levels**, click the option button under **Vector with coordinates** and enter `1e-3`.
7. Click **OK** to see the plot, which should look like Figure 3-46.

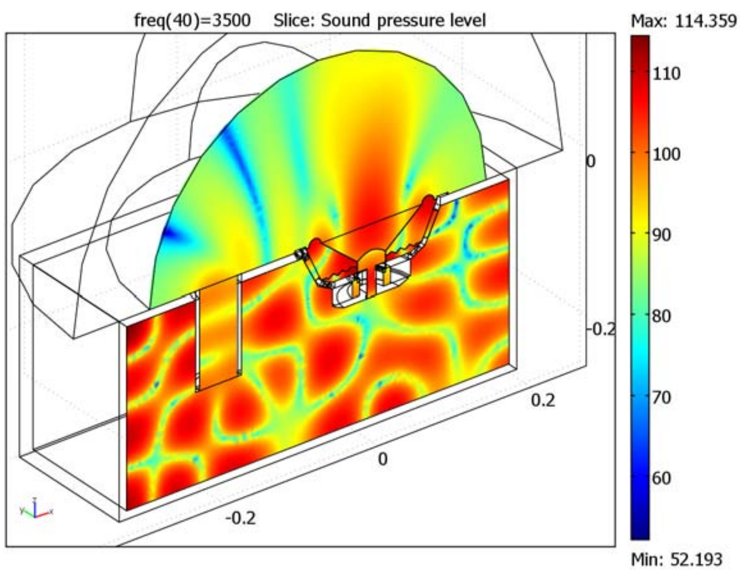


Figure 3-46: Sound pressure level (dB) at 3500 Hz.

8. To study the sensitivity, choose **Postprocessing>Domain Plot Parameters** and click the **Point** tab.
9. In the **Expression** edit field, enter `SPL`.

10 Select Point 1 and click **OK**.

11 Click the **x log** button to get logarithmically spaced frequencies and reproduce Figure 3-41.

Note: If you would like to compare the sensitivity in this model with that of the driver in an infinite baffle, keep the figure window open while opening the Loudspeaker Driver model from the Model Library. On the **General** page of the **Domain Plot Parameters** dialog box in that model, select to **Keep current plot** and **Plot in Figure 1** before plotting `SPL`. Now both sensitivity plots will appear.

Figure 3-39 shows the pressure distribution as isosurfaces at 1052 Hz. The boundaries are yellow on the driver and vent, red on the enclosure, and black on the infinite baffle. To find out which plot parameters were used in creating this plot, you can open the model from the Model Library and choose **Postprocessing>Plot Parameters**. Note that the expression `bndcol`, which is used on the **Boundary** page, was defined from **Physics>Expressions>Boundary Expressions** to take on different values on the different boundaries. Note also that certain edges have been suppressed, using **Options>Suppress>Suppress Edges**.

Muffler with Perforates

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.

Introduction

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 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 in this model 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 a recent 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.

A schematic cross-section of the muffler geometry is depicted in Figure 3-47.

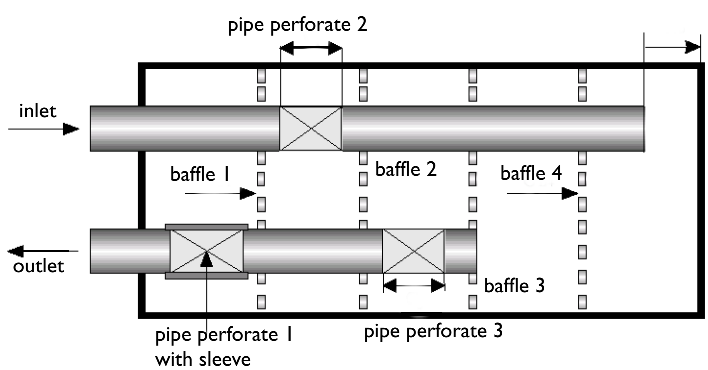


Figure 3-47: Muffler geometry cross section.

The detailed design and dimensions of the outlet pipe and the four baffles (as seen from the right in Figure 3-47) are given in Figure 3-48 through Figure 3-51.

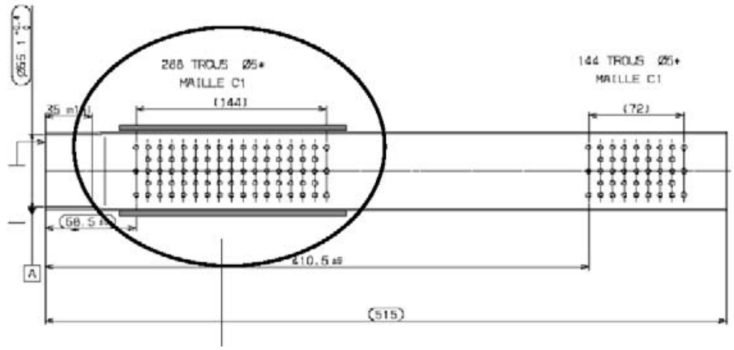


Figure 3-48: Outlet pipe. A stainless steel sleeve is located above the left perforated section with 288 holes. The other two pipe perforates contain 144 holes each.

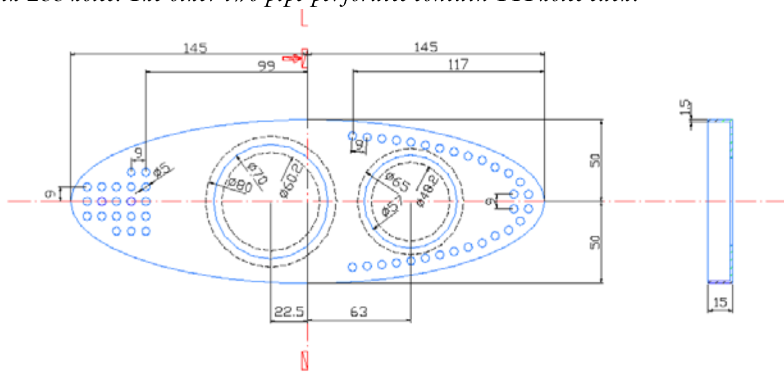


Figure 3-49: Baffle number 1, outlet side to the left and inlet side to the right.

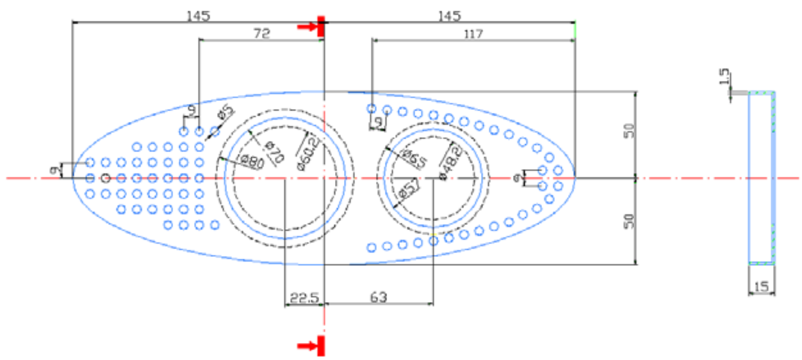


Figure 3-50: Baffles number 2 and 3.

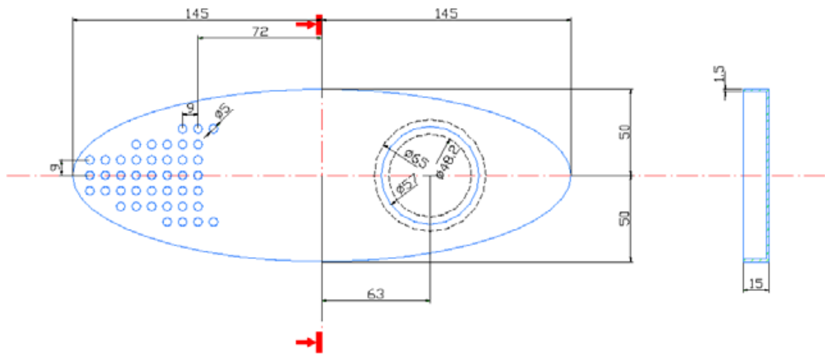


Figure 3-51: Baffle number 4.

The model geometry is supplied as a CAD file. To reduce computation time only the upper half of the muffler is included; imposing the homogeneous Neumann sound-hard boundary condition at $z = 0$ accounts for the reflection symmetry in the central horizontal plane through the muffler. (A careful inspection of the drawings in Figure 3-49–Figure 3-51 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 3-52 where the perforated regions have been shaded for emphasis.

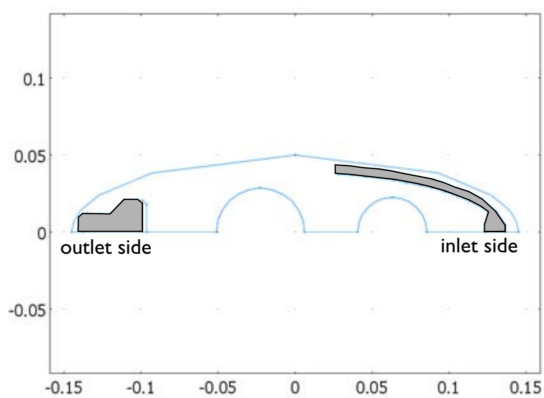


Figure 3-52: Perforated regions of baffle number 1.

You model the acoustic effects of the perforates by applying the Acoustics Module's lumped perforated-plate impedance pair boundary condition on these regions. The semiempirical expression for the impedance, Z, then reads (Refs 1–2):

$$\frac{Z}{\rho c_s} = \frac{1}{\sigma}(\theta + i\chi) + \theta_f$$

Here σ denotes the *area porosity*, that is, the fraction of a model-geometry perforate region that is covered by holes in the real muffler geometry. Furthermore, θ is the specific resistance and χ the specific reactance, given by

$$\theta = \sqrt{\frac{8\mu k}{\rho c_s}}\left(1 + \frac{t_p}{d_h}\right) \quad \text{and} \quad \chi = k(t_p + \delta_h)$$

where μ is the dynamic viscosity, k is the wave number, ρ is the density, c_s the speed of sound, t_p is the plate thickness, d_h is the hole diameter, and δ_h is the end correction. The default expression for the end correction is $0.25d_h$.

Finally, the term θ_f allows you to specify additional contributions to the specific impedance, for example the resistance caused by a mean flow in the muffler. Different models have been proposed for such a flow resistance; for a discussion and further references, see Refs 3–4. In this model, you use θ_f to include the effects of the metallic sleeve above pipe perforate number 1.

The relevant input parameters for the model are listed in Table 3-2. 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 3-2: 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
ρ	1.25 kg/m^3	Density of air
c_s	343 m/s	Speed of sound
μ	$1.8\cdot10^{-5}$ Pa·s	Dynamic viscosity

The wave number, k, is given by $2\pi f/c_s$ where f denotes the frequency. You run the simulation for a range of frequencies between 20 Hz and 600 Hz.

Results and Discussion

The transmission loss in the muffler is defined as

$$\mathrm{TL} = 10\log_{10}\left(\frac{P_{\mathrm{in}}}{P_{\mathrm{out}}}\right) \tag{3-8}$$

where P_{in} and P_{out} denote the total acoustic power at the inlet and the outlet, respectively. Figure 3-53 displays the Acoustics Module modeling results for the transmission loss as a function of sound frequency together with experimentally measured data.

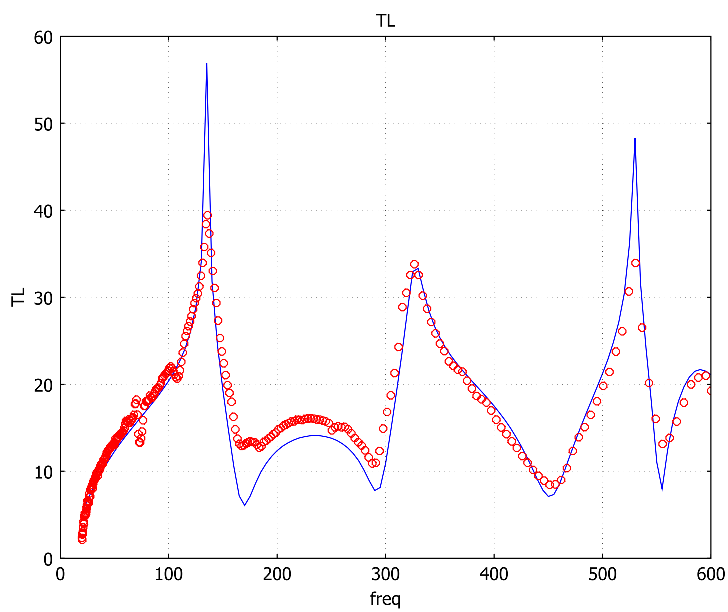


Figure 3-53: Transmission loss versus frequency.

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.

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 3-54 display this field for the frequencies 530 Hz and 555 Hz, respectively; as Figure 3-53 shows, the former frequency corresponds to a local maximum for the transmission loss whereas the latter gives a local minimum. In Figure 3-54 you can see these how these properties are related to the sound pressure level distributions near the muffler inlet and outlet.

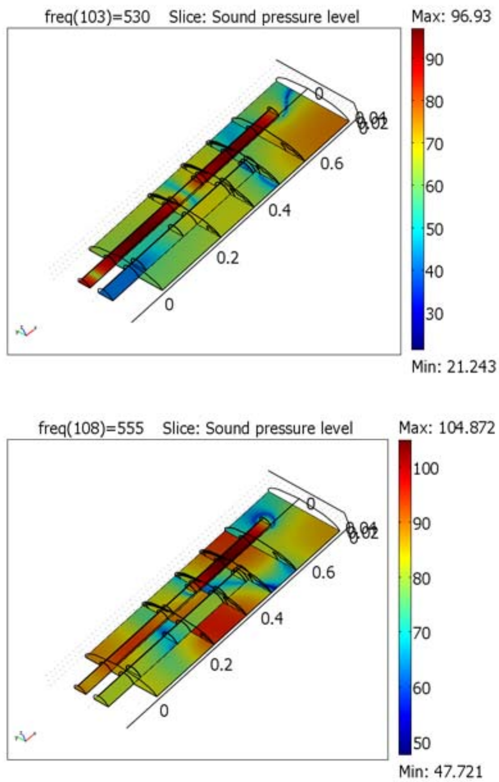


Figure 3-54: Sound pressure level distributions at 530 Hz (top) and 555 Hz.

References

1. A. Bauer, "Impedance Theory and Measurements on Porous Acoustic Liners," *Journal of Aircraft*, vol. 14, no. 8, pp. 720–728, 1977.

2. 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.

3. 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.

4. 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.

Model Library path: `Acoustics_Module/Industrial_Models/perforated_muffler`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **3D** from the **Space dimension** list.
2 From the list of application modes select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis**.
3 Click **OK**.

GEOMETRY MODELING

A horizontal symmetry plane through the muffler means that it is sufficient to model only half of the geometry. This geometry is available as a CAD file that comes with the Acoustics Module. To import the geometry, perform the following steps:

1 From the **File** menu select **Import>CAD Data From File**.
2 Browse to the folder `Models>Acoustics Module>Industrial Models`.
3 Select the file `perforated_muffler.mphbin`, then click **Import**.

   When the import procedure has finished, you should see the upper half of a muffler in the drawing area. At this point, the geometry consists of a number of composite objects; to allow you to impose the appropriate boundary conditions on the baffles and the interior parts of the pipes, you need to turn it into an assembly and create boundary pairs.
4 From the **Draw** menu, select **Use Assembly**.
5 Select all objects in the drawing area by pressing Ctrl+A, then click the **Create Pairs and Imprints** button on the Draw toolbar.

This completes the geometry-modeling stage. After repeated clicks on the **Increase Transparency** button on the Camera toolbar, the geometry in the drawing area of the user interface on your screen should look like that in the Figure 3-55. (To return to the return to the default visualization settings, click the **Decrease Transparency** button an equal number of times.

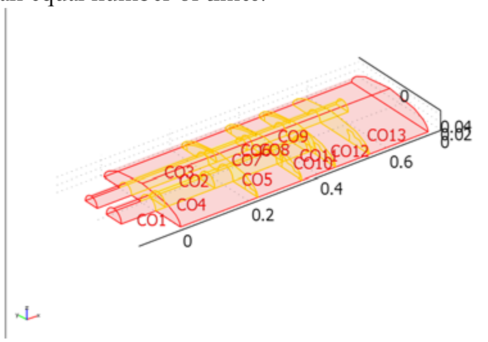


Figure 3-55: The muffler geometry as an assembly.

OPTIONS AND SETTINGS

Constants

1 From the **Options** menu choose **Constants**.

2 Enter the names, values, and descriptions listed in the table. When finished, click **OK**.

NAME	VALUE	DESCRIPTION
p0	1[Pa]	Inlet pressure
t_w	1.5[mm]	Wall thickness
d_h	5[mm]	Hole diameter
sigma_p	0.24	Porosity, pipe perforates
sigma_bi	0.35	Porosity, baffle panels on inlet side
sigma_bo	0.30	Porosity, baffle panels on outlet side
freq	20[Hz]	Frequency

The frequency value in this table is the initial frequency in the range of 20–600 Hz for which you later solve the model using the parametric solver.

Integration Coupling Variables

To determine the transmission loss in the muffler you first need to calculate the total acoustic power at the inlet and at the outlet. To this end, define two integration coupling variables as follows:

1 Choose **Options>Integration Coupling Variables>Boundary Variables**.
2 On the **Source** page, select Boundary 8 (the inlet).
3 Define the surface integral

$$P_{\text{in}} = 2\int_{S_5} \frac{p_0^2}{2\rho c_{\text{s}}}$$

as an integration coupling variable by typing `P_in` in the **Name** edit field and `2*p0^2/(2*rho_acpr*cs_acpr)` in the **Expression** edit field of the first row of the table. Leave the default settings for the **Integration order** and the **Global destination**.
4 Select Boundary 1 (the outlet).
5 Now define the integral

$$P_{\text{out}} = 2\int_{S_1} \frac{|p|^2}{2\rho c_{\text{s}}}$$

by typing `P_out` in the **Name** edit field and `2*p*conj(p)/(2*rho_acpr*cs_acpr)` in the **Expression** edit field of the second row of the table. As above, leave the **Integration order** and **Global destination** settings at their default values.

Global Expressions

Next, make the transmission loss (see Equation 3-8) available at the postprocessing stage as a global expression:

1 Choose **Options>Expressions>Global Expressions**.
2 Define a global expression by the **Name** `TL` and the **Expression** `10*log10(P_in/P_out)`.
3 Click **OK**.

PHYSICS SETTINGS

Identity Pairs

Before applying the boundary conditions, you need to create separate pairs for the perforated regions of the baffles.

1 Choose **Physics>Identity Pairs>Identity Boundary Pairs**.
2 In the **Identity pairs** list select **Pair 8**.
3 In the **Source boundaries** list, clear the check boxes for Boundaries 32 and 33.
4 With the check box for Boundary 31 still selected click the **Select Source** button.

5 In the **Destination boundaries** list, clear the check boxes for Boundaries 51 and 56.

6 With the check box for Boundary 48 still selected click the **Select Destination** button.

7 Click the **New** button.

In the **Name** edit field it now reads Pair 26; keep this entry as it is.

8 In the **Source boundaries** list select Boundary 32, then click the **Check Selected** button below the list. (Alternatively, select the check box for Boundary 32 directly.)

9 Click the **Select Source** button.

10 In the **Destination boundaries** list select Boundary 51, then click the **Check Selected** button below the list.

11 Click the **Select Destination** button.

12 Define Pair 27 by repeating Steps 7 through 11, but use Boundary 33 as the source and Boundary 56 as the destination.

13 Repeat this procedure (Steps 2–12) for the remaining baffles and the inner ends of the pipes using the data in the table below; already existing pairs (to be modified in analogy with Pair 8 above) are marked with an asterisk. When done, click **OK**.

PAIR	SOURCE BOUNDARIES	BOUNDARIES TO CLEAR	DESTINATION BOUNDARY	BOUNDARIES TO CLEAR
*15	64	65, 66	83	86, 91
28	65		86	
29	66		91	
*22	99	100, 101	114	117, 121
30	100		117	
31	101		121	
*25	126	127	128	131
32	127		131	
*24	111, 112	113	118, 119	125
33	113		125	
*19	80, 81	82	133, 134	135
34	82		135	

Boundary Conditions

1 From the **Physics** menu select **Boundary Settings**.

2 Select Boundary 8 (the inlet). From the **Boundary condition** list select **Radiation condition**. In the **Pressure source** edit field type `p0`.

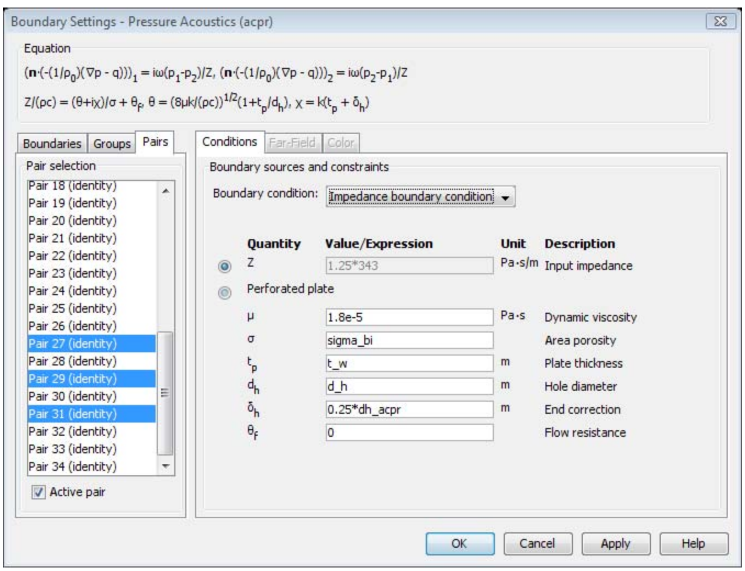
3 Select Boundary 1 (the outlet). From the **Boundary condition** list select **Radiation condition**. For this boundary, keep the default value (0) in the **Pressure source** edit field.

4 Click the **Pairs** tab.

5 Define boundary conditions for the pairs according to the following table (for pairs and parameters not explicitly mentioned, leave the default settings):

SETTINGS	PAIRS 1, 3, 4, 7, 8, 10, 11, 14, 15, 17–19, 21, 22, 24, 25	PAIR 6	PAIRS 13, 20	PAIRS 26, 28, 30, 32	PAIRS 27, 29, 31
Type	Sound hard boundary (wall)	Impedance boundary condition, Perforated plate			
σ		sigma_p	sigma_p	sigma_bo	sigma_bi
θ_f		1			
t_p		t_w	t_w	t_w	t_w
d_h		d_h	d_h	d_h	d_h

The setting $\theta_f = 1$ for Pair 6 represents the impedance contribution ρc_s from the metallic sleeve outside the perforated pipe section closest to the outlet.



6 Click **OK**.

APPLICATION SCALAR VARIABLES

1 From the **Physics** menu select **Scalar Variables**.

2 In the **Expression** edit field for **freq_acpr** type `freq`.

3 Click **OK**.

GENERATING THE MESH

Because boundary conditions are defined on the destination boundary in a pair, you obtain the best numerical result if the mesh on the destination boundary is finer than that on the source boundary. With the following steps you specify a finer mesh on the destination boundaries for pairs with an impedance boundary condition. For the pairs using the sound-hard boundary condition there is no connection between the parts. Therefore, there is no reason to use different mesh sizes on the two parts of those pairs.

1 From the **Mesh** menu select **Free Mesh Parameters**.

2 Go to the **Boundary** page and select Boundaries 35, 37, 51, 56, 68, 70, 86, 91, 103, 105, 117, 121, and 131. Set the **Maximum element size** to `0.012`.

3 Click **OK**.

4 Click the **Initialize Mesh** button on the Main toolbar.

For this model, the limiting factor when determining the mesh size is resolving the geometry details—not the smallest wave length, which is roughly 57 cm.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 In the **Solver** list select **Parametric**.

3 In the **Parameter** area on the **General** page, type `freq` in the **Parameter name** edit field and type `20:5:600` in the **Parameter values** edit field.

   This setting gives a frequency sweep from 20 Hz to 600 Hz in steps of 5 Hz. This suffices to resolve the main features in the muffler's frequency response. However, it leads to a solution time on the order of 100 minutes. If you want to reduce the solution time you can either reduce the range or increase the step size.

4 Click **OK**.

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot is a slice plot of the acoustic pressure field for the final frequency in the sweep. For many purposes, visualizing the sound pressure level field is more informative.

1 Click the **Plot Parameters** button on the Main toolbar.

2 Click the **Slice** tab. From the **Predefined quantities** list, select **Sound pressure level**.

3 In the **Slice positioning** area, type 0 in the **Number of levels** edit fields for **x levels** and **y levels**. For **z levels**, click the option button next to the **Vector with coordinates** edit field and enter the level `1e-4`.

4 Click **Apply** to generate a plot of the sound pressure level distribution near the muffler's symmetry plane.

   To include information about the sound pressure level away from this symmetry plane, add some isolevel curves:

5 On the **Isosurface** page, select the **Isosurface plot** check box.

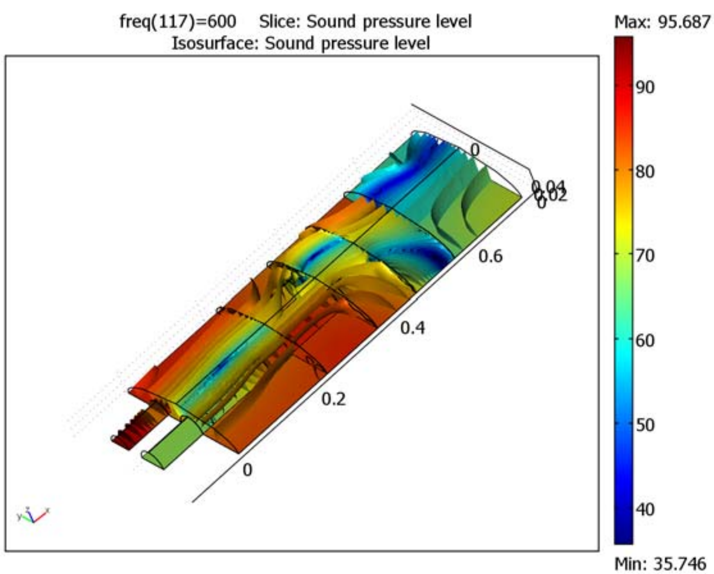
6 On the **Isosurface Data** page, select **Sound pressure level** from the **Predefined quantities** list.

7 In the **Isosurface levels** area, type 25 in the **Number of levels** edit field.

8 In the **Isosurface color** area, clear the **Color scale** check box.

9 Click **Apply**.

After clicking the **Headlight** button on the Camera toolbar, the plot in the drawing area should resemble that in the figure below.



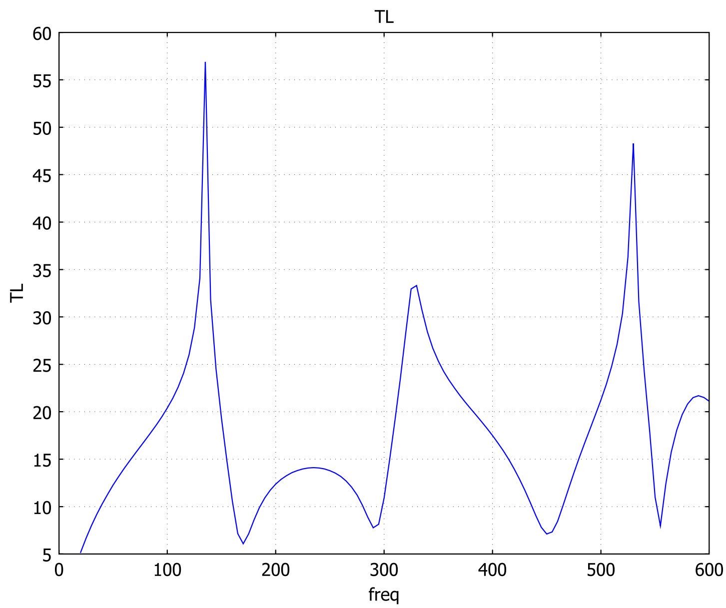
To generate the plot of transmission loss vs. frequency displayed by the solid blue line in Figure 3-53, follow these instructions:

1 From the **Postprocessing** menu select **Cross-Section Plot Parameters**.

2 Click the **Point** tab, then in the **Expression** edit field type `TL`.

   Because you are plotting a global expression, the choice of point is irrelevant. Therefore, leave the default plot-point coordinates.

3 Click **OK** to generate the graph displayed in the following figure.



The plots in Figure 3-54, comparing the sound pressure level fields for the local transmission-loss maximum near 530 Hz and the corresponding local minimum near 555 Hz, are obtained as follows:

1 Click the **General** tab in the **Plot Parameters** dialog box.

2 In the **Plot type** area, clear the **Isosurface** check box.

3 In the **Solution to use** area, select **530** from the **Parameter value** list.

4 Click **Apply** to generate the upper plot in Figure 3-54 in the drawing area.

   The contrast in color near the inlet and outlet reflects the high transmission-loss value. Compare this plot to the corresponding one at the frequency 555 Hz:

5 In the **Solution to use** area, select **555** from the **Parameter value** list.

6 To keep the previous plot for comparison, select **New figure** from the **Plot in** list.

7 Click **Apply** to generate the lower plot in Figure 3-54.

   In this case, the color contrast between inlet and outlet is significantly less marked. Indeed, as the transmission loss vs. frequency curve shows, the transmission loss at 555 Hz is below 10 dB compared to roughly 48 dB at 530 Hz.

The plot displayed in the drawing area when you open the model is generated with the following commands:

1 On the **General** page of the **Plot Parameters** dialog box, select **600** from the **Parameter value** list.

2 In the **Plot type** area, select the **Streamline** check box, then click the **Streamline** tab.

3 From the **Predefined quantities** list inside the **Streamline data** area, select **Intensity**.

4 From the **Streamline plot type** list, select **Magnitude controlled**.

5 On the **Density** page, set the **Min distance** to `0.03` and the **Max distance** to `0.04`.

6 On the **Line Color** page, select the **Uniform color** option button, then click the **Color** button.

7 From the palette, select a yellow color, then click **OK**.

8 From the **Line type** list select **Tube**, then click the **Tube Radius** button.

9 Clear the **Auto** check box for the **Radius scale factor**, then type `0.3` in the associated edit field. Click **OK** to close the **Tube Radius Parameters** dialog box.

10 From the **Tube resolution** list, select **Low**.

11 Click **OK** to generate the plot and close the **Plot Parameters** dialog box.

Postprocessing with COMSOL Script/MATLAB

The experimental data visualized with red rings in Figure 3-53 is included with the Acoustics Module. If you have a license for COMSOL Script or MATLAB, you can add these data to the transmission-loss plot, thereby completing the reproduction of Figure 3-53, as follows.

1 First, if you are running COMSOL via a server, choose **File>Client/Server/MATLAB>Disconnect from Server/MATLAB**. Click **Yes** to confirm the closure of the server connection. This step allows you to open COMSOL Script on your local desktop.

2 Choose **File>COMSOL Script** (or **File>Client/Server/MATLAB>Connect to MATLAB**).

Next, import the data:

3 In the **COMSOL Script** window that opens, use the commands `pwd` and `cd` to navigate to the root directory of your COMSOL installation and from there to the directory `models/Acoustics_Module/Industrial_Models`.

4 At the command-line prompt, type `perforated_muffler_exp_data` to load the data.

5 Type `who` to inspect the variables in your COMSOL Script workspace.

The list displayed should include the variable `TLexp`. This is a 296-by-2 matrix whose first and second columns contain a set of frequencies in the range 20–600 Hz and the corresponding transmission-loss measurements, respectively.

6 Finally, plot the data in the open figure window by executing the following commands:

```
hold on
plot(TLexp(:,1),TLexp(:,2),'linestyle','none',...
     'marker','o','color','r')
```

SAW Gas Sensor

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) in air. This causes a shift in resonance to a slightly lower frequency.

Model Definition

Figure 3-56 shows a conceptual view of the gas sensor in this model. 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 3-57. 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. In the model, this boundary is fixed to a zero displacement.

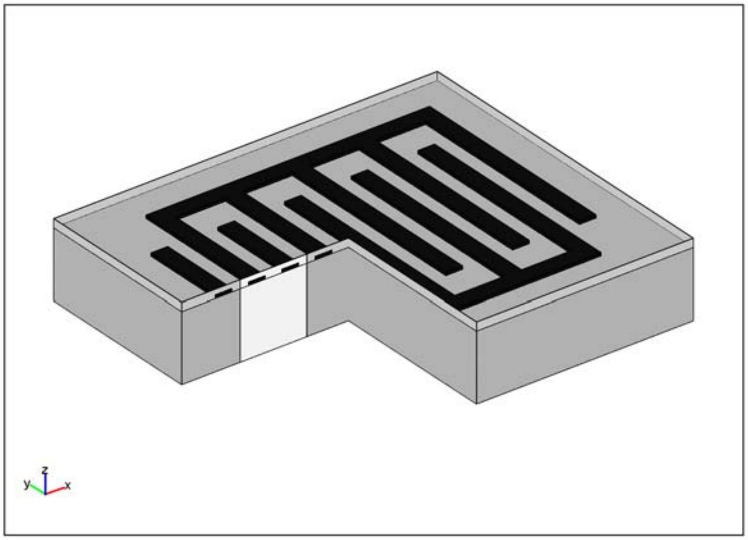


Figure 3-56: Conceptual view of the SAW gas sensor, showing the IDT electrodes (in black), the thin PIB film (light gray), and the $LiNbO_3$ substrate (dark gray). For the sake of clarity, the dimensions are not to scale and the IDT has fewer electrodes than in common devices. A slice of the geometry is removed to reveal the modeled unit cell (in white).

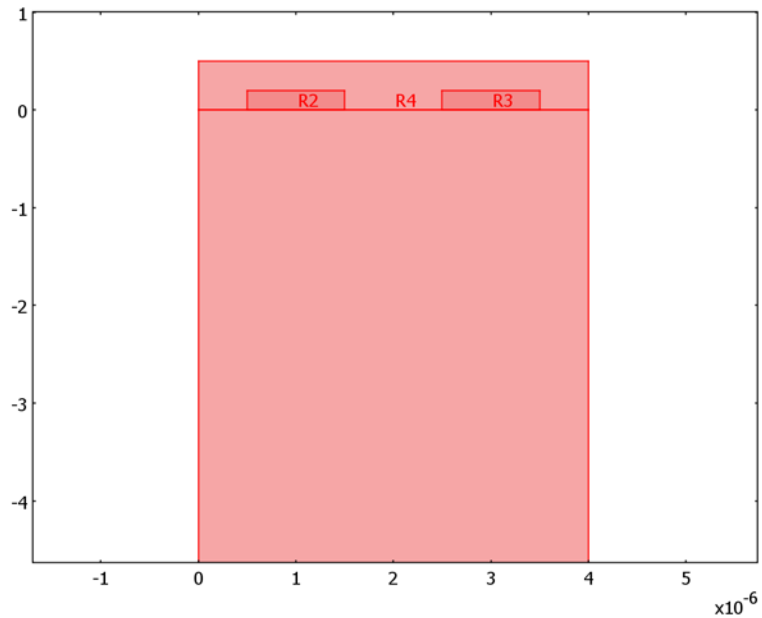


Figure 3-57: The modeled geometry of the model. A 500 nm PIB film covers two 1 µm-wide electrodes on top of the $LiNbO_3$ substrate. The substrate subdomain continues below the lower frame of the picture and has a total height of 22 µm. In the first version of the model, the substrate is the only active subdomain.

You set up the model in the Piezo Plane Strain application mode, which requires the out-of-plane strain component to be zero. This should be a valid assumption, considering that the SAW is generated in the model plane and that the sensor is thick in the out-of-plane direction.

The first version of the model deals only with free SAW propagation in the $LiNbO_3$ substrate, without any applied electric field. In order to find the velocity of the wave, we use periodic boundary conditions to dictate that the voltage and the displacements be the same along both vertical boundaries of the geometry. This implies that the wavelength will be an integer fraction of the width of the geometry. The lowest SAW eigenmode has its wavelength equal to the width of the geometry, 4 µm. The eigenfrequency of this mode multiplied by 4 µm hence gives the velocity of the wave.

In a second version of the model, the aluminum IDT electrodes and the PIB film are added. This causes the lowest SAW mode to split up in two eigensolutions, the lowest one representing a series resonance, where propagating waves interfere constructively and the other one a parallel ("anti-") resonance, where they interfere destructively. These two frequencies constitute the edges of the stopband, within which no waves can propagate through the IDT.

The adsorption of DCM gas is represented as a slight increase of the density of the PIB film. In the third and final version of the model, 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 is then calculated as

$$\rho_{\mathrm{DCM,PIB}} = KMc\,,$$

where $K = 10^{1.4821}$ (Ref. 1) is the air/PIB partition coefficient for DCM, M is its molar mass, and

$$c = 100 \cdot 10^{-6} \cdot p/(RT)$$

is its concentration in air.

The substrate used in the simulation is YZ-cut $LiNbO_3$ with properties cited in Ref. 2. The density of the PIB film is from Ref. 1. The Poisson's ratio is taken to be 0.48, which corresponds to a rather rubbery material. The Young's modulus is set to 10 GPa. Even at the comparatively high frequencies considered in this model, this is likely an overestimation. However, a much lower value would result in a multitude of eigenmodes located inside the film. While those may be important to consider in designing a SAW sensor, the focus in this model is on the SAW modes. Also, any effects of the DCM adsorption on other material properties than the density are neglected.

Results

Figure 3-58 shows the SAW as it propagates along the surface of the piezoelectric substrate. The frequency corresponding to a 4 μm wavelength computes to 870 MHz, giving a phase velocity of 3479 m/s.

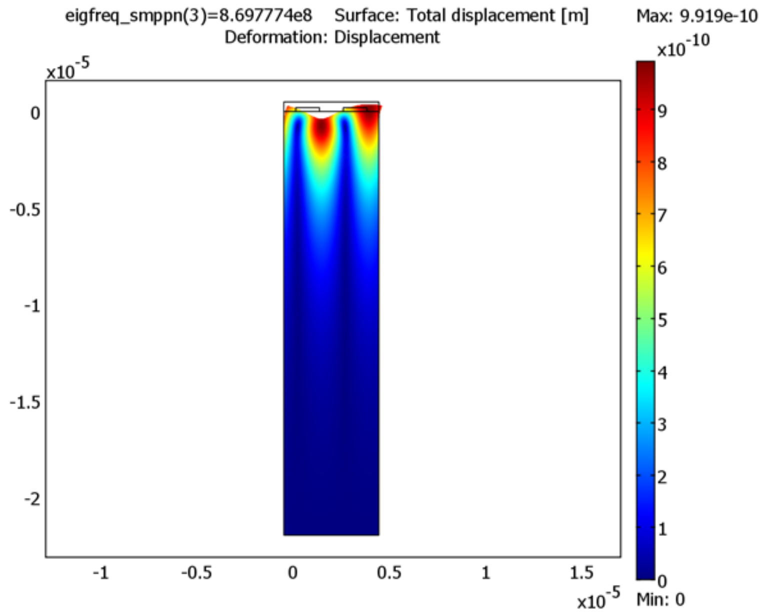


Figure 3-58: Deformed shape plot of a freely propagating SAW in the substrate. The color scale shows the magnitude of the displacements.

In the full model with the periodic IDT and the thin film included, the resonance and anti-resonance frequencies evaluate to 841 MHz and 850 MHz, respectively. Figure 3-59 and Figure 3-60 show the electric potential distribution characteristics for these solutions.

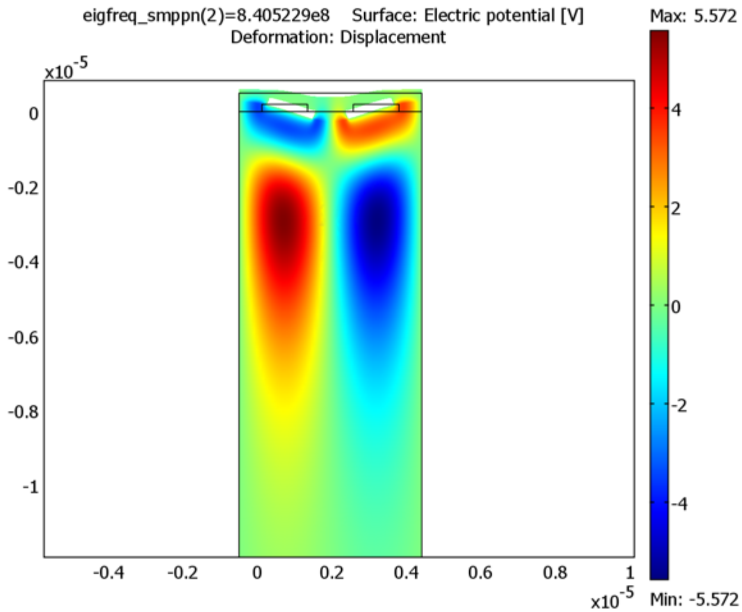


Figure 3-59: Electric potential distribution and deformations at resonance, 841 MHz. The potential is symmetric with respect to the center of each electrode.

Exposing the sensor to a 100 ppm concentration of DCM in air leads to a resonance frequency shift of 227 Hz downwards. This is computed by evaluating the resonance frequency before and after increasing 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.

In a real setup, the drift is often measured by mixing the signal from a sensor exposed to a gas with a reference signal from one protected from the gas. The beat frequency then gives the shift.

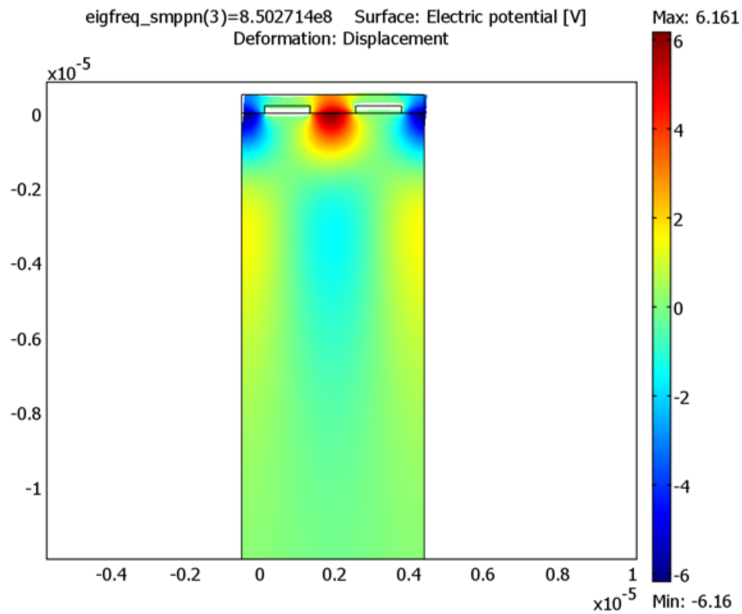


Figure 3-60: Electric potential distribution and deformations at antiresonance, 851 MHz. The potential is antisymmetric with respect to the center of the electrodes.

References

1. K. Ho and others, "Development of a Surface Acoustic Wave Sensor for In-Situ Monitoring of Volatile Organic Compounds", *Sensors* vol. 3, pp. 236–247, 2003.

2. Ahmadi and others, "Characterization of multi- and single-layer structure SAW sensor", *Sensors 2004*, *Proceedings of IEEE*, vol. 3, pp. 1129–1132, 2004.

Model Library path: `Acoustics_Module/Industrial_Models/SAW_gas_sensor`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator** and click the **New** tab.

2 From the **Space dimension** list, select **2D**.

3 In the list of application modes, select **Acoustics Module>Piezo Plane Strain>Eigenfrequency analysis**.

4 Click **OK**.

GEOMETRY MODELING

1 Create the following rectangles by repeatedly using **Draw>Specify Objects>Rectangle**. Click **OK** after specifying the data for each rectangle.

WIDTH	HEIGHT	BASES: CORNER X	BASE: CORNER Y
4	22	0	-22
1	0.2	0.5	0
1	0.2	2.5	0
4	0.5	0	0

2 Select all objects and choose **Draw>Modify>Scale**. In the dialog box that appears, enter `1e-6` for both scale factors; then click **OK**.

3 Click the **Zoom Extents** button on the Main toolbar to zoom in on the now micron-sized geometry.

OPTIONS AND SETTINGS

1 Choose **Options>Constants**.

2 Define the following constant names, expressions, and (optionally) descriptions:

NAME	EXPRESSION	DESCRIPTION
p	101.325[kPa]	Air pressure
T	25[degC]	Air temperature
R	8.3145[Pa*m^3/(K*mol)]	Gas constant
c_DCM_air	100e-6*p/(R*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

3 Click **OK**.

PHYSICS SETTINGS

In the first version of the model, you compute the velocity for SAW propagation in a homogenous, electrically insulated $LiNbO_3$ substrate. The supplied material data are with reference to the *xy*-plane.

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.

2 Select Subdomains 2–4, then clear the **Active in this domain** check box.

3 Select Subdomain 1, then select **xy plane** from the **Material orientation** list.

4 Click the **Edit** button associated with $\mathbf{c_E}$. Enter the following values in the **Elasticity matrix** dialog box; when finished, click **OK**.

$$\begin{bmatrix} 2.424e11 & 0.752e11 & 0.752e11 & 0 & 0 & 0 \\ & 2.03e11 & 0.573e11 & 0 & 0.085e11 & 0 \\ & & 2.03e11 & 0 & -0.085e11 & 0 \\ & & & 0.752e11 & 0 & 0.085e11 \\ & & & & 0.595e11 & 0 \\ & & & & & 0.595e11 \end{bmatrix}$$

5 Click the **Edit** button associated with **e**. Enter the following values in the **Coupling matrix** dialog box; when finished, click **OK**.

$$\begin{bmatrix} 1.33 & 0.23 & 0.23 & 0 & 0 & 0 \\ 0 & 0 & 0 & -2.5 & 0 & 3.7 \\ 0 & -2.5 & 2.5 & 0 & 3.7 & 0 \end{bmatrix}$$

6 Click the **Edit** button associated with $\varepsilon_{\mathbf{rS}}$. Enter the following values in the **Relative permittivity** dialog box; when finished, click **OK**.

$$\begin{bmatrix} 28.7 & 0 & 0 \\ & 85.2 & 0 \\ & & 85.2 \end{bmatrix}$$

7 in the **Density** edit field, type `4647`.

8 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 From the **Physics** menu, choose **Boundary Settings**.

2 Select Boundary 2, then set the **Constraint condition** to **Fixed**.

3 Select all exterior boundaries (Boundaries 1, 2, 4, 7, 10, 12, 15, and 16).

4 On the **Electric BC** page, set the **Boundary condition** to **Zero charge/Symmetry**.

5 Click **OK**.

6 Choose **Physics>Periodic Conditions>Periodic Boundary Conditions**.

7 On the **Source** page, select Boundary 1. On the first row in the **Expression** column, type `u`, then press Enter; the software automatically adds the corresponding **Constraint name** `pconstr1` to the table.

8 On the **Destination** page, check Boundary 16 and type `u` in the **Expression** edit field.

9 On the **Source Vertices** page, Ctrl-click to select Vertices 1 and 2, then click the **>>** button.

10 On the **Destination Vertices** page, Ctrl-click to select Vertices 12 and 13, then click the **>>** button.

11 Repeat Steps 7 through 10 to define the expressions `v` and `V` in an analogous fashion, starting by entering them in the **Expression** edit field on the **Source** page, on the 2nd and 3rd row, respectively.

12 When done, click **OK** to close the **Periodic Boundary Conditions** dialog box.

MESH GENERATION

1 Choose **Mesh>Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list, choose **Extremely fine**.

3 On the **Subdomain** page, select all subdomains and set the **Method** to **Quad**.

4 On the **Boundary** page, select the upper boundaries of the substrate (Boundaries 4, 7, 10, 12, and 15) and set the **Maximum element size** to `0.05e-6`.

5 Click **Remesh**, then click **OK**. When done, a zoom-in on the upper part of the geometry should look like Figure 3-61.

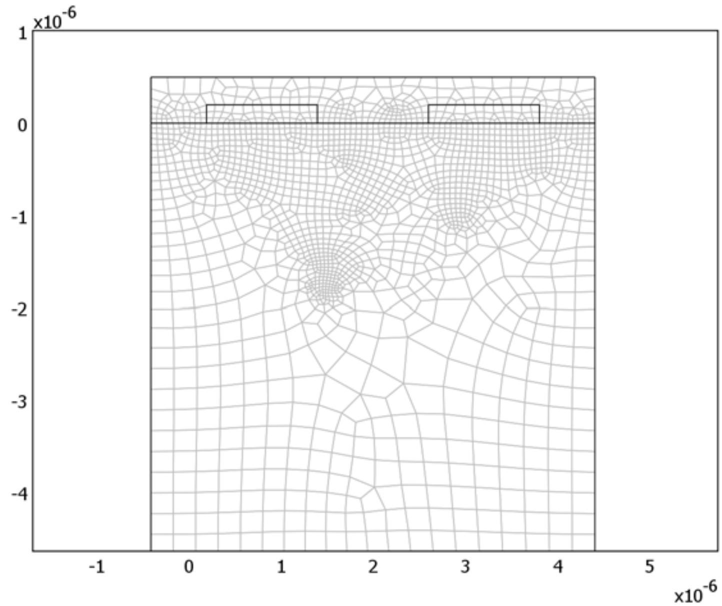


Figure 3-61: The meshed geometry.

COMPUTING THE SOLUTION

1 From the **Solve** menu, open the **Solver Parameters** dialog box.

2 Enter `850e6` in the **Search for eigenfrequencies around** edit field, then click **OK**.

3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The solver returns 6 eigensolutions with eigenfrequencies in the vicinity of 850 MHz. At 869.8 MHz, two of them are—within the numerical accuracy—the same. These show the shape and the frequency for a SAW with wavelength equal to the width of the geometry.

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.

2 On the **General** page, select one of the eigenfrequencies equal to 869.8 MHz.

3 On the **Deform** page, select the **Deformed shape plot** check box. Clear the **Auto** check box and type `400` in the **Scale factor** edit field.

4 Click **OK** to close the dialog box and see a plot of the total displacement. If you want to, you can repeat the procedure with the other solution to verify that they are the same, only shifted by 90 degrees. One of the solutions look like that in Figure 3-58.
5 To evaluate the velocity, choose **Postprocessing>Data Display>Global**.
6 In the **Expression** edit field, type `eigfreq_smppn*4[um]`.
7 In the **Eigenfrequency** list, select one of the 869.8 MHz entries.
8 Click **OK** to see the value of the velocity in the message log. It evaluates to approximately 3479 m/s.

This concludes the first part of the model. Proceed to find out how the electrodes and the PIB film affect the behavior of the SAW.

Sensor without Gas Exposure

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.
2 Select Subdomains 2–4, then select the **Active in this domain** check box.
3 From the **Material model** list, select **Decoupled, isotropic**.
4 Select only Subdomain 2.
5 On the **Structural** page, enter `E_PIB` for the **Young's modulus**, `nu_PIB` for the **Poisson's ratio**, and `rho_PIB` for the **Density**.
6 On the **Electrical** page, select the **Enable electrical equation** check box and enter `eps_PIB` in the **Relative permittivity** edit field.
7 Select Subdomains 3 and 4, then click the **Load** button.
8 In the **Materials/Coefficients Library** dialog box, select **Basic Material Properties>Aluminum** and click **OK**.
9 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 From the **Physics** menu, choose **Boundary Settings**.
2 Select the **Interior boundaries** check box.
3 Select Boundaries 6–9 and 11–14. On the **Electric BC** page, set the condition to **Electric potential**. Keep the default zero potential.
4 Select Boundaries 3, 5, and 17, and set the condition to **Zero charge/Symmetry**.
5 Click **OK** to close the dialog box.

Note: The eigenfrequencies and hence the stopband do not depend on the values of the potentials. In fact, for linear eigenfrequency problems, they are automatically set to zero at the electrodes, regardless of the applied value. You can solve the corresponding driven problem by switching to a frequency response analysis and applying different potentials to the electrodes.

---

6 Choose **Physics>Periodic Conditions>Periodic Boundary Conditions**.
7 On the **Source** page, select Boundary 3. Enter u in the first **Expression** edit field and v in the row below, and V in the third row.
8 Click the **Destination** tab.
9 From the **Constraint name** list, select **pconstr1**; check Boundary 17 (in addition to the already selected Boundary 16); and type u in the **Expression** edit field.
10 From the **Constraint name** list, select **pconstr2**; check Boundary 17; and type v in the **Expression** edit field.
11 From the **Constraint name** list, select **pconstr3**; check Boundary 17; and type V in the **Expression** edit field.
12 Click **OK** to close the dialog box.

You have now established the periodicity in the PIB film.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

If you are still using the manual scaling of the deformations from the previous exercise, the plot that appears after solving will look rather distorted. Proceed as follows to find the SAW modes and use more suitable plot parameters:

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.
2 On the **General** page, select the 850 MHz eigenfrequency.
3 On the **Deform** page, type 40 in the **Scale factor** edit field.
4 Click **Apply** to view a plot of the total displacement at anti-resonance.
5 On the **General** page, select the 841 MHz eigenfrequency, then click **Apply** to see the deformations at resonance. This plot should look like Figure 3-62.

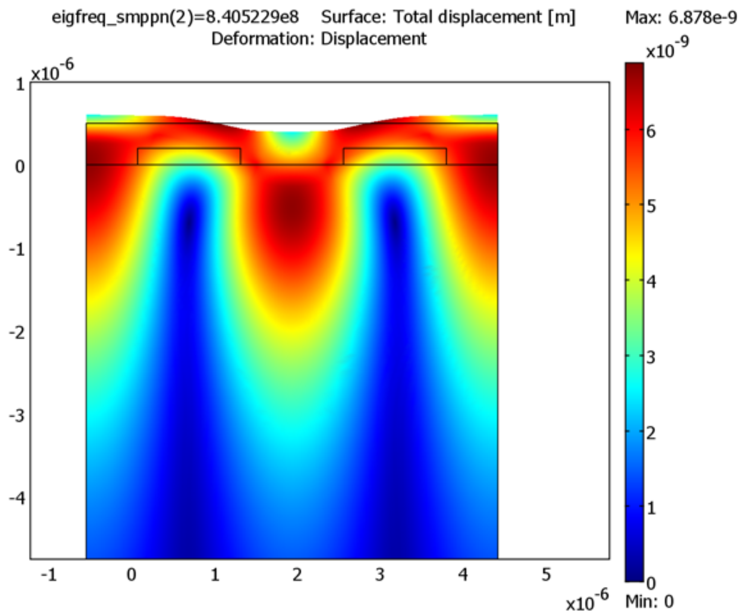


Figure 3-62: Deformations at resonance.

A plot of the electric potential shows a qualitative difference between the two solutions.

6 On the **Surface** page, select **Piezo Plane Strain (smppn)>Electric potential** from the **Predefined quantities** list.

7 Click **Apply** to see the potential distribution at resonance, as shown in Figure 3-59 on page 215. Notice that it is symmetric with respect to each individual electrode.

8 On the **General** page, select the 850 MHz eigenfrequency and click **OK** to see the potential distribution at anti-resonance, as in Figure 3-60 on page 216. This time, it is antisymmetric.

Sensor with Gas Exposure

In the final version of this model, you expose the sensor to DCM gas. The eigenfrequencies then shift by a very small amount. To see the shift, you need to include more digits in the output.

1 Choose **Postprocessing>Data Display>Global**.

2 Enter the expression `eigfreq_smppn` and select the 841 MHz eigenfrequency.

3 Select the **Display result in full precision** check box, then click **OK**.

The message log now shows all computed digits of the eigenfrequency.

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 Select Subdomain 2. On the **Structural** page, change the **Density** so that it reads `rho_PIB+rho_DCM_PIB`.

3 Click **OK** to close the **Subdomain Settings** dialog box.

COMPUTING THE SOLUTION

Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

1 Choose **Postprocessing>Data Display>Global**.

2 Make sure that the expression still says `eigfreq_smppn` and select the 841 MHz eigenfrequency.

3 Click **OK**.

The first 6 digits of the eigenfrequency are the same as before. Subtracting the new value from the previous value (which is most easily done by copying and pasting the results from the message log) shows that the eigenfrequency with gas exposure is lower by 227 Hz.

4

Benchmark Models

This chapter covers benchmarks models solving acoustics problems with an established or analytic solution. The benchmark data provide the possibility to validate results obtained with the Acoustics Module.

Vibrations of a Disk Backed by an Air-Filled Cylinder

Introduction

The vibration modes of a thin or thick circular disk 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 disk? This is the question you address in this model.

Note: This model requires the Acoustics Module and the Structural Mechanics Module.

Model Definition

In COMSOL Multiphysics you can model an air-filled cylinder sealed by a thin disk in one end using at least two different approaches. You describe the pressure in the cavity with a Pressure Acoustics application mode, while the model of the disk can be either a thin shell in 3D, using shell elements, or a 2D plate. The latter approach to modeling this acoustic-structure interaction is possible thanks to nonlocal couplings and COMSOL Multiphysics' ability to model in different numbers of space dimensions at the same time—*extended multiphysics*.

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 and simulations. 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 disk only 0.38 mm thick. Some of the theoretical eigenfrequencies of a thin disk in vacuum and of a rigidly sealed chamber are given in the following table (according to Ref. 1).

TABLE 4-1: BENCHMARK VALUES FOR EIGENFREQUENCIES OF THE DISK AND THE CYLINDER

NUMBER	CLAMPED DISK IN VACUUM (HZ)	RIGIDLY SEALED CYLINDER (HZ)
1	671.8	672.5
2	1398	1345
3	2293	2018
4	2615	2645
5	3356	2690
6	4000	4387

Here you model the coupled system using the extended multiphysics approach. This means that you draw the disk in a 2D geometry and model it with Mindlin-theory DRM-plate elements, while you draw the cylinder in a separate 3D geometry. 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 disk and the cylinder separately and compare them with the theoretical values in Table 4-1. This way you can verify the basic components of the model and assess the accuracy of the FEM solution before modeling the coupled system.

Results and Discussion

Most of the modes show rather weak coupling between the structural bending of the disk and the pressure field in the cylinder. It is, however, interesting to note that some of the uncoupled modes have been split into one co-vibrating and one contra-vibrating mode with distinct eigenfrequencies. This is the case for modes 1 and 2 and for modes 9 and 12 in the FEM solution. The table below shows a comparison of the eigenfrequencies from the COMSOL Multiphysics analysis with the semi-analytical and experimental frequencies reported by D. G. Gorman and others in Ref. 1. The table also states whether the modes are structurally dominated (str), acoustically dominated (ac), or tightly coupled (str/ac).

TABLE 4-2: RESULTS FROM SEMI-ANALYTICAL AND COMSOL MULTIPHYSICS ANALYSES AND EXPERIMENTAL DATA

TYPE	SEMI-ANALYTICAL (HZ)	COMSOL MULTIPHYSICS (HZ)	EXPERIMENTAL (HZ)
str/ac	636.9	637.2	630
str/ac	707.7	707.7	685
ac	1347	1347.4	1348
str	1394	1395.3	1376
ac	2018	2018.6	2040
str	2289	2293.1	2170
str/ac	2607	2612.1	2596
ac	2645	2646.3	–
str/ac	2697	2697.1	2689
ac	2730	2730.9	2756
ac	2968	2969.3	2971

As the table shows, the FEM solution is in good agreement with both the theoretical predictions and the experimentally measured values for the eigenfrequencies. As you might expect from the evaluation of the accuracies for the uncoupled problems, the precision is generally better for the acoustics-dominated modes.

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.*, vol. 215, Part C, 2001.

Model Library path: `Acoustics_Module/Benchmark_Models/coupled_vibration`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** from the **Space dimension** list.

2 In the list of application modes, select **Structural Mechanics Module>Mindlin Plate>Eigenfrequency analysis**.

3 Click **OK**.

GEOMETRY MODELING

The geometry of the disk is a solid circle. Its location does not really matter because you embed it in 3D using coupling variables, but the transformations are trivial if you center the disk at the origin:

1 Press the Shift key and click the **Ellipse/Circle (Centered)** button.

2 In the **Circle** dialog box, type `0.038` in the **Radius** edit field and click **OK** to create a circle of radius 0.038 m, centered at the origin.

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

PHYSICS SETTINGS

Boundary Conditions

The edges of the disk are welded to the cylinder and can therefore be described as rigidly *clamped* or fixed.

1 From the **Physics** menu, choose **Boundary Settings**.

2 Select one of the boundaries and then press Ctrl+A to select all boundaries.

3 Make sure that you have selected **Tangent and normal coord. sys. (t,n)** in the **Coordinate system** list.

4 Select **Fixed** in the **Condition** list.

5 Click **OK**.

Subdomain Settings—Material Properties

The steel disk has the following material properties (in the default SI units):

- Young's modulus, $E = 2.1 \cdot 10^{11}$
- Poisson's ratio, $\nu = 0.3$
- Density, $\rho = 7800$

1 From the **Physics** menu, choose **Subdomain Settings**.

2 Select Subdomain 1.

3 Enter material data according to the following table:

MATERIAL PROPERTY	VALUE
E	2.1e11
ν	0.3
ρ	7800
thickness	0.00038

4 Click **OK**.

MESH GENERATION

To obtain accurate values of the eigenfrequencies of the disk, you need a mesh that is finer than the one produced with the default settings.

1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.

2 Click the **Custom mesh size** option button and type `0.002` in the **Maximum element size** edit field.

3 Click the **Remesh** button and then click **OK**.

COMPUTING THE SOLUTION

When solving for the eigenfrequencies of the disk in vacuum, only the frequency interval between 500 Hz and 3250 Hz is of interest. Start by searching for the 20 first eigenfrequencies (some of these are almost identical and come from double eigenvalues) and make the solver start its search around 500 Hz:

1 From the **Solve** menu, choose **Solver Parameters**.

2 Type 20 in the **Desired number of eigenfrequencies** edit field.

3 Type 500 in the **Search for eigenfrequencies around** edit field.

4 Click **OK**.

5 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

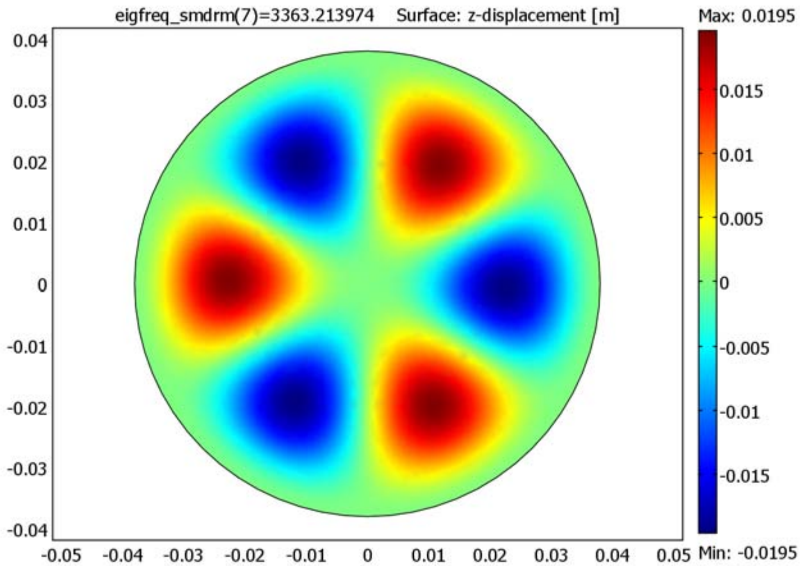
1 Click the **3D Surface Plot** button to see the deflection of the disk.

Next, try looking at some of the eigenmodes.

2 From the **Postprocessing** menu, choose **Plot Parameters**.

3 On the **General** page, choose an eigenfrequency from the **Eigenfrequency** list in the **Solution to use** area; click **Apply** to generate the corresponding plot.

4 When done, click **Cancel** or **OK** to close the **Plot Parameters** dialog box.



The eigenmode associated with the eigenfrequency around 3360 Hz.

You can now compare the eigenfrequencies with the theoretical values for a thin disk. The discrepancy is below 2% for all eigenmodes in the interval, so the conclusion is that the mesh resolution is sufficient.

Adding the 3D Pressure Acoustics Application Mode

Now add a second geometry that will contain the cylinder and the acoustic pressure variable using a 3D Pressure Acoustics application mode.

1 From the **Multiphysics** menu, choose **Model Navigator**.

2 Click the **Add Geometry** button to add a second geometry.

3 In the **Add Geometry** dialog box, select **3D** from the **Space dimension** list.

4 Click **OK**.

5 In the list of application modes, select **Acoustics Module>Pressure Acoustics>Eigenfrequency analysis**.

6 Click **Add**.

7 Click **OK**.

GEOMETRY MODELING

1 Click the **Cylinder** toolbar button.

2 Type `0.038` in the **Radius** edit field and `0.255` in the **Height** edit field.

3 Click **OK**.

4 Click the **Zoom Extents** button on the Main toolbar.

PHYSICS SETTINGS

Boundary Conditions

For the moment, assume that all boundaries are perfect hard walls, which is the default boundary condition.

Subdomain Settings

1 From the **Physics** menu, choose **Subdomain Settings**.

2 Select Subdomain 1.

3 Type `1.2` in the **Fluid density** (ρ_0) edit field. Leave the other properties at their default value.

4 Click **OK**.

MESH GENERATION

Click the **Initialize Mesh** button to create a mesh using the default parameters.

COMPUTING THE SOLUTION

To solve for the acoustic modes only, you must deactivate the Mindlin Plate application mode during the solution. If the plate is not deactivated, COMSOL Multiphysics solves the two eigenvalue problems simultaneously but independently of one another.

1 From the **Solve** menu, choose **Solver Manager**.

2 Click the **Solve For** tab.

3 Ctrl-click on the **Mindlin Plate (smdrm)** folder to clear that application mode's variables from the list of variables to solve for, then click **OK**.

4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 Click the **Plot Parameters** button on the Main toolbar.

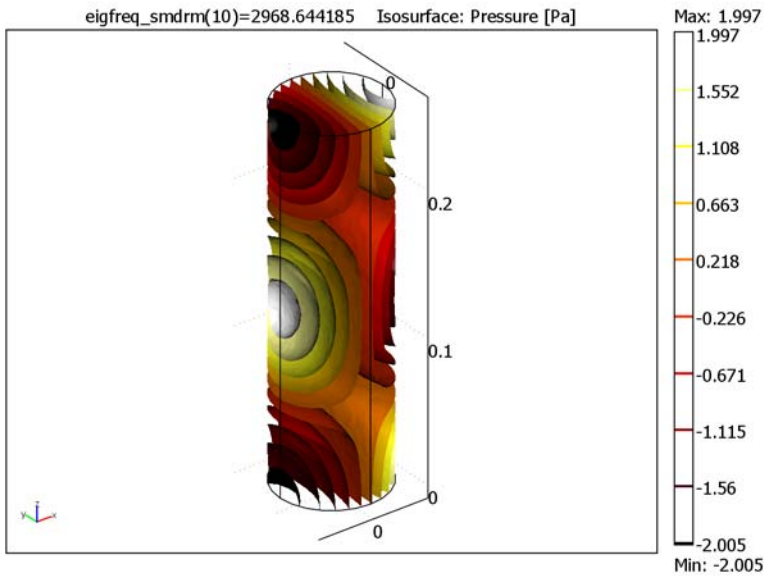
2 On the **General** page, clear the **Slice** check box and select the **Isosurface** check box.

3 In the **Solution to use** area, select one of the solutions near 2730 Hz from the **Eigenfrequency** list.

4 Click the **Isosurface** tab. In the edit field for isosurface levels under **Number of levels**, type 10. From the **Colormap** list select **hot**, then click **Apply**.

5 Click the **Headlight** button on the Camera toolbar.

6 Return to the **General** page and try a few different eigenmodes by selecting the corresponding eigenfrequencies, clicking **Apply** to generate each plot.



Pressure isosurfaces for one of two eigenmodes associated with an eigenfrequency near 2969 Hz.

You can also compare these eigenfrequencies with the theoretical values in Table 4-2 on page 228. This time, the relative error seems to be much smaller than for the disk, which means that any additional mesh refinement should be done on the plate part.

Coupling the Equations

The first step in the process of coupling the Mindlin plate elements to the acoustic equation is to create the nonlocal couplings; use coupling variables to make the pressure available as a load on the plate and the out-of-plane displacement of the plate a valid parameter in the coefficients for the acoustic equation.

OPTIONS AND SETTINGS—COUPLING VARIABLES

First define a coupling variable for the acoustic pressure from the top face of the cylinder to the disk (Mindlin plate):

1. On the **Options** menu, point to **Extrusion Coupling Variables** and then click **Boundary Variables**.
2. In the **Boundary Extrusion Variables** dialog box, select Boundary 4 (the top face) of the cylinder.
3. Type `p` in the top row under both **Name** and **Expression**.
4. Click the **Destination** tab.
5. Select **Geom1** in the **Geometry** list, then select Subdomain 1 (the disk) in the 2D geometry.
6. Click the **Source Vertices** tab.
7. In the **Vertex selection** list, select Vertices 2, 4, and 8. Click the **>>** button.
8. Click the **Destination Vertices** tab.
9. In the **Vertex selection** list, select Vertices 1, 2, and 4. Click the **>>** button.
10. Click **OK**.

Now define a coupling variable for the out-of-plane displacement, w, from the disk to the top face of the cylinder. Also the acceleration, w_{tt}, is needed.

1. If you are in the Pressure Acoustics application mode, switch to the Mindlin Plate application mode by choosing **Geom1: Mindlin Plate (smdrm)** from the **Multiphysics** menu.
2. On the **Options** menu, point to **Extrusion Coupling Variables** and then click **Subdomain Variables**.
3. In the **Subdomain Extrusion Variables** dialog box, select Subdomain 1.
4. Type `w` in the top row under both **Name** and **Expression**. Type `wtt` in the second row under both **Name** and **Expression**.
5. Click the **Destination** tab.
6. Select `w` from the **Variable** list.
7. Select **Geom2** in the **Geometry** list, then select the check box next to Boundary 4.
8. Click the **Source Vertices** tab.
9. In the **Vertex selection** list, select Vertices 1, 2, and 4. Click the **>>** button.
10. Click the **Destination Vertices** tab.
11. In the **Vertex selection** list, select Vertices 2, 4, and 8. Click the **>>** button.

12 Click the **Destination** tab.

13 Select `wtt` from the **Variable** list.

14 Select **Geom2** in the **Geometry** list, then select the check box next to Boundary 4.

15 Click the **Source Vertices** tab.

16 In the **Vertex selection** list, select Vertices 1, 2, and 4. Click the **>>** button.

17 Click the **Destination Vertices** tab.

18 In the **Vertex selection** list, select Vertices 2, 4, and 8. Click the **>>** button.

19 Click **OK**.

PHYSICS SETTINGS

Boundary Conditions

The sound-hard boundary condition for a rigid wall is that the normal acceleration vanishes. For a moving wall, such as the thin disk that now seals the cylinder, the appropriate condition is instead

$$\frac{\mathbf{n} \cdot \nabla p}{\rho_a} = -a$$

where a is the normal acceleration of the wall.

1 If the Mindlin Plate application mode is selected, switch to the Pressure Acoustics application mode by choosing **2 Geom2: Pressure Acoustics (acpr)** from the **Multiphysics** menu.

2 From the **Physics** menu, choose **Boundary Settings**.

3 Select the top of the cylinder where the plate is located, that is, Boundary 4.

4 From the **Boundary condition** list, select **Normal acceleration**.

5 In the $\mathbf{a_n}$ edit field, type `-wtt` (the structural acceleration in the negative z direction).

6 Click **OK**.

Subdomain Settings

The acoustic pressure acting as a normal load constitutes the coupling in the opposite direction.

1 From the **Multiphysics** menu, choose **1 Geom1: Mindlin Plate (smdrm)**.

2 Open the **Subdomain Settings** dialog box.

3 Click the **Load** tab.

4 Select Subdomain 1.

5 In the **F_z** edit field, type `p` to specify the pressure as a surface load on the disk.

6 Click **OK**.

7 Switch back to the 3D geometry by choosing **2 Geom2: Pressure Acoustics (acpr)** from the **Multiphysics** menu.

COMPUTING THE SOLUTION

1 From the **Solve** menu, choose **Solver Manager**.

2 In the **Solver Manager** dialog box, click the **Solve For** tab.

3 Reactivate the Mindlin Plate application mode by selecting both **Geom1 (2D)** and **Geom2 (3D)** and all corresponding variables in the **Solve for** list.

4 Click the **Solve** button to compute the solution. When done, click **OK**.

POSTPROCESSING AND VISUALIZATION

1 Open the **Plot Parameters** dialog box.

2 On the **General** page, add boundary and deformed shape plots to the isosurface plot by selecting the **Boundary** and **Deformed shape** check boxes in the **Plot type** area.

3 Click the **Boundary** tab.

4 In the **Boundary data** area, enter the **Expression** `lambda^2*w`, that is, the normal acceleration of the disk. On the other boundaries, w is not defined so those boundaries are invisible.

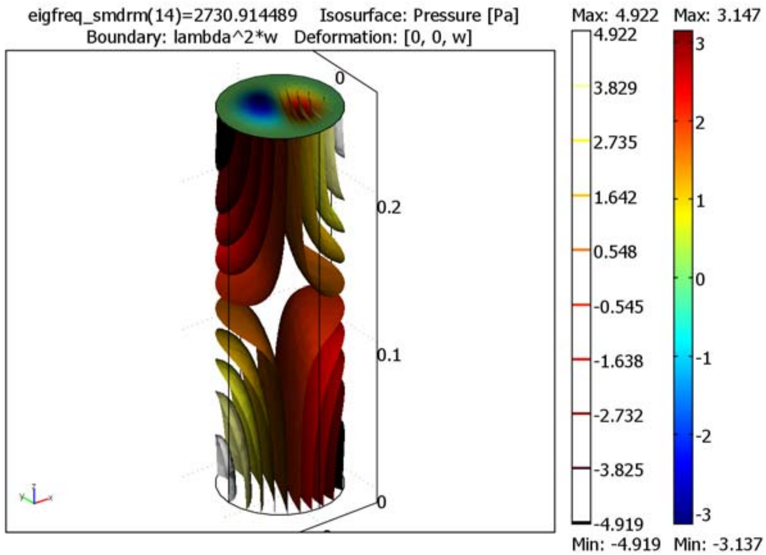
5 Click the **Deform** tab and select the **Boundary** check box only in the **Domain types to deform** area.

6 In the **Deformation data** area, click the **Boundary Data** tab and type `0`, `0`, and `w` in the **x component**, **y component**, and **z component** edit field, respectively.

7 Click the **General** tab.

8 Select an entry from the **Eigenfrequency** list, then click **Apply** to examine the corresponding eigenmode.

9 When finished, click **Cancel** or **OK** to close the **Plot Parameters** dialog box.



One of the two eigenmodes with an eigenfrequency near 2730 Hz.

Open Pipe

Introduction

If you plan on setting up a large, complicated acoustics model, it can be useful to break it up in smaller, simpler problems. In this example, a vibrating piston is mounted inside one end of a cylindrical pipe. The other end is open and set in a plane baffle. A first version of the model studies the air outside the baffle as a PML (perfectly matched layer) region. A second version measures and then uses the impedance load on the piston, where it replaces the air region with an impedance boundary condition applied to the tube's open end. The impedance boundary condition uses a radiation impedance given as a function of the measured impedance at the piston. You can employ the technique of measuring impedances and reusing them in impedance boundary conditions to handle arbitrary kinds of pipe openings.

Model Definition

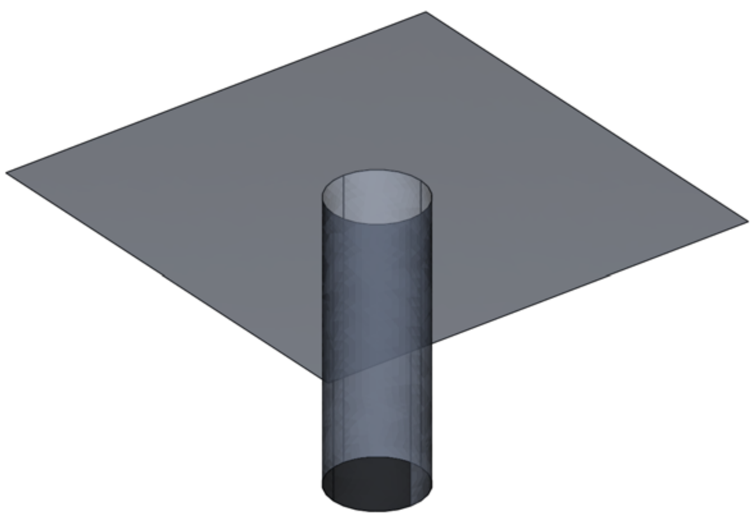


Figure 4-1: Pipe geometry. The flange is cut off in the illustration but is assumed to extend to infinity. The piston makes up the bottom of the pipe.

Figure 4-1 shows the geometry simulated in this model. A pipe of length $L = 1.5$ m and radius $a = 0.25$ m has a driving piston at one end. The other end is open and flush with the infinite hard wall in which it is set. The piston vibrates harmonically with a velocity $v = v_0 e^{i\omega t}$ where $v_0 = 1$ m/s, and $\omega = 2\pi f$ is the angular frequency (rad/s). The model sweeps the frequency, f, through a range of values between 10 Hz and 700 Hz. The acoustic medium is air with a density of 1.25 kg/m^3 and a speed of sound of 343 m/s.

The axial symmetry of the geometry and the physics makes it natural to set the model up in a 2D axisymmetric application mode. Two cylindrical PMLs represent the air outside the pipe as shown in Figure 4-2.

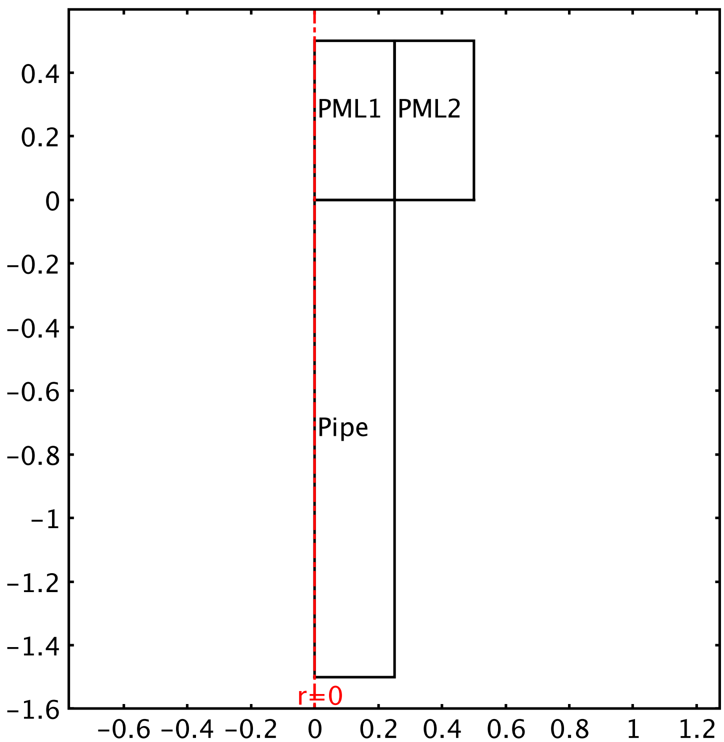


Figure 4-2: Model geometry. PML 1 is damping only in the z direction, while PML 2 is damping in both the r direction and the z direction.

The PMLs serve to absorb the outgoing waves so that the nonphysical reflections at their exterior boundaries have a minimal influence on the pressure field inside the pipe. PMLs require the tangential component of the damping to be continuous across a boundary. Hence the PML just above the pipe is damping only in the z direction, while the one outside the pipe is damping both in the r direction and the z direction. For further details about the PML implementation in the Acoustics Module, please refer to "Perfectly Matched Layers (PMLs)" on page 37 in the *Acoustics Module User's Guide*.

DOMAIN EQUATIONS

For harmonic sound waves this model uses the frequency-domain Helmholtz equation for sound pressure:

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

Here the acoustic pressure is a harmonic quantity, $p = p_0 e^{i\omega t}$ (N/m^2), ρ_0 is the density (kg/m^3), $\mathbf{q}$ denotes an optional *dipole source* (N /m^3), and c_s is the speed of sound (m/s). The model under study includes no dipole source.

BOUNDARY CONDITIONS

The first version of the model uses two distinct boundary conditions. First, it represents the hard walls of the pipe and the flange by the equation

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

where $\mathbf{n}$ is the outward-pointing unit normal vector seen from inside the acoustics domain. The model also uses this condition at the exterior boundaries of the PMLs. Second, the piston is modeled with a normal acceleration condition:

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

where the normal acceleration, a_n, is defined as $i\omega v_0$.

The second version of the model replaces the PML domains with an impedance boundary condition at the opening of the pipe:

$$-\mathbf{n} \cdot \left(\frac{1}{\rho_0}(\nabla p - q_s)\right) = \frac{i\omega p}{Z}$$

The impedance Z is a function of the frequency obtained by measuring and transforming the radiation load on the piston. Given a piston impedance $Z_0 = p/v$ evaluated at the piston, introduce the quantities α_0 and β_0 such that

$$Z_0 = \rho_0 c_s \tanh\pi(\alpha_0 + i\beta_0) \tag{4-1}$$

With $\alpha_L = \alpha_0$ and $\beta_L = \beta_0 - kL/\pi$, where $k = \omega/c_s$, define

$$Z = Z_L = \rho_0 c_s \tanh\pi(\alpha_L + i\beta_L) \tag{4-2}$$

This transformation is derived in Ref. 1 as a valid approximation in the long-wavelength limit where only plane waves can propagate inside the tube. Note that this reference uses the definition $p = p_0 e^{i\omega t}$, which makes the formulas look slightly different. Because Z corresponds to the radiation impedance of a membrane set in an infinite hard wall plane, it is possible to analytically derive values for α_L and β_L; these values appear in Ref. 1.

Results and Discussion

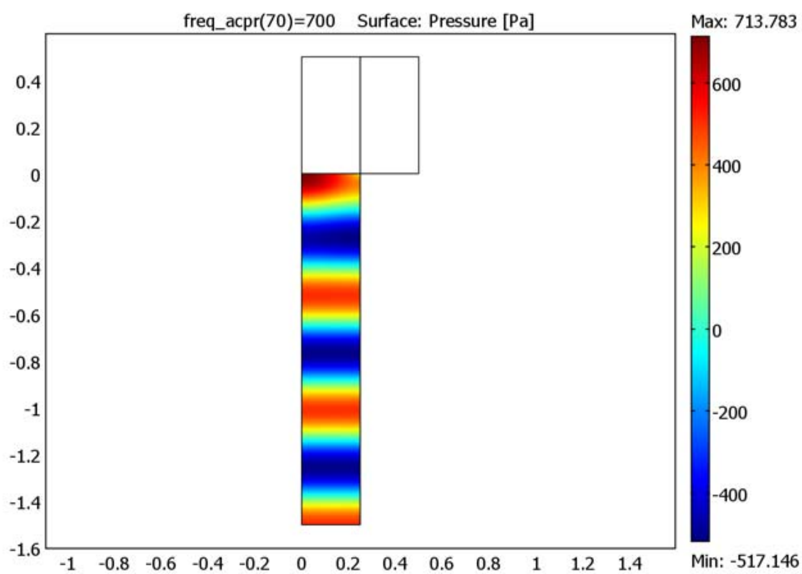


Figure 4-3: Acoustic pressure field in the pipe at 700 Hz.

The pressure field in the pipe at 700 Hz (Figure 4-3) is dominated by an outgoing and a reflected plane wave except close to the opening.

Figure 4-4 shows the reactance measured at the piston. The zero crossings of the reactance plot occur at the acoustic eigenfrequencies of the tube and the outside air. Comparing these to the simple analytic case of zero pressure at the open end, the eigenfrequencies of the open tube are consistently lower. This is to be expected because the computed eigenmodes are not restrained to the inside of the pipe but can spill out into the open.

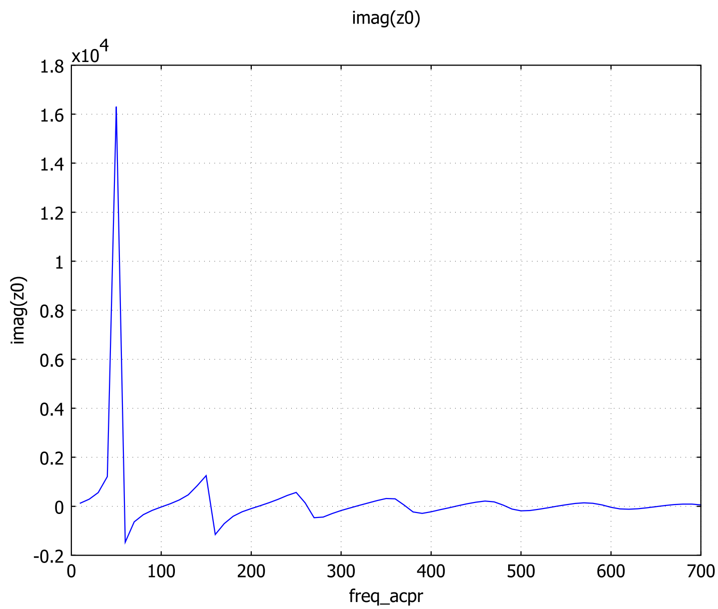


Figure 4-4: The reactance at the piston as a function of frequency.

The analytical eigenfrequencies are given by the expression

$$f_n = \frac{nc_s}{4L}$$

The accuracy of the computed eigenfrequencies is limited by the pitch of the frequency sweep. The following table is based on a pitch of 1 Hz.

TABLE 4-3: COMPUTED AND SIMPLIFIED ANALYTIC FREQUENCIES IN THE OPEN PIPE

COMPUTED (HZ)	SIMPLIFIED ANALYTIC (HZ)
50.4	57.2
101.8	114.3
154.2	171.5

TABLE 4-3: COMPUTED AND SIMPLIFIED ANALYTIC FREQUENCIES IN THE OPEN PIPE

COMPUTED (HZ)	**SIMPLIFIED ANALYTIC (HZ)**
207.5	228.7
261.6	285.8
316.3	343.0
371.4	400.2
426.8	457.3
482.7	514.5
538.9	571.7
595.6	628.8
652.8	686.0

Using values for α_L and β_L from Ref. 1 in Equation 4-1 and Equation 4-2 lets you calculate semi-analytical values for the piston impedance, Z_0. Figure 4-5 compares this impedance with the computed impedance.

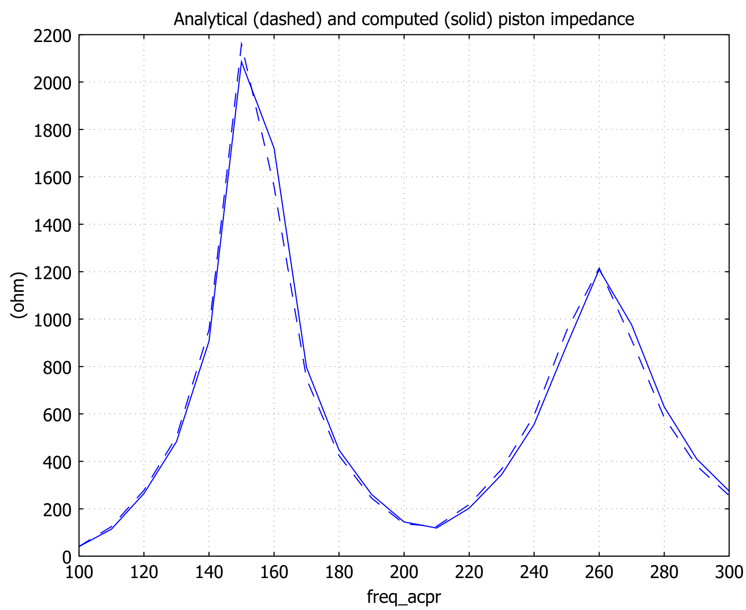


Figure 4-5: Measured (solid) and semi-analytical (dashed) impedance at the piston. Because the fit is so good that it is difficult to tell the curves apart, the figure zooms in on the frequency range from 100 to 300 Hz.

Modeling in COMSOL Multiphysics

This model is set up in 2D axisymmetry using the Pressure Acoustics application mode of the Acoustics Module. This application mode has automated support for PMLs, making it a straightforward task to set up the model. Using the Solver Manager, you store the PML solution for reuse in the impedance boundary condition of the second version of the model.

Reference

1. P.M. Morse and K.U. Ingard, *Theoretical Acoustics*, Princeton Univ. Press, 1986.

Model Library path: `Acoustics_Module/Benchmark_Models/open_pipe`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Start COMSOL Multiphysics.

2 In the **Model Navigator**, select **Axial symmetry (2D)** from the **Space dimension** list.

3 From the list of application modes select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis**.

4 Click **OK**.

OPTIONS

1 Open the **Constants** dialog box from the **Options** menu and enter the values in the following table (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
v0	1[m/s]	Maximum piston velocity
a	0.25[m]	Pipe radius
L	1.5[m]	Pipe length
rho_air	1.25[kg/m^3]	Air density
cs_air	343[m/s]	Speed of sound in air

2 Choose **Options>Expressions>Scalar Expressions** and enter the expressions in the following table. Alternatively, to avoid entering the expressions, click the

Import Variables From File button in the same dialog box. The file open_pipe_expr.txt contains all the expressions. Load this file from the COMSOL Multiphysics installation directory under /models/Acoustics_Module/Benchmark_Models/.

NAME	EXPRESSION	DESCRIPTION
k_air	omega_acpr/cs_air	Wave number
a0	i*omega_acpr*v0	Piston acceleration
z0	P/v0	Piston impedance
eta0	z0/(rho_air*cs_air)	Normalized piston impedance
alpha0	real(atanh(eta0))/pi	Hyperbolic piston impedance factor
beta0	imag(atanh(eta0))/pi	Hyperbolic piston impedance factor
alphaL	alpha0	Hyperbolic radiation impedance factor
betaL	beta0-k_air*L/pi	Hyperbolic radiation impedance factor
etaL	tanh(pi*(alphaL+i*betaL))	Normalized radiation impedance
zL	nojac(etaL)*rho_air*cs_air	Radiation impedance

The nojac operator ensures that zL does not contribute to the Jacobian matrix. In practice, this means that when you eventually use zL in a boundary condition, it depends on the previous solution and does not affect the current one.

3 Click **OK**.

GEOMETRY MODELING

1 Use the **Rectangle** dialog box (which you reach, for example, by shift-clicking the **Rectangle/Square** button on the Draw toolbar) to create two rectangles with the following properties:

WIDTH	HEIGHT	BASE CORNER R	BASE CORNER Z
0.5	0.5	0	0
0.25	2	0	-1.5

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

PHYSICS SETTINGS

Subdomain Settings

Choose **Physics>Subdomain Settings** and apply the following settings; when done, click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3
ρ_0	`rho_air`	`rho_air`	`rho_air`
c_s	`cs_air`	`cs_air`	`cs_air`
Type of PML	None	Cylindrical	Cylindrical
Absorbing in r direction	-	unchecked	`0.25`
Absorbing in z direction	-	`0.5`	`0.5`
R_0	-	-	`0.25`

Boundary Conditions

Choose **Physics>Boundary Settings** and apply the following boundary conditions; when done, click **OK**.

SETTINGS	BOUNDARIES 1, 3	BOUNDARY 2	BOUNDARIES 5, 6, 8–10
	Axial symmetry	Normal acceleration	Sound hard boundary (wall)
a_n	-	`a0`	-

Boundary Integration Variables

Choose **Options>Integration Coupling Variables>Boundary Variables**. Select boundary 2 and define the following boundary integration coupling variable; when done, click **OK**.

NAME	EXPRESSION
P	`2*r*p/a^2`

This variable represents the mean value of the pressure over the piston.

MESH GENERATION

1 Choose **Mesh>Free Mesh Parameters**.

2 On the **Global** page, click the **Custom mesh size** button and in the **Maximum element size** edit field type `0.04`.

3 Click the **Boundary** tab and select Boundaries 2 and 4. In the **Maximum element size** edit field type `0.02`.

4 Click **OK**.

5 Click the **Initialize Mesh** button on the Main toolbar to create the mesh.

COMPUTING THE SOLUTION

1 Choose **Solve>Solver Parameters**.

2 From the **Solver** list select **Parametric**.

3 In the **Parameter name** edit field type `freq_acpr`, and in the **Parameter values** edit field type `10:10:700`.

4 Click the **Stationary** tab, then select **Linear** from the **Linearity** list to ensure that the parametric sweep uses the linear solver.

5 Click **OK**.

6 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

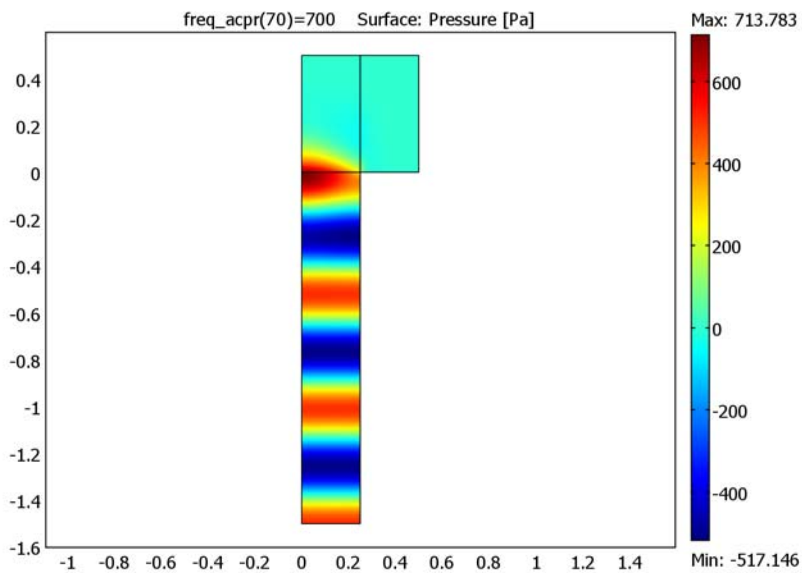


Figure 4-6: The acoustic pressure in the tube and in the PMLs at 700 Hz.

The default plot should look like Figure 4-6. It displays the acoustic pressure in the tube at the final frequency of the sweep, 700 Hz. It also shows that the damping is efficient in the PML regions.

To get a better view of the damping, try plotting the sound level in dB:

1 Choose **Postprocessing>Plot Parameters**.
2 On the **Surface** page choose **Pressure Acoustics (acpr)>Sound pressure level** from the **Predefined quantities** list. Click **OK**.

With a visual inspection you can see that the pressure drop from the open end of the tube to the top boundary of the PML region that makes up the air domain is roughly 50 dB. This means that the part of the wave that is reflected at this boundary experiences a total of 100 dB damping before it returns to the tube. This is more than enough for all practical purposes.

From now on suppress the PMLs and look exclusively at the physical pressure field inside the tube.

1 Choose **Options>Suppress>Suppress Subdomains**.
2 Select Subdomains 2 and 3, then click **OK**.
3 Open the **Plot Parameters** dialog box again and click the **Surface** tab.
4 From the **Predefined quantities** list choose **Pressure Acoustics (acpr)>Pressure**. Click **OK**.

   Note how the pressure field deviates from the simple plane wave behavior in the vicinity of the open end of the tube. Now examine the impedance at the piston.
5 Choose **Postprocessing>Domain Plot Parameters** and click the **Point** tab.
6 Select an arbitrary point in the **Point selection** list. In the **Expression** field, type `real(z0)`, then click **Apply** to see the plot.

   You are now looking at the acoustic resistance as a function of frequency. The peaks show the resonance frequencies for the semi-open tube with a resolution given by the pitch of the frequency sweep.

   You can get the same resonance frequencies with the accuracy improved by the linear interpolation between neighboring frequencies if you instead study the zeros of the reactance plot.
7 Remaining on the **Point** page, enter `imag(z0)` in the **Expression** field and click **Apply** to see the plot.

By zooming in on the zeros in this plot, you should find frequencies similar to those in Table 4-3. To get the same accuracy as in this table, though, you would have to resolve the model with a frequency pitch of 1 Hz.

Finally, plot the real and the imaginary parts of the radiation impedance:

1 Still on the **Point** page, type `real(zL)` in the **Expression** field and click **Apply**.

2 On the **General** page select the **Keep current plot** check box.

3 Click the **Title/Axis** button. In the **Title** edit field type `Radiation resistance (solid) and reactance (dashed)`, and in the **Second axis label** edit field type `(ohm)`. Click **OK**.

4 On the **Point** page click the **Line Settings** button. Set the **Line style** to **Dashed line**. Click **OK**.

5 In the **Expression** field type `imag(zL)`, then click **OK** to reproduce the plot in Figure 4-7.

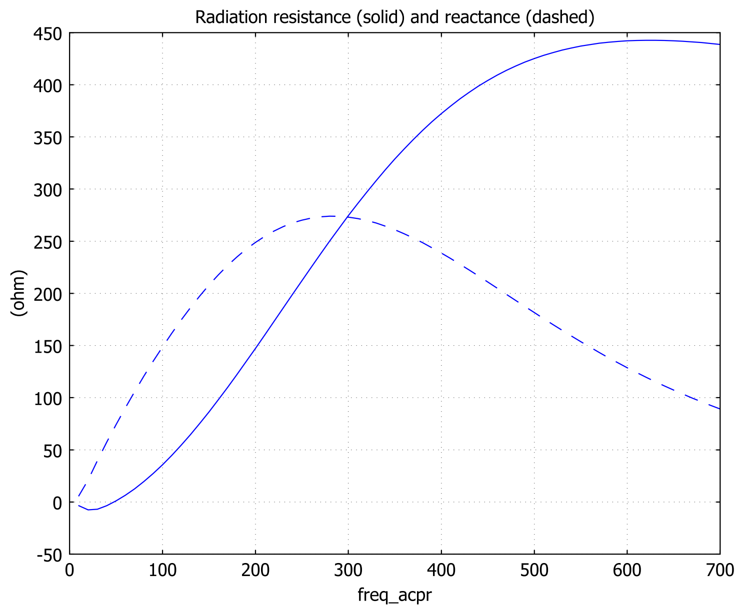


Figure 4-7: Radiation resistance and reactance at the opening of the pipe.

Lumped Impedance Version

In this version of the model, replace the PMLs with an impedance boundary condition using the radiation impedance that you already calculated.

SUBDOMAIN SETTINGS

From the **Physics** menu open the **Subdomain Settings** dialog box. Select Subdomains 2 and 3, clear the **Active in this domain** check box, and then click **OK**.

BOUNDARY CONDITIONS

From the **Physics** menu open the **Boundary Settings** dialog box. Select Boundary 4. In the **Boundary condition** list select **Impedance boundary condition**. Enter `zL` for the **Input impedance**, then click **OK**.

COMPUTING THE SOLUTION

1. Click the **Solver Manager** button on the Main toolbar.
2. Click the **Store Solution** button. In the dialog box that appears, make sure that all frequencies are selected, then click **OK**.
3. In the **Values of variables not solved for and linearization point** list select **Stored solution**.
4. From the **Parameter value** list select **All**.
5. Click **OK**.
6. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The pressure field is now everywhere independent of the *r* coordinate. The impedance boundary condition still leaves the piston impedance unchanged.

1. Choose **Postprocessing>Domain Plot Parameters** and click the **General** tab.
2. Clear the **Keep current plot** check box. Click the **Title/Axis** button. Select the **Auto** buttons for all options, then click **OK** to close the **Title/Axis** dialog box.
3. On the **Point** page, click **Line Settings**. Set the **Line style** to **Solid line**, then click **OK**.
4. In the **Expression** edit field type `real(z0)`. Click **Apply** to see the plot.
5. Enter `imag(z0)` in the **Expression** edit field. Click **OK** to see the plot.

The resistance and reactance plots appear to be virtually identical to those from the solution using PMLs. If you want to compare these results to the analytical solution, follow these steps:

1. Choose **Options>Functions**.
2. In the **Functions** dialog box, click the **New** button.
3. In the **New Function** dialog box, enter the **Function name** `alphaL_ana`.
4. Click the **Interpolation** button. In the **Use data from** list, select **File**.
5. Browse to find the file `alpha.txt`, which is located in the COMSOL Multiphysics installation directory under `/models/Acoustics_Module/Benchmark_Models/`. Click **Open**, then click **OK** to close the **New Function** dialog box.

6 Repeat Steps 2 through 5 to create another interpolation function, but this time use the function name `betaL_ana` and the data file `beta.txt`, which you find in the same folder as `alpha.txt`.

7 Click **OK** to close the **Functions** dialog box dialog box.

8 If you previously entered the scalar expressions manually, now choose **Options>Expressions>Scalar Expressions** and add the following scalar expressions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`alpha0_ana`	`alphaL_ana(k_air*a)`	`Hyp. piston imp. factor`
`beta0_ana`	`betaL_ana(k_air*a)+k_air*L/pi`	`Hyp. piston imp. factor`
`eta0_ana`	`tanh(pi*(alpha0_ana+i*beta0_ana))`	`Normalized piston imp.`
`z0_ana`	`eta0_ana*rho_air*cs_air`	`Piston impedance`

9 Choose **Solve>Update Model**.

10 Choose **Postprocessing>Domain Plot Parameters**.

11 Click the **Point** tab. In the **Expression** edit field type `abs(z0)`, then click **Apply**.

12 On the **General** page select the **Keep current plot** check box.

13 Click the **Title/Axis** button. In the **Title** edit field type `Analytical (dashed) and computed (solid) piston impedance`, and in the **Second axis label** edit field type `(ohm)`. Click **OK**.

14 On the **Point** page click the **Line Settings** button. Set the **Line style** to **Dashed line**, then click **OK**.

15 In the **Expression** field type `abs(z0_ana)`, then click **OK** to generate the plot.

If you zoom in on the range between 100 Hz and 300 Hz, what you get should look like the image in Figure 4-5.

Scattering from a Plate with Ribs

Introduction

The following model has been suggested as a benchmark for acoustic scattering from a thin 3D structure (Ref. 1). Problems of this kind have traditionally been the domain of boundary element (BEM) codes, and the test case was designed with such solvers in mind. The purpose of the present model is to show that scattering problems can be handled very well also by a finite element software like COMSOL Multiphysics, using the built-in PML functionality to reduce the model domain to a minimum.

There is no closed-form reference solution to this benchmark problem, but the presented results correlate very well with the boundary element solutions reported in Ref. 1.

Model Definition

The test geometry consists of a quadratic plate with a pair of stiffening ribs welded on top along the two lines of symmetry. The side length is 60 cm and the ribs are 15 cm high.

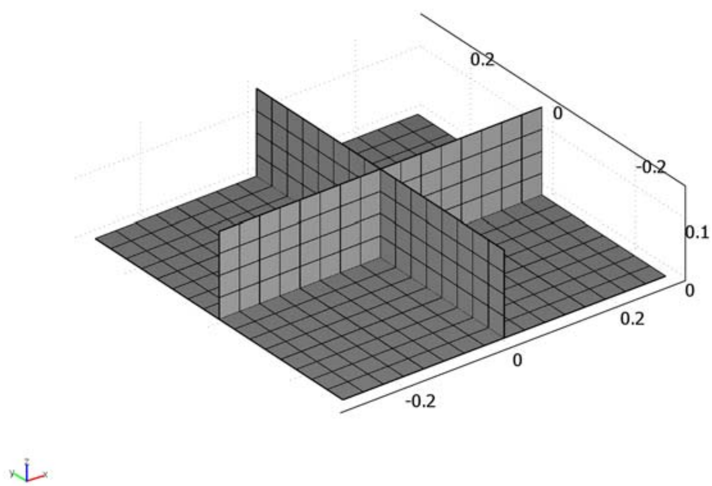


Figure 4-8: Target geometry and surface mesh.

All parts are assumed to have negligible thickness and to be perfectly rigid. You can therefore model the entire structure as boundaries with sound-hard surfaces on both sides. To be able to set separate boundary conditions on the two sides of a boundary, you must create the geometry as an assembly and disconnect the boundaries representing the structure.

Scattering problems are best modeled using the scattered-field formulation. This means that you write the acoustic pressure as the sum of a known incident field, p_i, and an unknown scattered field, p_s. Inserting this sum in the standard acoustic Helmholtz equation and assuming that the incident field by itself is a solution to the same equation, you are left with an equation for the scattered field:

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

Here ρ_0 is the equilibrium density (kg/m^3), $\omega = 2\pi f$ denotes the angular frequency (rad/s), and c_s refers to the speed of sound (m/s). The incident field does not appear in the equation but it modifies the boundary conditions.

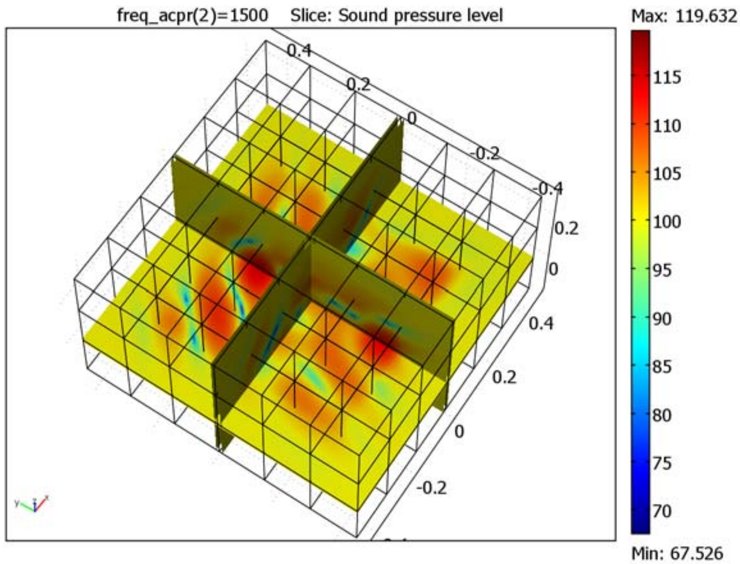
The benchmark case specifies the material properties of air as ρ_0 = 1.225 kg/m^3 and c_s = 340 m/s. Two frequencies are used in the test: 1000 Hz and 1500 Hz. In each case, a plane wave with an intensity of 100 dB SPL is travelling in the (1,–1,–1) direction.

To avoid the scattered waves spuriously reflecting back from the model boundaries onto the target structure, the latter can be padded tightly in PMLs. In this case, it is easiest to define axis-parallel PMLs outside the minimal box around the structure. Note that this means you have a PML directly on the bottom target surface, with no air domain in between, and that there is no reason not to build the model in this way. The PMLs are set to be just one half wavelength thick and three quadratic elements across, which is by far enough to match the accuracy in the reference solutions. For an introduction to using PMLs when modeling with the Acoustics Module, see "Perfectly Matched Layers (PMLs)" on page 73 of the *Acoustics Module User's Guide*.

The entire model can be constructed as a six-by-six-by-three grid of identical cubes. This gives you more subdomains than strictly necessary, but it greatly simplifies the process of creating a structured hexahedral mesh. The final geometry consists of an assembly of six separate parts, which are joined with identity pairs on all internal boundaries that are not part of the target structure. (For an explanation of the terminology concerning assemblies, refer to the section "The Object Properties dialog box for a 3D solid object." on page 76 of the *COMSOL Multiphysics User's Guide*.)

Results and Discussion

Of the 108 domains making up the geometry, 92 are PMLs. The pressure solution inside the PMLs has very little to do with the true scattered field. In the 16 domains representing the actual geometry, the pressure field is correct to the extent the discretization allows.



The reference solutions are given as SPL values along a vertical line from $Z = -2$ to $Z = 2$ situated at $X = 1$, $Y = -1$. Producing these values at an intermediate distance from the target is easy for a boundary element code. However, while solving this benchmark problem in the presented way is straightforward, obtaining numbers comparable to the reference solutions requires some additional work.

The easiest way to evaluate the pressure at an intermediate distance from a radiating object is by using the Helmholtz-Kirchhoff integral representation formula. For the particular case of a thin scatterer this formula simplifies to

$$p_s(\mathbf{R}) = \frac{1}{4\pi}\int_S \frac{e^{-ik|\mathbf{r}-\mathbf{R}|}}{|\mathbf{r}-\mathbf{R}|} p_s(\mathbf{r}) \frac{(1+ik|\mathbf{r}-\mathbf{R}|)}{|\mathbf{r}-\mathbf{R}|^2}(\mathbf{n}\cdot(\mathbf{r}-\mathbf{R}))dv \qquad (4\text{-}3)$$

where k is the wave number, $\mathbf{n}$ is the outward-facing normal vector, and the integral is taken over the target surface, S. Note that the two sides of a thin structure are separate surfaces in the integral, with opposite normal directions. The far-field feature in the Acoustics Module allows you to solve the above integral.

Knowing the scattered pressure field at the probe positions, the total sound pressure level can be calculated as

$$L_p(\mathbf{R}) = 10\log\left(\frac{|p_i(\mathbf{R}) + p_s(\mathbf{R})|^2}{2p_{\mathrm{ref}}^2}\right)$$

where the reference pressure for air is taken to be $p_{\mathrm{ref}} = 2\cdot10^{-5}$ Pa. The results are presented in Table 4-4.

TABLE 4-4: TOTAL SOUND PRESSURE LEVEL FOR VARYING Z AT X=1, Y=-1

Z COORDINATE	1000 HZ	1500 HZ
-2.0	99.2	101.4
-1.6	97.6	96.3
-1.2	100.1	100.0
-0.8	100.3	100.4
-0.4	96.7	97.6
0.0	101.7	98.7
0.4	99.2	99.7
0.8	98.8	101.6
1.2	101.6	102.4
1.6	98.4	100.3
2.0	101.4	99.5

The obtained values are indistinguishable from one of the reference solutions in Ref. 1, with the other being slightly off.

Modeling in COMSOL Multiphysics

You set up this benchmark model using the scattered-field formulation in the Pressure Acoustics application mode. Perfectly matched layers (PMLs) allow you to truncate the domain close to the target geometry. To further reduce memory requirements, you can use a multigrid solver, although direct solvers are faster when applicable. With the far-field variable for the scattered pressure field at the probe positions you can extract the benchmark results.

Reference

1. A.J.Svobodnik, G.Hofstetter & O. von Estorff, "Benchmarks for Radiation and Scattering of Sound," *NAFEMS Ref. -R0083*, 2003.

Model Library path:
`Acoustics_Module/Benchmark_Models/plate_with_ribs`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator** select **3D** from the **Space dimension** list.

2 From the **Application Modes** tree select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis, scattered wave**.

3 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

The geometry consists of 108 identical cubes. You start by creating one such cube and thereafter, step by step, duplicate and copy this single cube until the final geometry is obtained.

1 Choose **Draw>Use Assembly** to put COMSOL Multiphysics in assembly mode.

2 Choose **Draw>Block**. In the dialog box that appears, modify the following entries; when finished, click **OK**.

Length	X	`0.15`
	Y	`0.15`
	Z	`0.15`

3 With the single cube selected, choose **Edit>Copy** or press Ctrl+C to put a copy of the cube on the clipboard.

4 Choose **Edit>Paste** or press Ctrl+V to paste a copy of the cube into the drawing area. Enter the following information into the **Paste** dialog box; when finished click **OK**.

Displacement	x	`-0.3`

	y	-0.3
	z	-0.15

5 Repeat Step 4 to paste a second copy in the same position.

6 Paste one more copy, specifying:

Displacement	x	-0.45
	y	-0.45
	z	-0.15

7 Now select the first block, BLK1, and choose **Draw>Modify>Array** or press the **Array** button on the Draw toolbar. Enter the following data; when finished, click **OK**.

Displacement	x	0.15
	y	0.15
	z	0
Array size	x	2
	y	2
	z	1

8 Select the original block together with the newly created three, then press the **Union** button on the Draw toolbar.

9 With the new composite object still selected, again open the **Array** dialog box and set the following values; when finished, click **OK**.

Displacement	x	-0.3
	y	-0.3
	z	0
Array size	x	2
	y	2
	z	1

10 Return to the second cube created, BLK2, select it and open the **Array** dialog box. Enter the following data; when finished, click **OK**.

Displacement	x	0.15
	y	0.15
	z	0

Array size	x	4
	y	4
	z	1

11 Join the original and the 15 new copies in a single object by selecting them and then clicking the **Union** button.

12 Return to the **Array** dialog box, but with the cube BLK3 as selected source object. Create copies as follows; when finished, click **OK**.

Displacement	x	`0.15`
	y	`0.15`
	z	`0.15`
Array size	x	4
	y	4
	z	2

13 Select the new copies together with BLK3, then click the **Union** button.

14 Select the last of the original four cubes, BLK4, and for the final time open the **Array** dialog box. This time you will create 107 copies.

Displacement	x	`0.15`
	y	`0.15`
	z	`0.15`
Array size	x	6
	y	6
	z	3

15 Add the original cube BLK4 to the selection and click the **Union** button.

16 Finally, add the composite object CO6 to the selection and click the **Difference** button on the Draw toolbar.

17 Before leaving draw mode you must create identity pairs on the objects' common surfaces. Do this by selecting all objects and clicking the **Create Pairs** button.

OPTIONS AND SETTINGS

1 Choose **Options>Constants** and create the following constants; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
Lp_i	100	Incident wave sound pressure level
p0	sqrt(2*2e-5^2*10^(Lp_i/10))	Incident wave amplitude (Pa)
kx	1/sqrt(3)	Incident wave direction, x component
ky	-1/sqrt(3)	Incident wave direction, y component
kz	-1/sqrt(3)	Incident wave direction, z component
X	1	Probe x coordinate
Y	-1	Probe y coordinate
Z	0	Probe z coordinate

PHYSICS SETTINGS

Subdomain Settings

The material properties are the same in all domains, those of air, while the PMLs have different settings depending on in which directions they are absorbing. It is easiest to set the same PML parameters everywhere first, and then disable the damping in directions where it is not needed.

1 Choose **Physics>Subdomain Settings**.

2 Select all domains by pressing Ctrl+A. In the **Fluid density** edit field type `1.225` and in the **Speed of sound** edit field type `340`.

3 With all domains still selected, go to the **PML** page and select **Cartesian** from the **Type of PML** list.

4 For all three directions, select the check box for absorption and change the corresponding **Scaled PML width** to `0.5*cs_acpr/freq_acpr`, which is half the default value. When combined with a radiation boundary condition applied on the outer PML boundary, this setting typically results in sufficient damping while requiring half the number of mesh layers in the PML to resolve the acoustic waves.

5 Now select all subdomains which do not have a boundary face at $x = -0.45$ or at $x = 0.45$ and clear the **Absorbing in x direction** check box. You do this most easily by clicking the **Go to XY view** toolbar button for a top view, then clicking the **Orbit/Pan/Zoom** button to clear it, and finally drawing a rubber-band box around all domains except the leftmost and rightmost columns.

6 Repeat Step 5 for the y direction, that is, clear the **Absorbing in y direction** check box for all domains which do not have a boundary at $y = -0.45$ or $y = 0.45$.

7 Click the **Go to YZ view** toolbar button, use a rubber band to select the middle row of domains and clear the **Absorbing in z** direction check box.

8 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

To define the scattering structure, use the sound-hard boundary condition on the pairs, which sets sound-hard boundary conditions on all boundaries in each pair. Also define the far-field variable for the scattered field at the probe positions (`p_s_probe`).

1 Choose **Physics>Boundary Settings**.

2 Select Boundary 1, then double-click the **Select by group** check box to select all exterior boundaries. From the **Boundary condition** list, select **Radiation condition**. The PMLs work best with a radiation condition on the outside.

3 Click the **Pairs** tab to enable pair selection. Select Pairs 6–13 and **Sound hard boundary (wall)** from the **Boundary condition** list.

4 Click the **Boundaries** tab, then go to the **Far-Field** page.

5 Type `p_s_probe` in the **Name** column of the first table row. When moving to the next cell in the table default values appear in the **Field** and **Normal derivative** columns.

   All the selected boundaries have a sound-hard boundary condition. This means that the normal derivative of the *total* pressure, p_{t}, is zero. Because you use the scattered field formulation, the normal derivative for the pressure variable, $p = p_{\mathrm{t}} - p_{\mathrm{i}}$, is equal to $-\mathbf{n} \cdot \nabla p_{\mathrm{i}}$. The accuracy of the far-field calculation increases if you enter this expression as the normal derivative. However, because the scatterer is thin, you can simplify the problem further by noting that the normal derivative of the incident pressure, p_{i}, is equal in magnitude but of opposite signs for two points on opposite sides of the thin scatterer surface. Therefore, as is expressed in Equation 4-3, the corresponding contributions to the Helmholtz-Kirchhoff integral cancel.

6 In view of the preceding discussion, set the **Normal derivative** to `0`.

7 Set the **Type of integral** to **Full integral**.

8 Click **OK** to close the dialog box.

Application Scalar Variables

The incident wave field is a global quantity defined in the **Application Scalar Variables** dialog box.

1 Choose **Physics>Scalar Variables**.

2 Set the incident pressure wave to `p0*exp(-i*k_acpr*(kx*x+ky*y+kz*z))`.

3 Click **OK** to close the dialog box.

GENERATING THE MESH

1 Click the **Boundary Mode** button on the Main toolbar, then click the **Decrease Mesh Size** button on the Mesh toolbar twice to set the mesh size to **Finer**.

2 Press Ctrl+A to select all boundaries, then click the **Mesh Selected (Mapped)** button to mesh all boundary faces with regular quadrilateral elements.

3 Click the **Subdomain Mode** button on the Main toolbar, then select all subdomains.

4 Click the **Mesh Remaining (Swept)** button to mesh the interior of the model with solid hexahedra.

5 Choose **Mesh>Mesh Statistics** and verify that you have 64,854 degrees of freedom and 6912 hexahedral elements. When done, click **OK**.

COMPUTING THE SOLUTION

On many computers, you can solve the model directly with the default SPOOLES direct symmetric solver. When memory is not a limiting factor, a direct solver is usually the fastest option, but for the sake of illustration, the following steps describe a possible set of iterative solver settings:

1 Choose **Solve>Solver Parameters** or click the corresponding button on the Main toolbar to open the **Solver Parameters** dialog box.

2 Select the **Parametric** solver, set **Parameter name** to `freq_acpr` and set the **Parameter values** to `1000 1500`. If you want to use the default direct solver, you can now jump directly to Step 12.

3 Click the **Settings** button in the **Linear system solver** area to open the **Linear System Solver Settings** dialog box.

4 Select **Geometric multigrid** as **Linear system solver**. New nodes labeled **Presmoother**, **Postsmoother**, and **Coarse solver** appear in the tree to the left. Leave the geometric multigrid settings as they are to create a coarse level by reducing the element order only.

5 Select the **Presmoother** node in the tree and select **GMRES** from the **Presmoother** list.

6 Keep the **Number of iterations** at 2 but change the **Number of iterations before restart** to 2.

7 Expand the **Presmoother** tree node by clicking the plus sign to its left and select the **Preconditioner** node.

8 Select **SSOR** as **Preconditioner** for the GMRES presmoother.

9 Repeat the presmoother settings in Steps 5–8 for the **Postsmoother** node, that is, select GMRES with two iterations before restart as smoother and SSOR as preconditioner for the smoother.

10 The **Coarse solver** setting can be left at the default choice. Using SPOOLES would save some memory by exploiting the matrix symmetries but UMFPACK is faster for small problems.

11 Click **OK** to close the **Linear System Solver Settings** dialog box.

12 Click **OK** to close the **Solver Parameters** dialog box, then click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

The default plot shows the scattered wave pressure on five equidistant slices along the x-axis. To get a better view of the wave pattern, you can change the slice positions:

1 Choose **Postprocessing>Plot Parameters** or click the corresponding button on the Main toolbar.

2 Clear the **Element refinement: Auto** check box and type `3` or `4` in the edit field.

3 On the **Slice** page, select the three option buttons next to the **Vector with coordinates** edit fields in the **Slice positioning** area, then make the following entries:

FIELD	VALUE
x levels	`-0.01 0.01`
y levels	`-0.01 0.01`
z levels	`-0.01 0.01`

4 Select which quantity to visualize among the **Predefined quantities**, for example **Sound pressure level**.

5 Click **OK** to see the plot.

6 For better visibility, turn on some light by pressing the **Headlight** button on the Camera toolbar.

To extract the values in Table 4-4 do the following steps:

The line along which you want the values is not a part of the geometry. Therefore use the line $x = 0, y = 0, -0.1 \le z \le 0.1$. In the expression you use $X = x + 1$, $Y = y - 1$, $Z = 20z$.

1 Choose **Postprocessing>Data Display>Subdomain** to open the **Data Display** dialog box.

2 In the **Expression** field, type `10*log10(0.5*abs(p_s_probe(1,-1,20*z)+ p0*exp(-i*k_acpr*(kx-ky+kz*20*z)))^2/2e-5^2)`.

3 In the **z** edit field type `-2/20:0.4/20:2/20`, select an entry from the **Parameter value** list, and click **Apply**.

INDEX