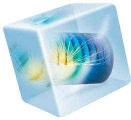


Introduction to Acoustics Module

VERSION 4.3

COMSOL

Introduction to the Acoustics Module

Version: May 2012 COMSOL 4.3

Contact Information

Visit www.comsol.com/contact for a searchable list of all COMSOL offices and local representatives. From this web page, search the contacts and find a local sales representative, go to other COMSOL websites, request information and pricing, submit technical support queries, subscribe to the monthly eNews email newsletter, and much more.

If you need to contact Technical Support, an online request form is located at www.comsol.com/support/contact.

Other useful links include:

- Technical Support www.comsol.com/support
- Software updates: www.comsol.com/support/updates
- Online community: www.comsol.com/community
- Events, conferences, and training: www.comsol.com/events
- Tutorials: www.comsol.com/products/tutorials
- Knowledge Base: www.comsol.com/support/knowledgebase

Part No. CM020204

Contents

Introduction

The Acoustics Module consists of a set of physics interfaces, which enable you to simulate the propagation of sound in fluids and in solids. The interface applications include pressure acoustics, acoustic-solid interaction, aeroacoustics, and thermoacoustics.

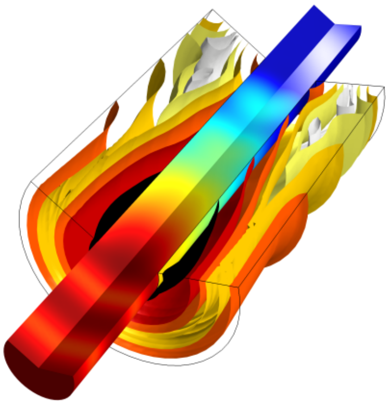


Figure 1: COMSOL model of the sound pressure level distribution in a muffler system.

Acoustic simulations using this module can easily model classical problems such as scattering, diffraction, emission, radiation, and transmission of sound. These problems are relevant to muffler design, loudspeaker construction, sound insulation for absorbers and diffusers, the evaluation of directional acoustic patterns like directivity, noise radiation problems, and much more. The acoustic-structure interaction features can model problems involving structure and fluid-born sound and their interaction. For example, acoustic-interaction is used in detailed muffler design, ultrasound piezo-actuators, sonar technology, and noise and vibration analysis of machinery in the automotive industry. Using the COMSOL Multiphysics capabilities enables the analysis and design of electroacoustic transducers such as loudspeakers, sensors, microphones, and receivers. The Aeroacoustics interface is used to model the interaction between an external flow and an acoustic field, applications range from jet-engine noise analysis to simulating wind sensors.

The Thermoacoustic interface can accurately model systems where small geometrical dimensions are present, which is relevant to the cell phone and hearing aid industries, and for all transducer designers.

Engineers benefit greatly from using the Acoustics Module. By using 3D simulations, existing products can be optimized and new products more quickly designed with virtual prototypes. Simulations also help designers, researchers, and engineers gain insight into problems that are difficult to handle experimentally. And by testing a design before manufacturing it, companies also save time and money.

The many interfaces available with this module are shown in Figure 2 and are available under the Acoustics branch in the Model Wizard. The next section "The Acoustics Module Interfaces" provides you with an overview of the interface functionality.

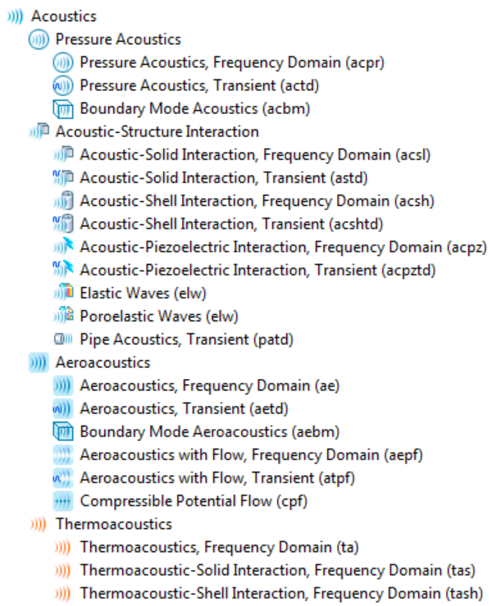


Figure 2: The Acoustics Module physics interfaces. Some interfaces require additional modules—both of the Acoustic-Shell Interaction interfaces and the Thermoacoustic-Shell Interaction interface require the Structural Mechanics Module. The Pipe Acoustics, Transient interface requires the Pipe Flow Module.

There are many application areas where these interfaces are used—from modeling simple pressure waves in air to examining complex interactions between elastic waves and pressure waves in porous materials. For a brief introduction to the basic concepts and theory of acoustics see the "Basics of Acoustics" starting on page 14.

The Acoustics Module Model Library has many examples of applications ranging from modeling sound insulation lining, modeling loudspeakers, microphones, and mufflers. These examples show how to simulate acoustic losses. The loss models range from empirical equivalent-fluid models for fibrous materials to a thermal and viscous loss model using the Thermoacoustics interface.

Another model uses the Aeroacoustics interface to show the influence a flow has on the sound field. Predefined couplings can be used to model the interaction between acoustic, structure, and electric fields in piezoelectric materials. See "Opening the Model Library" on page 12 to access these models. You can also get started with your own model by going to the tutorial "Model Example: Absorptive Muffler" starting on page 18.

The Acoustics Module Interfaces

There are four main branches—Pressure Acoustics, Acoustic-Structure Interaction, Aeroacoustics, and Thermoacoustics—and each of the interfaces are briefly described next, followed by a table on page 10 listing the physics interface availability by space dimension and preset study types.

Pressure Acoustics

The Pressure Acoustics branch ( ) has interfaces where the sound field is described and solved by the pressure p. The pressure represents the acoustic variations (or excess pressure) to the ambient stationary pressure. The ambient pressure is, in the absence of flow, simply the static absolute pressure.

The interfaces enable solving the acoustic problem both in the frequency domain using the Pressure Acoustics, Frequency Domain interface ( ), where the Helmholtz equation is solved, and as a transient system using the Pressure Acoustics, Transient interface ( ), where the classical wave equation is solved. The Boundary Mode Acoustics interface ( ) is used to study propagating modes in waveguides and ducts (only a finite set of shapes, or modes can propagate over a long distance).

A large variety of boundary conditions are available and include hard walls and impedance conditions, radiation, symmetry, and periodic conditions for modeling open boundaries, and conditions for applying sources. The interfaces also have several equivalent-fluid models, which mimic the behavior of sound propagation in more complex media such as porous materials, fibrous materials, as well as viscous and thermally conducting fluids. So-called perfectly matched layers (PMLs) are also available to truncate the computational domain. Finally, the far-field feature can be used to determine the pressure outside the computational domain. Dedicated results and analysis capabilities exist to visualize the far-field in 2D and 3D polar plots.

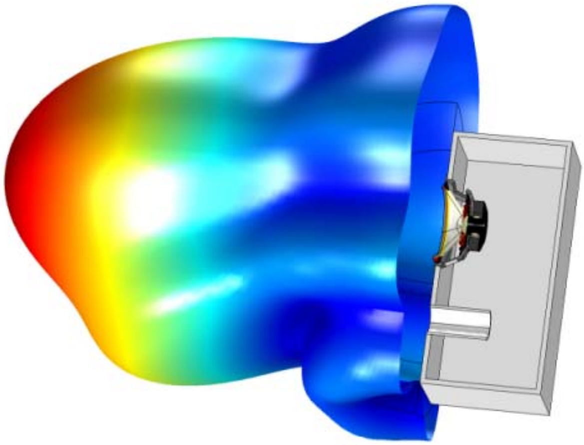


Figure 3: A 3D far-field polar plot of the loudspeaker sensitivity at 3000 Hz. From the Vented Loudspeaker Enclosure model found in the Model Library.

Acoustic-Structure Interaction

The Acoustic-Structure Interaction branch ( ) has interfaces that apply to a multiphysics phenomenon where the fluid's pressure causes a fluid load on the solid domain, and the structural acceleration affects the fluid domain as a normal acceleration across the fluid-solid boundary. The interfaces under this branch are the Acoustic-Solid Interaction, Frequency Domain ( ), the Acoustic-Solid Interaction, Transient ( ), the Acoustic-Shell Interaction, Frequency Domain ( ), the Acoustic-Shell Interaction, Transient ( ), the Acoustic-Piezoelectric Interaction, Frequency Domain ( ), and the Acoustic-Piezoelectric Interaction, Transient ( ) interfaces. The two Acoustic-Piezoelectric Interaction interfaces also support solving and modeling the electric field in a piezoelectric material. The piezoelectric coupling can be in stress-charge or strain-charge form.

The Pipe Acoustics, Transient interface ( )is available with the addition of the Pipe Flow Module. The interface is used for modeling the propagation of transient sound waves in 1D flexible pipe systems. The equations are formulated in a general way to include the possibility of a stationary background flow.

Two more interfaces are available under this branch—Elastic Waves ( ), which models elastic waves in solids, and Poroelastic Waves ( ), which precisely models the propagation of sound in a porous material, including the two way coupling between deformation of the solid matrix and the pressure waves in the saturating fluid.

All these interfaces are available for 3D, 2D, and 2D axisymmetric geometries and can model pressure acoustics and solid mechanics in the frequency domain and as a transient problem.

Aeroacoustics

The interaction of an acoustic field with fluid flow is modeled using the interfaces found under the Aeroacoustics branch ( ). The coupling between the fluid mechanics and the acoustics is based on the potential formulation of the fluid flow and acoustic equations. The Aeroacoustics, Frequency Domain ( ) and the Aeroacoustics, Transient ( ) interfaces model the interaction of a static background flow with the acoustic field. The Compressible Potential Flow interface ( ) models the flow of an inviscid, compressible fluid having no vorticity, as it is irrotational by nature. The Aerocaoustics with Flow, Frequency Domain interface ( ) and the Aeroacoustics with Flow, Transient interface ( ) both model the coupled acoustic and potential flow problem in the frequency domain or as a transient system. The Boundary Mode Aeroacoustics interface ( ) is used to study boundary mode acoustic problems in a background flow field.

Thermoacoustics

The interfaces under the Thermoacoustics branch ( ) are used to accurately model acoustics in geometries with small dimensions. Near walls, viscosity, and thermal conduction become important because a viscous and a thermal boundary layer are created, resulting in an acoustic boundary layer where losses are significant, making it necessary to include thermal conduction effects and viscous losses explicitly in the governing equations.

Because detailed descriptions are needed to model thermoacoustics, all the interfaces simultaneously solve for the acoustic pressure p, the particle velocity vector $\mathbf{u}$, and the acoustic temperature variations T.

In the Thermoacoustics, Frequency Domain interface ( ) the governing equations are implemented in the time harmonic formulation and solved in the frequency domain. Both mechanical and thermal boundary conditions exist. Coupling the thermoacoustic domain to a pressure acoustic domain is also straight forward with a predefined boundary condition.

A Thermoacoustic-Solid Interaction, Frequency Domain interface ( ) is available to solve coupled vibro-acoustic problems. This can, for example, be used to model small electroacoustic transducers. Predefined boundary conditions exist between solid domains and fluid domains.

The Thermoacoustic-Shell Interaction, Frequency Domain interface ( ) is used for modeling the interaction between shells and acoustics in small dimensions. This could be used to analyze the damped vibrations of shells in hearing aids and prevent feedback problems.

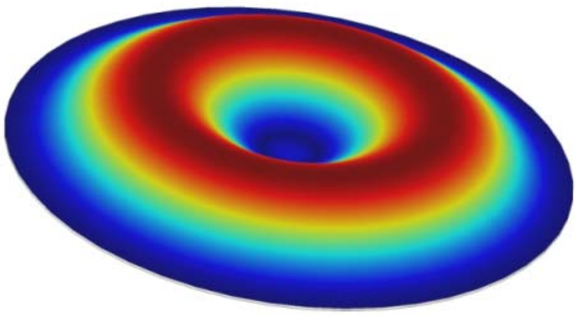


Figure 4: Deformation of the diaphragm (or membrane) at 12 kHz in the Axisymmetric Condenser Microphone model found in the Model Library.

Physics List by Space Dimension and Study Type

The table below list the interfaces available specifically with this module in addition to the COMSOL Multiphysics basic license.

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Acoustics				
Pressure Acoustics				
Pressure Acoustics, Frequency Domain*		acpr	all dimensions	eigenfrequency; frequency domain; frequency-domain modal; mode analysis (2D and 1D axisymmetric models only)
Pressure Acoustics, Transient		actd	all dimensions	eigenfrequency; frequency domain; frequency-domain modal; time dependent; time-dependent modal

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Boundary Mode Acoustics		acbm	3D, 2D axisymmetric	mode analysis
Acoustic-Structure Interaction				
Acoustic-Solid Interaction, Frequency Domain		acsl	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Acoustic-Solid Interaction, Transient		astd	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal; time dependent; time-dependent modal
Acoustic-Shell Interaction, Frequency Domain Requires a Structural Mechanics Module license		acsh	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Acoustic-Shell Interaction, Transient Requires a Structural Mechanics Module license		acshtd	3D	eigenfrequency; frequency domain; frequency-domain modal; time dependent; time-dependent modal
Acoustic-Piezoelectric Interaction, Frequency Domain		acpz	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Acoustic-Piezoelectric Interaction, Transient		acpztd	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal; time dependent; time-dependent modal
Elastic Waves		elw	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Poroelastic Waves		elw	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Pipe Acoustics, Transient Requires both the Pipe Flow Module and Acoustics Module.		patd	3D, 2D	time dependent
Aeroacoustics				
Aeroacoustics, Frequency Domain		ae	all dimensions	frequency domain; mode analysis (2D and 1D axisymmetric models only)
Aeroacoustics, Transient		aetd	all dimensions	frequency domain; time dependent

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Boundary Mode Aeroacoustics		aebm	3D, 2D axisymmetric	mode analysis
Aeroacoustics with Flow, Frequency Domain		aepf	all dimensions	frequency domain
Aeroacoustics with Flow, Transient		atpf	all dimensions	frequency domain; time dependent
Compressible Potential Flow		cpf	all dimensions	stationary; time dependent
Thermoacoustics				
Thermoacoustics, Frequency Domain		ta	all dimensions	eigenfrequency; frequency domain; frequency domain modal; mode analysis (2D and 1D axisymmetric models only)
Thermoacoustic-Solid Interaction, Frequency Domain		tas	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain modal
Thermoacoustic-Shell Interaction Requires a Structural Mechanics Module license		tash	3D	eigenfrequency; frequency domain; frequency domain modal
Structural Mechanics				
Solid Mechanics*		solid	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent
Piezoelectric Devices		pzd	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; time-dependent modal; frequency domain; frequency domain modal

* This is an enhanced interface, which is included with the base COMSOL package but has added functionality for this module.

Opening the Model Library

To open an Acoustics Module Model Library model, select **View > Model Library** from the main menu in COMSOL Multiphysics. In the Model Library window that opens, expand the Acoustics Module folder and browse or search the contents. Click **Open Model and PDF** to open the model in COMSOL Multiphysics and a PDF to read background theory about the model including the step-by-step instructions to build it.

The MPH-files in the COMSOL model libraries can have two formats—Full MPH-files or Compact MPH-files.

- Full MPH-files, including all meshes and solutions. In the Model Library these models appear with the icon. If the MPH-file's size exceeds 25MB, a tip with the text "Large file" and the file size appears when you position the cursor at the model's node in the Model Library tree.
- Compact MPH-files with all settings for the model but without built meshes and solution data to save space on the DVD (a few MPH-files have no solutions for other reasons). You can open these models to study the settings and to mesh and re-solve the models. It is also possible to download the full versions—with meshes and solutions—of most of these models through Model Library Update. In the Model Library these models appear with the icon. If you position the cursor at a compact model in the Model Library window, a No solutions stored message appears. If a full MPH-file is available for download, the corresponding node's context menu includes a Model Library Update item.

The Absorptive Muffler model tutorial starts on page 18 and the last section, starting on page 46, provides a brief overview of some of the other models also available in the Acoustics Module Model Library.

Basics of Acoustics

Acoustics is the physics of sound. Sound is the sensation, as detected by the ear, of very small rapid changes in the acoustic pressure p above and below a static value p_0. This static value is the atmospheric pressure p_0 (about 100,000 pascals). The wave crests are the pressure maxima while the troughs represent the pressure minima.

Sound is created when the air is disturbed by a source. An example is a vibrating object, such as a speaker cone in a sound system. It is possible to see the movement of a bass speaker cone when it generates sound at a very low frequency. As the cone moves forward it compresses the air in front of it, causing an increase in air pressure. Then it moves back past its resting position and causes a reduction in air pressure. This process continues, radiating a wave of alternating high and low pressure at the speed of sound.

The frequency f (SI unit: Hz = 1/s) is the number of vibrations (pressure peaks) perceived per second and the wavelength λ (SI unit: m) is the distance between two such peaks. The speed of sound c (SI unit: m/s) is given as the product of the frequency and the wavelength, $c = \lambda f$. It is often convenient to define the angular frequency ω (SI unit: rad/s) of the wave which is $\omega=2\pi f$, and relates the frequency to a full $360^{\mathbf{o}}$ phase shift. The wave number k (SI unit: rad/m) is defined as $k=2\pi/\lambda$. The wave number is also usually defined as a vector $\mathbf{k}$, such that it also contains information about the direction of propagation of the wave, with $|\mathbf{k}|=k$.

Governing Equations

The equations that describe the propagation of sound in fluids are derived from the governing equations of fluid flow. That is, conservation of mass, which is described by the continuity equation—the conservation of momentum that is often referred to as the Navier-Stokes equation—an energy conservation equation, and an equation of state to describe the relation between thermodynamic variables. In the classical case of pressure acoustics, which describes most acoustic phenomena accurately, the flow is assumed lossless, viscous effects are neglected, and a linearized isentropic equation of state is used.

Under these assumptions the acoustic field is described by one variable, the pressure p (SI unit: Pa), and is governed by the wave equation

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

where t is time (SI unit: s), ρ_0 is the density of the fluid (SI unit: kg/m^3), and $\mathbf{q}$ and Q are possible acoustic dipole and monopole source terms (SI units: N/m^3 and 1/s^2, respectively).

Acoustic problems often involve simple harmonic waves such as sinusoidal waves. More generally, any signal may be expanded into harmonic components via its Fourier series. The wave equation can then be solved in the frequency domain for one frequency at a time. A harmonic solution has the form

$$p(\mathbf{x})\sin(\omega t)$$

where the spatial $p(\mathbf{x})$ and temporal $\sin(\omega t)$ components are split. The pressure may be written in a more general way using complex variables

$$p = p(\mathbf{x})e^{i\omega t} \tag{1}$$

where the actual physical value of the pressure is the real part of Equation 1. Using this assumption for the pressure field, the time dependent wave equation reduces to the well-known Helmholtz equation

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

In the homogenous case where the two source terms $\mathbf{q}$ and Q are zero, one simple solution to the Helmholtz equation (Equation 2) is the plane wave

$$p = P_0 e^{i(\omega t-\mathbf{k}\cdot\mathbf{x})} \tag{3}$$

where P_0 is the wave amplitude and it is moving in the $\mathbf{k}$ direction with angular frequency ω and wave number $k = |\mathbf{k}|$.

Length and Time Scales

When solving acoustic problems it is important to think about the different basic length and time scales involved in the system. Some of the scales are set by the physics of the problem while others are set by the numerical solution method. The relative size of these scales may influence the accuracy of the solution, but also the selection of a physics interface to model the problem.

When working with acoustics in the frequency domain, that is, solving the Helmholtz equation, only one time scale exists and it is set by the frequency $T = 1/f$. Several length scales exist: the wavelength $\lambda = c/f$, the smallest geometrical dimension $L_{\min}$, the mesh size h, and the thickness of the acoustic boundary layer δ. The latter is discussed in "Models with Losses" on page 17. In order to get an accurate solution the mesh should be fine enough to both resolve the geometric features and the wavelength. As a rule of thumb the maximal mesh size should be less or equal to λ/N, where N is between 5 and 10.

For transient acoustic problems the same considerations apply. However, several new time scales are also introduced. One is given by the frequency contents of the signal and by the desired maximal frequency resolution: $T = 1/f_{\max}$. The other is given by the size of the time step Δt used by the numerical solver. A condition on the so-called CFL number dictates the relation between the time step size and the minimal mesh size $h_{\min}$. The CFL number is defined as

$$\mathrm{CFL} = \frac{c\Delta t}{h_{\min}} \tag{4}$$

where c is the speed of sound in the system. It is good practice to have CFL numbers of approximately 0.2. More information is included with the "Gaussian Explosion" tutorial model described on page 46.

In order to run accurate acoustic simulations it is important to think about these physical and numerical scales and about its influence on convergence and correctness of the numerical solution. A good practical approach is to test the robustness of a solution compared to changes in the mesh in all cases and the numerical time stepping in the transient problems.

Boundary Conditions

Boundary conditions define the nature of the boundaries of the computational domain. Some define real physical obstacles like a sound hard wall or a moving interface. Others, called artificial boundary conditions, are used to truncate the domain. The artificial boundary conditions are, for example, used to simulate an open boundary where no sound is reflected. It may also mimic a reacting boundary such as a perforated plate.

Elastic Waves

The propagation of sound in solids happens through small-amplitude elastic oscillations of its shape. These elastic waves are transmitted to surrounding fluids as ordinary sound waves. Through acoustic-structure interaction, the fluid's pressure causes a fluid load on the solid domain, and the structural acceleration affects the fluid domain as a normal acceleration across the fluid-solid boundary.

The Acoustics Module has the Poroelastic Waves interface available to model poroelastic waves that propagate in porous materials. These waves result from the complex interaction between pressure variations in the saturating fluid and the elastic deformation of the solid porous matrix.

Models with Losses

In order to accurately model acoustics in geometries with small dimensions, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations. Near walls, viscosity, and thermal conduction become important because the acoustic field creates a viscous and a thermal boundary layer where losses are significant. As detailed descriptions are needed to model thermoacoustics, the dedicated Thermoacoustics, Frequency Domain interface simultaneously solves for the acoustic pressure p, the particle velocity vector $\mathbf{u}$, and the acoustic temperature variations T.

The length scale at which the thermoacoustic description is necessary is given by the thickness of the viscous (v) and thermal (th) boundary layers

$$\delta_{\mathrm{v}} = \sqrt{\frac{\mu}{\pi f \rho}} \qquad \delta_{\mathrm{th}} = \sqrt{\frac{\mathrm{k}}{\pi f \rho C_{\mathrm{p}}}}$$

where μ is the dynamic viscosity, k is the coefficient of thermal conduction, and C_{p} is the specific heat capacity at constant pressure. These two length scales define an acoustic boundary layer that needs to be resolved by the computational mesh.

Another way to introduce losses in the governing equations is to use the equivalent fluid models available in the pressure acoustics interfaces. This introduces attenuation properties only to the bulk fluid in contrast to the thermoacoustics interfaces used to model losses including those in the acoustic boundary layer near walls. The fluid models include losses due to thermal conduction and viscosity, models for simulating the damping in certain porous materials, and macroscopic empirical models for certain fibrous materials. When applicable, the equivalent fluid models are computationally much less heavy than, for example, solving a corresponding full poroelastic model.

Model Example: Absorptive Muffler

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

The purpose of this model is to show how to analyze both inductive and resistive damping in pressure acoustics. The main output is the transmission loss for the frequency range 50 Hz–1500 Hz.

Model Definition

The muffler, schematically shown in Figure 5, 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.

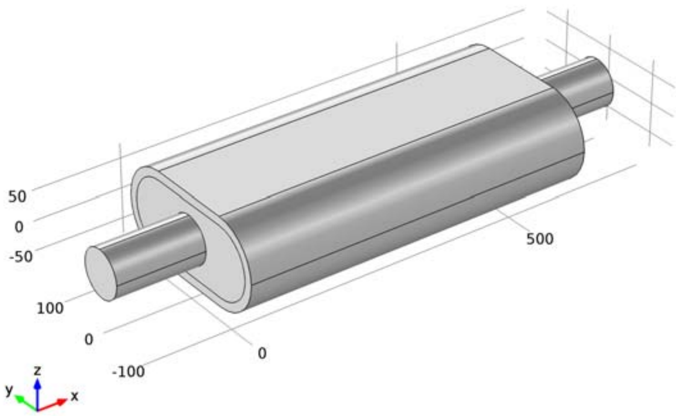


Figure 5: Geometry of the lined muffler with the upper half removed. The exhaust fumes enter through the left pipe (inlet) and exit through the right pipe (outlet).

Domain Equations

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

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

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 Delany and Bazley model 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 $k_a = \omega/c_a$ 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 lightweight glass wool with $\rho_{ap} = 12$ kg/m^3 and $d_{av} = 10$ μm.

Boundary Conditions

There are three boundary conditions used in this model.

- At the solid boundaries, which are the outer walls of the resonator chamber and the pipes, the model uses sound hard (wall) boundary conditions. The condition imposes that the normal velocity at the boundary is zero, and is specified mathematically by:

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

- The boundary condition at the inlet involves a combination of an incoming imposed plane wave and an outgoing radiating plane wave. Mathematically it is formulated as:

$$\mathbf{n} \cdot \frac{1}{\rho_0}\nabla p + ik\frac{p}{\rho_0} + \frac{i}{2k}\Delta_T p = \left(\frac{i}{2k}\Delta_T p_0 + (1-(\mathbf{k}\cdot\mathbf{n}))ik\frac{p_0}{\rho_0}\right)e^{-ik(\mathbf{k}\cdot\mathbf{r})} \qquad (5)$$

  In Equation 5, 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 radiating plane wave:

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

Results and Discussion

Figure 6 shows the isosurface plot for the total acoustic pressure field.

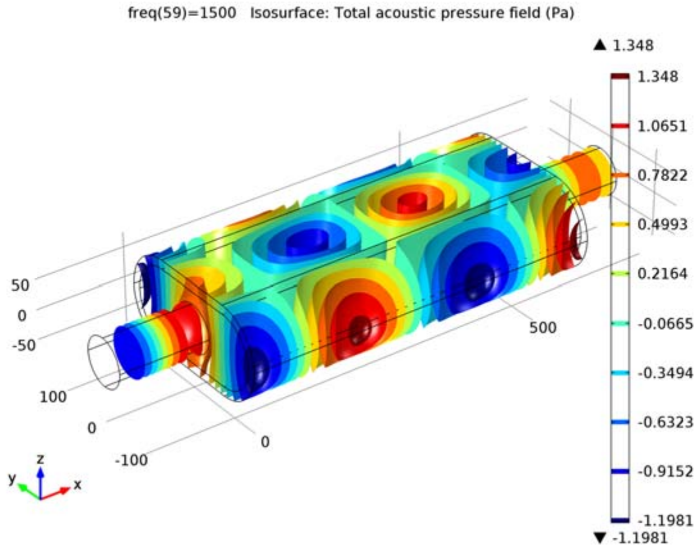


Figure 6: The pressure distribution in the absorptive muffler without the liner is shown for f = 1500 Hz.

The pressure is represented as iso-barometric surfaces in the muffler. From the figure it is clear that at this frequency longitudinal standing waves exist as well as transverse modes.

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

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

Here w_{in} and w_{out} denote the incoming power at the inlet and the outgoing power at the outlet, respectively. Each of these quantities can be calculated as an integral over the corresponding surface:

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

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

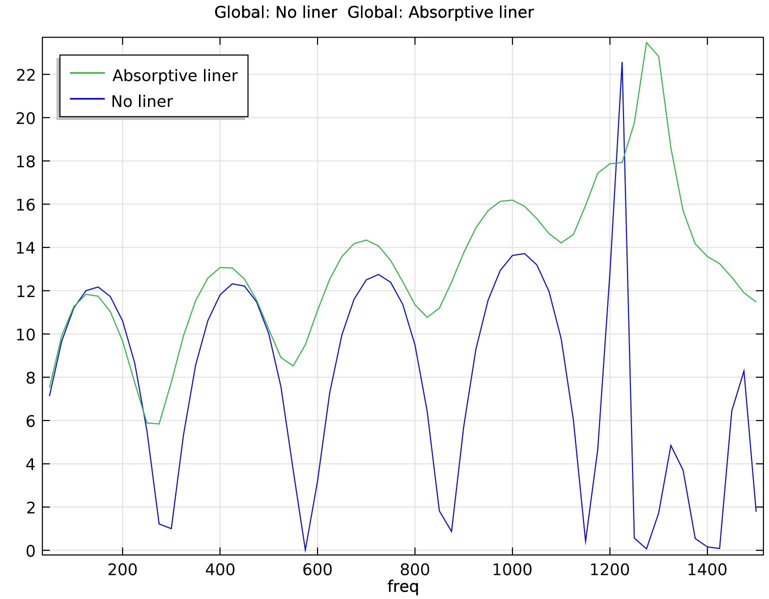


Figure 7: Attenuation (dB) in the empty muffler as a function of frequency. The first four dips are due to longitudinal resonances. In the muffler with absorbing liner the dips are still present, but the general trend is that the higher the frequency, the better the damping.

Figure 7 shows the result of a parametric frequency study. The two graphs represent the case of an empty muffler without any absorbing liner material (blue line) and the case with a layer of glass wool lining on the chamber's walls (green line).

The graph, for the undamped muffler, 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. 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. Not very far above this frequency, a range of modes that are combinations of this propagation mode and the longitudinal modes participate, making the damping properties increasingly unpredictable. For an analysis of these modes, see "Eigenmodes in Muffler" described on page 46. The glass wool lining improves the attenuation at the resonance frequencies as well as at higher frequencies.

References

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

2. D. A. Bies and C. H. Hansen, "Flow Resistance Information for Acoustical Design," *Appl. Acoust.*, vol. 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 WIZARD

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

1. Open COMSOL Multiphysics.
2. In the **Model Wizard**, the default **Space Dimension** selected is **3D**, Click **Next** .
3. On the **Add physics** page under **Acoustics>Pressure Acoustics**, double-click to select **Pressure Acoustics, Frequency Domain (acpr)** and add it to the **Selected physics** list.
4. Click **Next** .

5 On the **Select Study Type** page under **Studies>Preset Studies**, select **Frequency Domain** . Click **Finish** .

GLOBAL DEFINITIONS

Note: The location of the text files used in this exercise vary based on the installation. For example, if the installation is on your hard drive, the file path might be similar to `C:\Program Files\COMSOL43\models\`.

Parameters

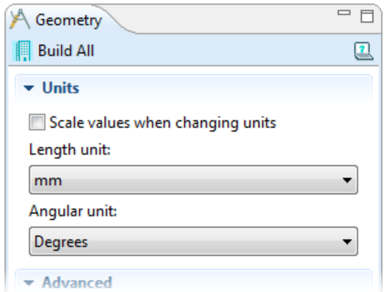
1 In the **Model Builder**, right-click **Global Definitions** and choose **Parameters** .
2 Go to the **Parameters** settings window. Under **Parameters** click **Load from File** .
3 Browse to the model's Model Library folder (**Acoustics_Module\Industrial Models**) and double-click the file `absorptive_muffler_parameters.txt`.

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

Parameters

Name	Expression	Value	Description
p0	1[Pa]	1.0000 Pa	Amplitude of incoming pressure wave
rho_ap	12[kg/m^3]	12.000 kg/m^3	Apparent density of glass wool
d_av	10[um]	1.0000E-5 m	Mean fiber diameter
R_f	3.18e-9[N*s/m...	1424.2 kg/(...	Flow resistivity
L	600[mm]	0.60000 m	Muffler length
H	150[mm]	0.15000 m	Muffler height
W	300[mm]	0.30000 m	Muffler width
L_io	150[mm]	0.15000 m	Inlet and outlet length
R_io	40[mm]	0.040000 m	Inlet and outlet radius
D	15[mm]	0.015000 m	Liner thickness

GEOMETRY 1

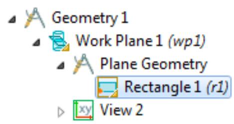


1. In the **Model Builder** under **Model 1**, click **Geometry 1**.
2. Go to the **Geometry** settings window. Under **Units** from the **Length unit** list, choose **mm**.

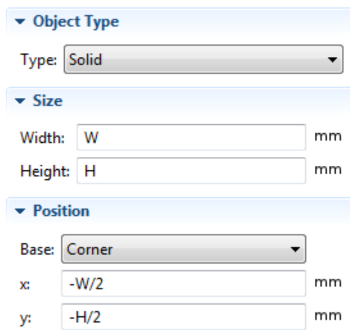
Work Plane 1

1. Under **Model 1**, right-click **Geometry 1** and choose **Work Plane**.
2. Go to the **Work Plane** settings window. Under **Work Plane** from the **Plane** list, choose **yz-plane**.

Rectangle 1



1. In the **Model Builder**, under **Model 1>Geometry 1>Work Plane 1**, right-click **Plane Geometry** and choose **Rectangle**. A **Rectangle** node is added to the **Model Builder**.
2. Go to the **Rectangle** settings window. Under **Size** in the:
   - **Width** field, enter `W`.
   - **Height** field, enter `H`.
3. Under **Position** in the:
   - **xw** field, enter `-W/2`.
   - **yw** field, enter `-H/2`.

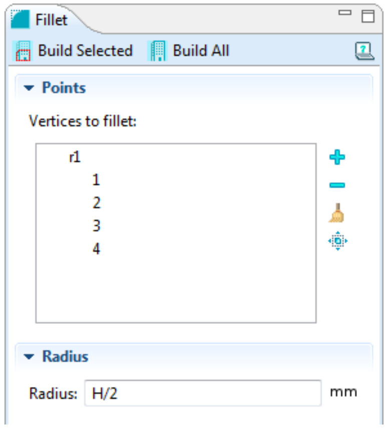


Fillet 1

1. In the **Model Builder** under **Model 1>Geometry 1>Work Plane 1**, right-click **Plane Geometry** and choose **Fillet**.
2. Click the **Zoom Extents** button on the **Graphics** toolbar.

3 On the object **r1**, select Points `1,2,3,4` only.

4 Go to the **Fillet** settings window. Under **Radius** in the **Radius** field, enter `H/2`.



Rectangle 2

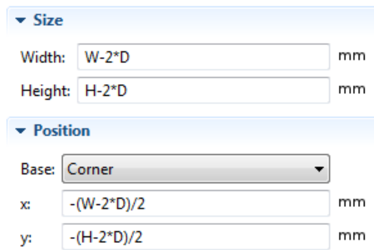
1 In the **Model Builder** under **Model 1>Geometry 1>Work Plane 1**, right-click **Plane Geometry** and choose **Rectangle**.

2 Go to the **Rectangle** settings window. Under **Size** in the:

- **Width** field, enter `W-2*D`.
- **Height** field, enter `H-2*D`.

3 Under **Position** in the:

- **xw** field, enter `-(W-2*D)/2`.
- **yw** field, enter `-(H-2*D)/2`.



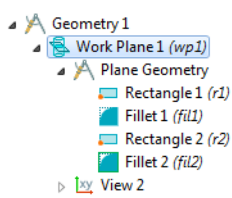
Fillet 2

1 In the **Model Builder** under **Model 1>Geometry 1>Work Plane 1**, right-click **Plane Geometry** and choose **Fillet**.

2 On the object **r2**, select Points `1,2,3,4` only.

3 Go to the **Fillet** settings window. Under **Radius** in the **Radius** field, enter `(H-2*D)/2`.

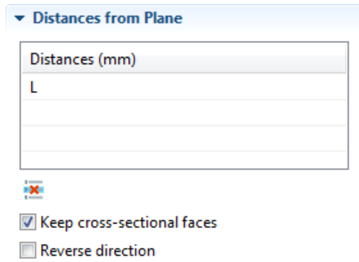
4 Click the **Build All** button. The node sequence under **Work Plane 1** should match this figure at this point:



Extrude 1

1 In the **Model Builder** under **Model 1>Geometry 1**, right-click **Work Plane 1** and choose **Extrude**.

2 Go to the **Extrude** settings window. Under **Distances from Plane** in the table under **Distances (mm)**, enter `L` in the column.



Cylinder 1

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

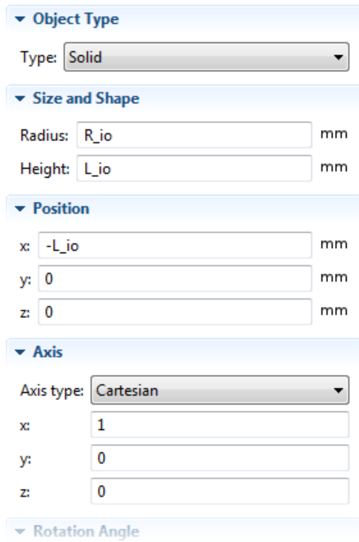
2 Go to the **Cylinder** settings window. Under **Size and Shape** in the:

- **Radius** field, enter `R_io`.
- **Height** field, enter `L_io`.

3 Under **Position** in the **x** field, enter `-L_io`.

4 Under **Axis** in the:

- **x** field, enter `1`.
- **z** field, enter `0`.



Cylinder 2

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

2 Go to the **Cylinder** settings window. Under **Size and Shape** in the:

- **Radius** field, enter `R_io`.
- **Height** field, enter `L_io`.

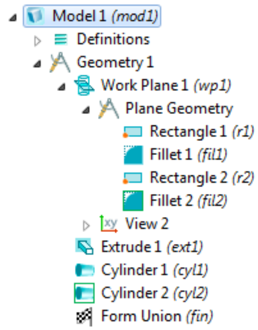
3 Under **Position** in the **x** field, enter `L`.

4 Under **Axis** in the:

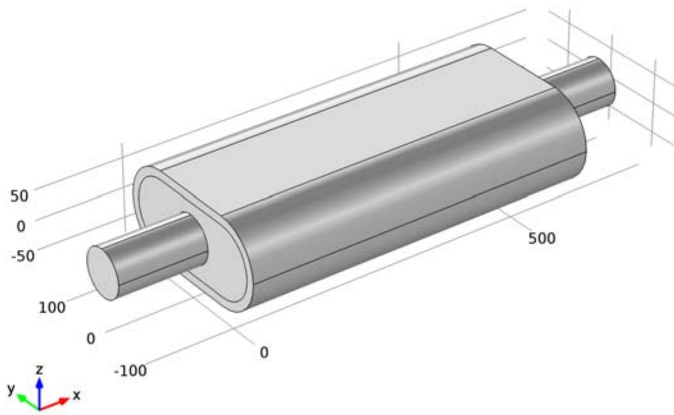
- **x** field, enter `1`.
- **z** field, enter `0`.

5 Click the **Build All** button and then click the **Zoom Extents** button on the **Graphics** toolbar.

The node sequence in the **Model Builder** should match the figure.



The finished geometry is shown in the figure below, the axis dimensions are in mm, as selected, and the system of coordinates is also shown.



DEFINITIONS

Explicit 1

1 In the **Model Builder** under **Model 1**, right-click **Definitions** and choose **Selections>Explicit**.
2 Under **Model 1>Definitions** right-click **Explicit 1** and choose **Rename** (or press F2).
3 Go to the **Rename Explicit** dialog box and enter `Inlet` in the **New name** field.
4 Click **OK**.

5 Go to the **Explicit** settings window. Under **Input Entities** from the **Geometric entity level** list, choose **Boundary**.

6 Select **Boundary** 1 only.

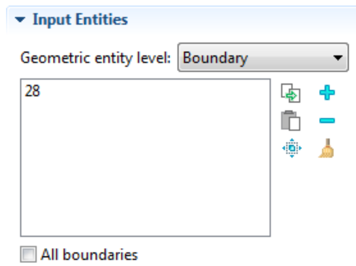
Explicit 2

1 In the **Model Builder**, right-click **Definitions** and choose **Selections>Explicit**.

2 Click **Explicit 2** and press F2. In the **Rename Explicit** dialog box, enter `Outlet` as the **New name**. Click **OK**.

3 Go to the **Explicit** settings window. Under **Input Entities** from the **Geometric entity level** list, choose **Boundary**.

4 Select **Boundary** 28 only.

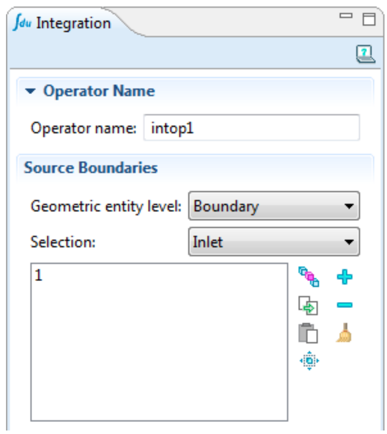


Integration 1

1 In the **Model Builder**, right-click **Definitions** and choose **Model Couplings>Integration**.

2 Go to the **Integration** settings window. Under **Source Boundaries** from the **Geometric entity level** list, choose **Boundary**.

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

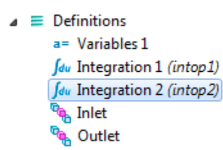


Integration 2

1 In the **Model Builder**, right-click **Definitions** and choose **Model Couplings>Integration**. An **Integration 2** node is added to the **Model Builder**.

2 Go to the **Integration** settings window. Under **Source Boundaries** from the **Geometric entity level** list, choose **Boundary**.

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



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

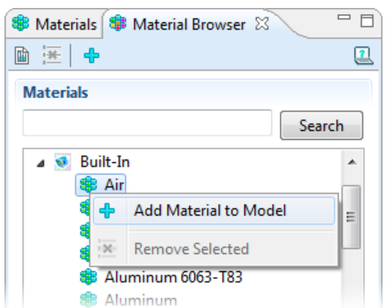
Variables 1

1. In the **Model Builder**, right-click **Definitions** and choose **Variables**.
2. Go to the **Variables** settings window. Under **Variables** in the table, enter the following settings:

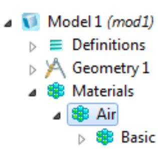
NAME	EXPRESSION	DESCRIPTION
`w_in`	`intop1(p0^2/(2*acpr.rho*acpr.c))`	`Power of the incoming wave`
`w_out`	`intop2(abs(p)^2/(2*acpr.rho*acpr.c))`	`Power of the outgoing wave`

MATERIALS

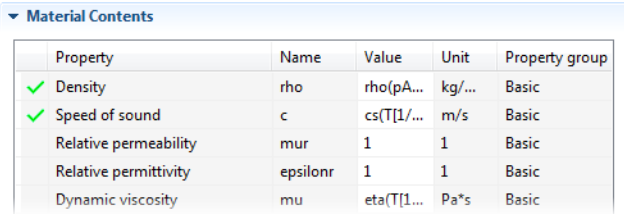
1. From the main menu, select **View>Material Browser**.
2. In the **Material Browser** window, under **Built-In** right-click **Air** and select **Add Material to Model**.



3. Click the **Air** node under **Materials** to look at the **Material Contents** in the settings window.



The green check marks indicate which material parameters are necessary in the model.



Material Contents

	Property	Name	Value	Unit	Property group
✓	Density	rho	rho(pA...	kg/...	Basic
✓	Speed of sound	c	cs(T[1/...	m/s	Basic
	Relative permeability	mur	1	1	Basic
	Relative permittivity	epsilonr	1	1	Basic
	Dynamic viscosity	mu	eta(T[1...	Pa*s	Basic

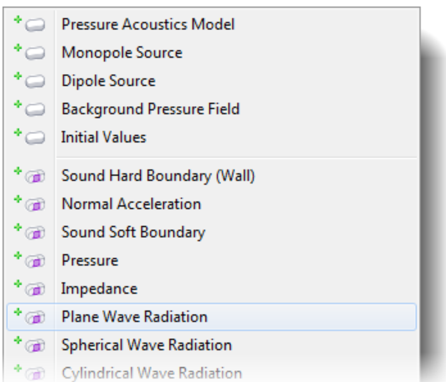
Note: By default the first material added applies to all domains so the geometric scope settings do not need to be changed.

In the second version of this model, a lining material is inserted in Domain 2. For now, the muffler is completely hollow.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN

Plane Wave Radiation 1

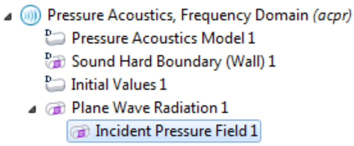
1 In the **Model Builder** under **Model 1**, right-click **Pressure Acoustics, Frequency Domain** and from the boundary level, choose **Plane Wave Radiation** .



2 Select Boundaries **1** and **28** only.

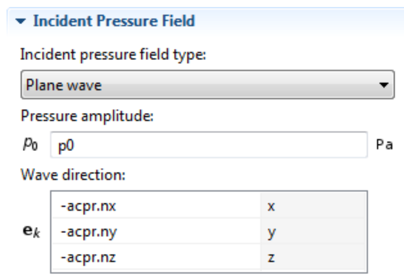
Incident Pressure Field 1

1 Under **Pressure Acoustics, Frequency Domain**, right-click **Plane Wave Radiation 1** and choose **Incident Pressure Field**. An **Incident Pressure Field** node is added to the **Model Builder**. Nodes with a 'D' in the upper left corner indicate it is a default node.



2 Go to the **Incident Pressure Field** settings window. Under **Boundary Selection** from the **Selection** list, choose **Inlet**.

3 Under **Incident Pressure Field** in the p_0 field, enter p0.



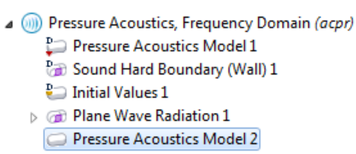
You have now specified the plane wave radiation condition to be active on both the inlet and outlet boundaries, but with an incident wave on the inlet. The remaining boundaries by default use the sound hard condition.

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

PRESSURE ACOUSTICS, FREQUENCY DOMAIN

Pressure Acoustics Model 2

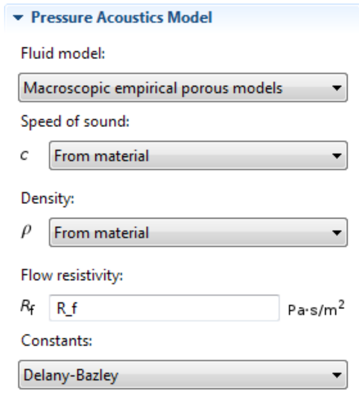
1 In the **Model Builder**, right-click **Pressure Acoustics, Frequency Domain** and from the domain level, choose **Pressure Acoustics Model**.



2 Go to the **Pressure Acoustics Model** settings window. Select Domain 2 only.

3 Under **Pressure Acoustics Model**:

- From the **Fluid model** list, choose **Macroscopic empirical porous models**.
- In the R_f field, enter `R_f`.



MESH 1

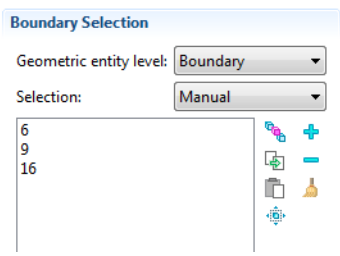
Because the geometry is long and slender and it has constant cross sections an extruded mesh is used. This reduces the number of mesh elements while still retaining the desired mesh resolution of the acoustic field.

Free Triangular 1

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

2 To more easily locate and select a boundary, click the **Wireframe Rendering** button on the **Graphics** window toolbar.

3 Select Boundaries 6, 9, and 16 only.



Size

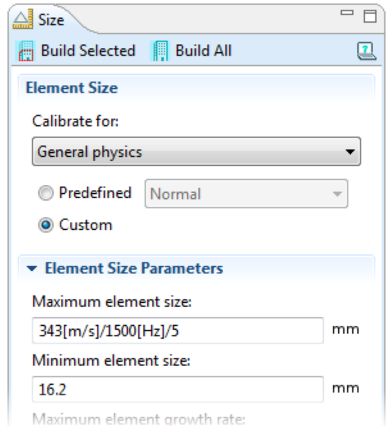
1 In the **Model Builder** under **Model 1>Mesh 1**, click **Size**.

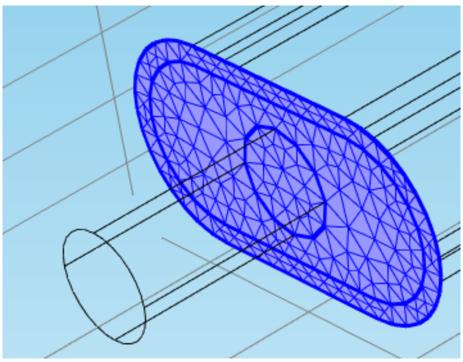
2 Go to the **Size** settings window. Under **Element Size** click the **Custom** button.

3 Under **Element Size Parameters** in the **Maximum element size** field, enter `343[m/s]/1500[Hz]/5`.

   The global maximum element size is set equal to the minimal wavelength divided by 5, that is, $\lambda/5 = c_0/f_{\max}/5$, where c_0 is the speed of sound.

4 Click the **Build All** button.





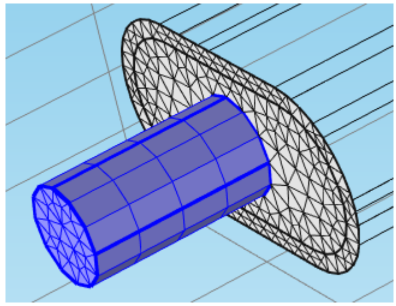
Swept 1

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

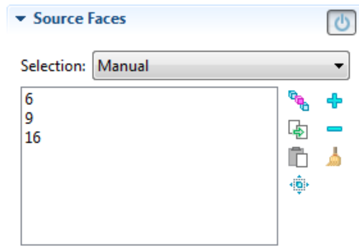
2 Go to the **Swept** settings window. Under **Domain Selection** from the **Geometric entity level** list, choose **Domain**.

3 Select Domain 1 only. Click the **Build Selected** button.

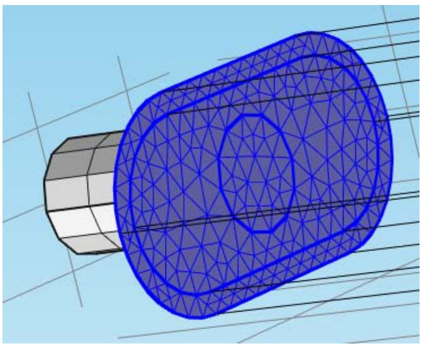
Swept 2

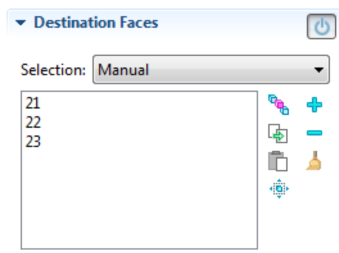


1. In the **Model Builder**, right-click **Mesh 1** and choose **Swept**.
2. Go to the **Swept** settings window. Under **Domain Selection** from the **Geometric entity level** list, choose **Domain**.
3. Select Domains 2 and 3 only.



4. In the upper-right corner of the **Source Faces** section, click the **Activate Selection** button.
5. Select Boundaries 6, 9, and 16 only.

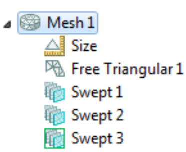




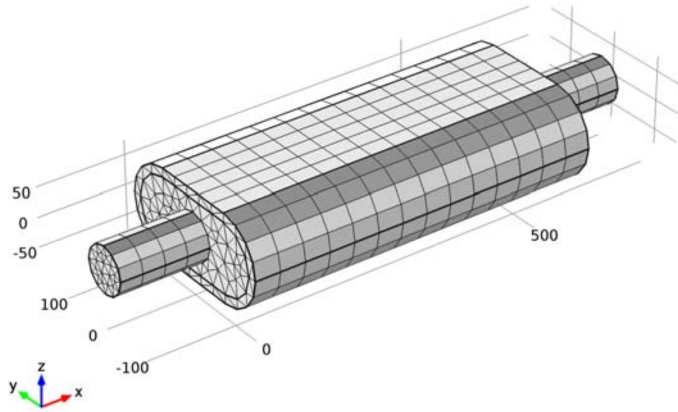
6. In the upper-right corner of the **Destination Faces** section, click the **Activate Selection** button.
7. Select Boundaries 21, 22, and 23 only.
8. Click the **Build Selected** button.

Swept 3

1. In the **Model Builder**, right-click **Mesh 1** and choose **Swept**.
2. Under **Mesh 1**, right-click **Swept 3** and choose **Build Selected**. The node sequence in the **Model Builder** should match this figure:



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



STUDY 1

Step 1: Frequency Domain

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

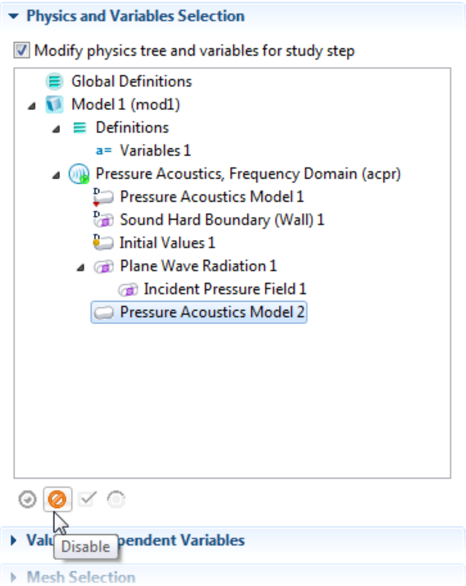
2 Go to the **Frequency Domain** settings window. Under **Study Settings** in the **Frequencies** field, enter `range(50,25,1500)`.

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

4 In the tree, under **Model 1>Pressure Acoustics, Frequency Domain**, click **Pressure Acoustics Model 2**.

5 Click the **Disable** button under the table.

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

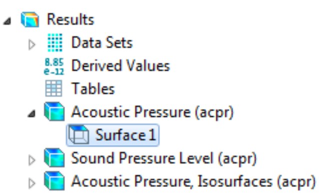


RESULTS

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

Acoustic Pressure (acpr)

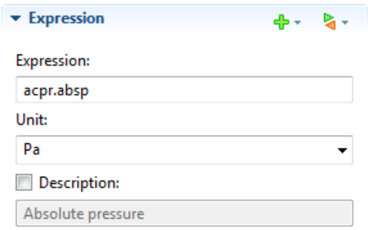
1 In the **Model Builder** under **Results**, expand the **Acoustic Pressure (acpr)** node then click **Surface 1**.

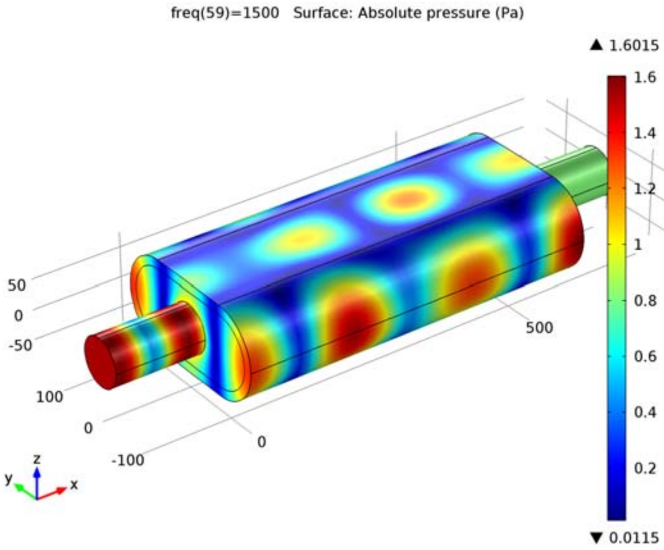


2 Go to the **Surface** settings window. In the upper-right corner of the **Expression** section, click **Replace Expression** .

3 From the menu, choose **Pressure Acoustics, Frequency Domain>Absolute pressure (acpr.absp)** (or enter `acpr.absp` in the **Expression** field).

4 Click the **Plot** button .



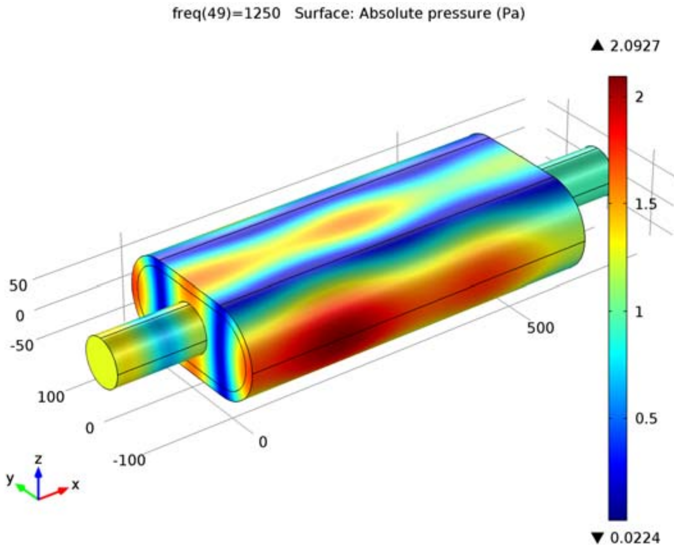


The pattern is very different at different frequencies. See for example what happens at 1250 Hz.

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

2 Go to the **3D Plot Group** settings window. Under **Data** from the **Parameter value (freq)** list, choose **1250**.

3 Click the **Plot** button.

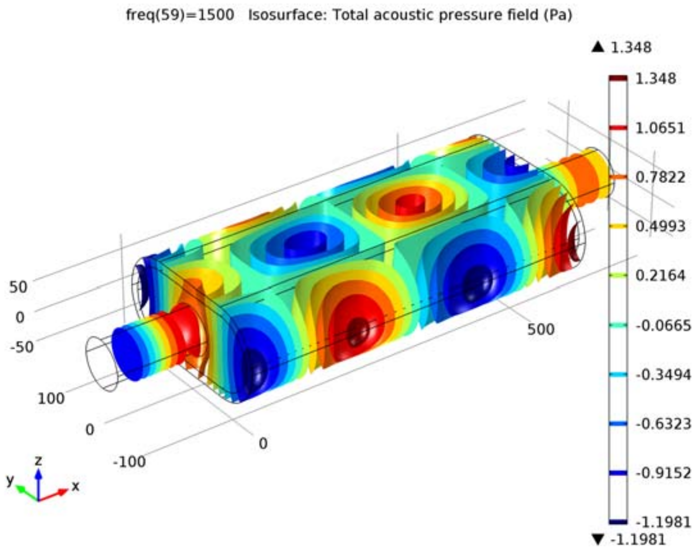


At 1250 Hz, the absolute value of the pressure does not vary much with the x-coordinate. The reason is that this is just higher than the cutoff frequency for the first symmetric propagating mode, which is excited by the incoming wave. For a separate analysis of the propagating modes in the chamber, see the description for the "Eigenmodes in Muffler" model starting on page 46.

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

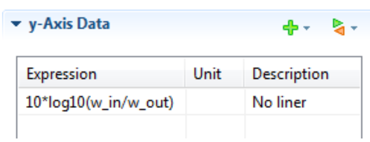
Acoustic Pressure, Isosurfaces (acpr)

In the **Model Builder**, under **Results>Acoustic Pressure, Isosurfaces (acpr)**, click the **Isosurface** node to display the plot in the next figure.



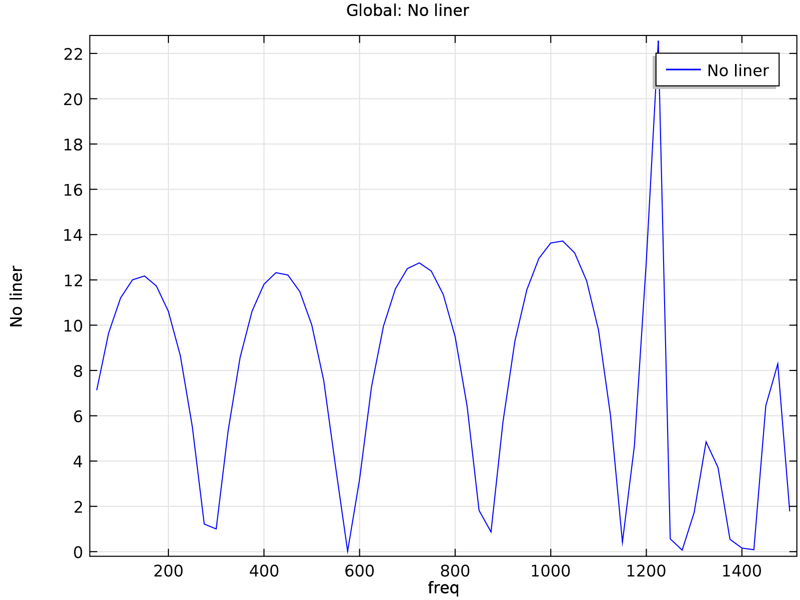
1D Plot Group 4

1. Right-click **Results** and choose **1D Plot Group**.
2. In the **Model Builder** under **Results**, right-click **1D Plot Group 4** and choose **Global**.
3. Go to the **Global** settings window. Under **y-Axis Data** in the table, enter the following settings:
   - In the **Expression** column, enter `10*log10(w_in/w_out)`
   - In the **Description** column, enter `No liner`.



Expression	Unit	Description
10*log10(w_in/w_out)		No liner

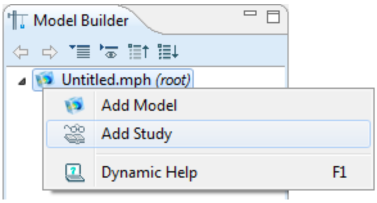
4 Click the **Plot** button. Your plot should be a reproduction of the blue curve in Figure 5 on page 18.



MODEL WIZARD

In this, the second version of the model, the muffler is lined with a layer of absorptive glass wool. Continue working with the model developed so far and make the following changes. The first task is to add a second study to keep the existing results intact.

1 In the **Model Builder**, right-click the root node and choose **Add Study**.

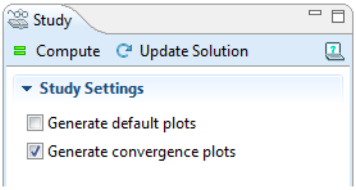


2 Go to the **Model Wizard**. Under **Studies>Preset Studies** click **Frequency Domain**.

3 Click **Finish**. A **Study 2** node is added to the **Model Builder**.

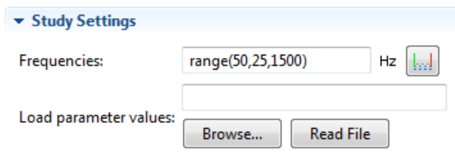
STUDY 2

1. In the **Model Builder**, click **Study 2**.
2. In the **Study** settings window, under the **Study Settings** section, click to clear the **Generate default plots** check box.



Step 1: Frequency Domain

1. In the **Model Builder** under **Study 2**, click **Step 1: Frequency Domain**.
2. In the **Frequency Domain** settings window, locate the **Study Settings** section.
3. In the **Frequencies** field, enter `range(50,25,1500)`.
4. In the **Model Builder**, right-click **Study 1** and choose **Compute**.

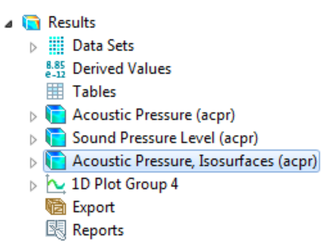


RESULTS

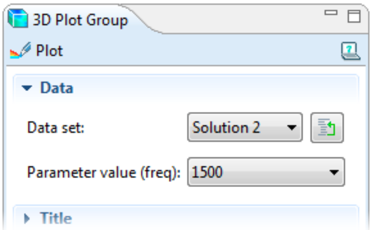
You chose not to have new default plots generated. Once the solution process is finished you can use the existing plot groups and just switch the data set to see how the damping material affects the solution.

Acoustic Pressure, Isosurfaces (acpr)

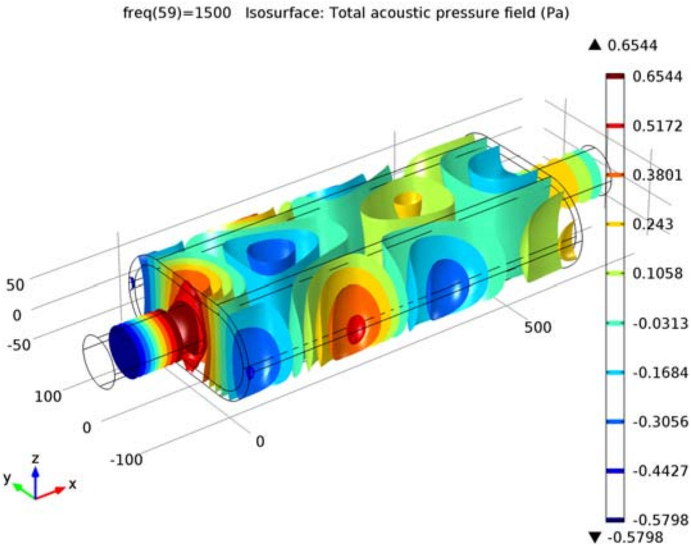
1. In the **Model Builder** under **Results**, click the **Acoustic Pressure, Isosurfaces (acpr)** node.



2 In the **3D Plot Group** settings window under **Data**, from the **Data set** list choose **Solution 2**.



3 Click the **Plot** button.



At 1500 Hz, the pressure in the chamber is much lower than before. Continue to study how the transmission has changed.

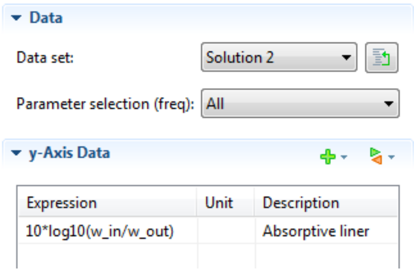
1D Plot Group 4

1 In the **Model Builder** under **Results**, right-click **1D Plot Group 4** and choose **Global**.

2 Go to the **Global** settings window. Under **Data** from the **Data set** list, choose **Solution 2**.

3 Under **y-Axis Data** enter these settings in the table:

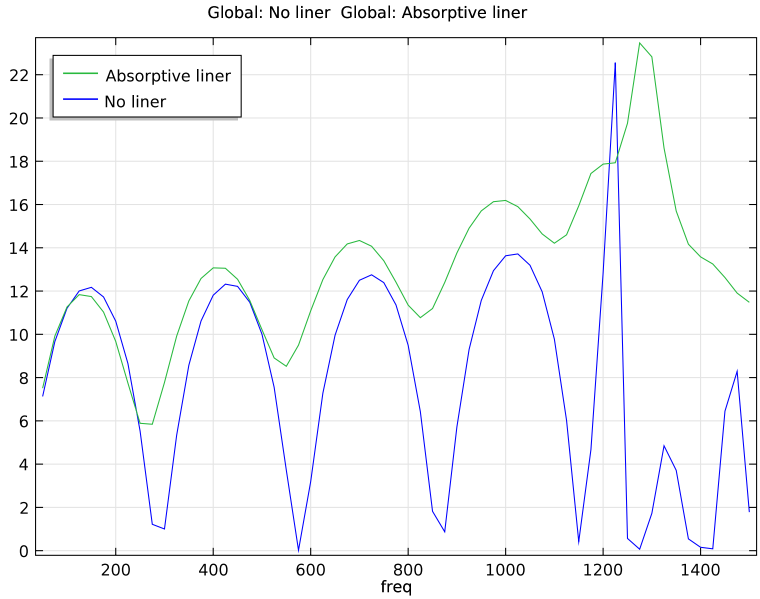
- In the **Expression** column, enter `10*log10(w_in/w_out)`
- In the **Description** column, enter `Absorptive liner`.



4 Click the **Plot** button.

5 In the **Model Builder**, click **1D Plot Group 4**. Go to the **1D Plot Group** settings window. Click to expand the **Legend** section.

6 From the **Position** list, choose **Upper left**.



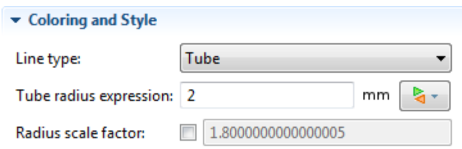
Now, create a plot that represents the intensity flux through the muffler system. Use streamlines that follow the intensity vector (flux of energy through the muffler). You can change between solutions and frequencies to study and visualize the muffler's sound-absorbing properties.

3D PLOT GROUP 5

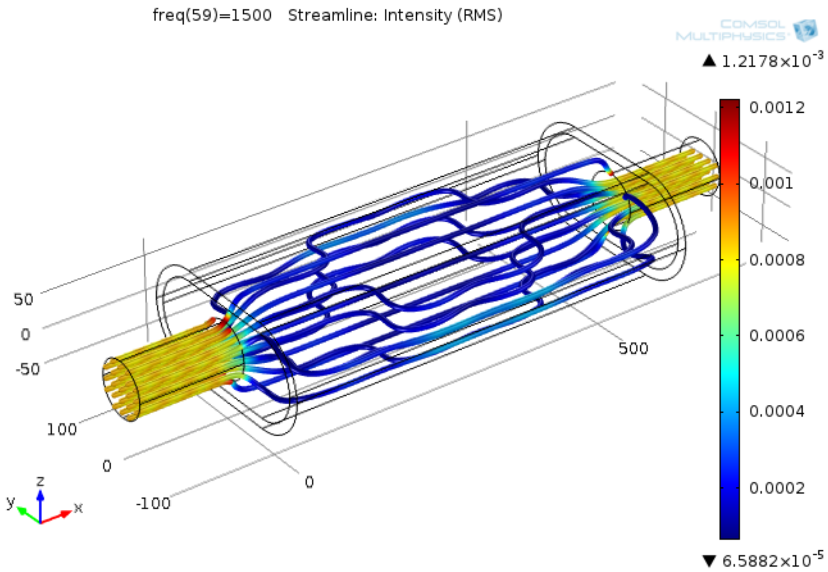
1 In the **Model Builder**, right-click **Results** and choose **3D Plot Group**.
2 Click **3D Plot Group 5** and press F2. In the **Rename 3D Plot Group** dialog box enter `Intensity` as the **New name**.
3 Click **OK**.

Intensity

1 Under **Results**, right-click **Intensity** and choose **Streamline**.
2 In the **Streamline** settings window, click the **Replace Expression** button. From the menu, choose **Pressure Acoustics, Frequency Domain>Intensity (RMS) (acpr.Ix,acpr.Iy,acpr.Iz)**.
3 Under **Selection**, select Boundary 1 only.
4 Under **Coloring and Style**:
   - From the **Line type** list, choose **Tube**.
   - In the **Tube radius expression** field, enter 2.



5 Under **Intensity**, right-click **Streamline 1** and choose **Color Expression**.
6 In the **Color Expression** settings window, click the **Replace Expression** button. From the menu, choose **Pressure Acoustics, Frequency Domain>Intensity magnitude (RMS) (acpr.I_rms)**.
7 Click the **Plot** button.



As a final step, pick one of the plots to use as a model thumbnail.

1 In the **Model Builder** under **Results** click **Acoustic Pressure, Isosurfaces**.
2 From the **File** menu, choose **Save Model Thumbnail**.

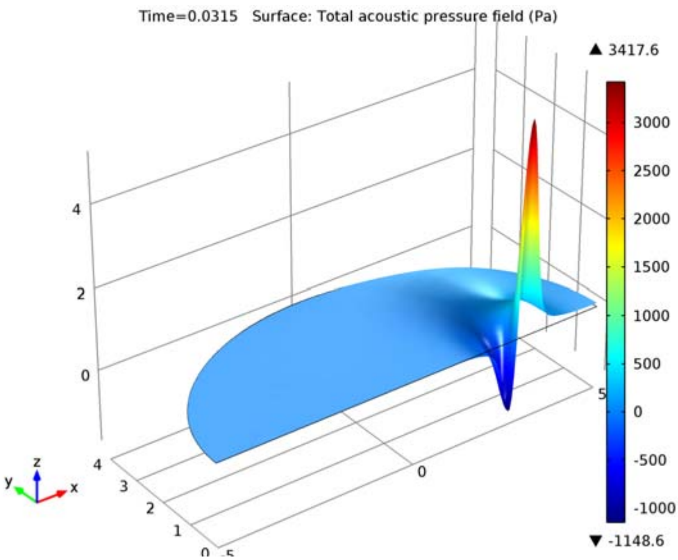
To view the thumbnail image, click the **Root** node and look under the **Model Thumbnail** section. Make adjustments to the image in the **Graphics** window using the toolbar buttons until the image is one that is suitable to your purposes.

Additional Model Library Examples

The Acoustics Module Model Library has other tutorial models available as well as advanced industrial and verification models. Short explanations and examples are given below for a cross-section of these models. Go to "Opening the Model Library" to learn how to access these model files from COMSOL Multiphysics.

Gaussian Explosion

This model introduces important concepts to remember 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.

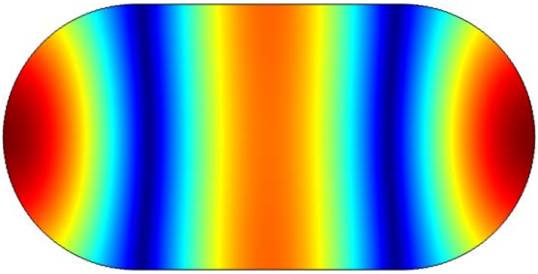


Representation of the pulse at time t=0.0315s.

Eigenmodes in Muffler

In this model, the propagating modes in the chamber of an automotive muffler are computed. The geometry is a cross-section of the chamber as described in the Absorptive Muffler example in this guide.

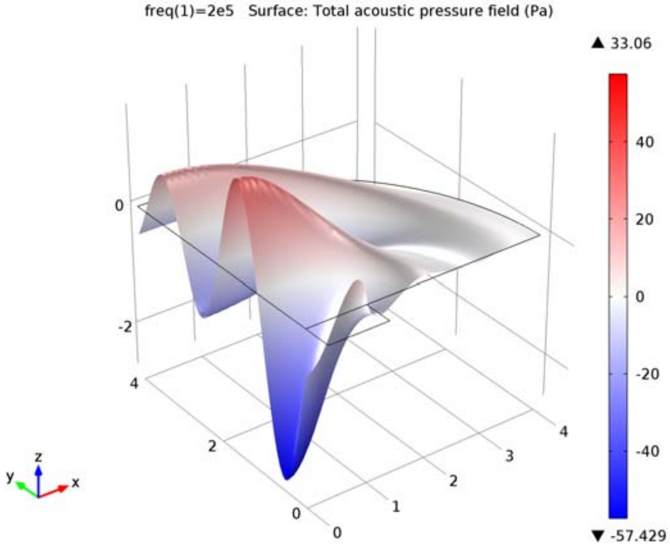
The purpose of the model is to study the shape of the propagating modes and to find the cut-off frequencies. As discussed in the Absorptive Muffler tutorial, some of the modes significantly affect the damping of the muffler at frequencies above the cut-off. In the Eigenmodes in Muffler model, modes with cut-off frequencies up to 1500 Hz are studied.



First fully symmetric propagation mode of the muffler chamber (with no absorbing liner). The plot shows the absolute value of the pressure.

Piezoacoustic Transducer

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.



Surface and height plot of the pressure distribution created bu the piezoactuator at f = 100 kHz.

This concludes this introduction.