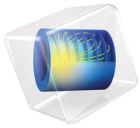

Acoustics Module

User's Guide

5.3

COMSOL

Acoustics Module User's Guide

Version: COMSOL 5.3

Contact Information

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

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

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

Part number: CM020201

Contents

Chapter 1: Introduction

Chapter 2: Pressure Acoustics Interfaces

Chapter 3: Acoustic-Structure Interaction Interfaces

Chapter 4: Aeroacoustics Interfaces

Chapter 5: Thermoviscous Acoustics Interfaces

Chapter 6: Ultrasound Interfaces

Chapter 7: Geometrical Acoustics Interfaces

Chapter 8: Multiphysics Couplings

Chapter 9: Structural Mechanics with the Acoustics Module

Chapter 10: Glossary

1

Introduction

The Acoustics Module is an optional package that extends the COMSOL Multiphysics® environment with customized interfaces and functionality optimized for the analysis of acoustics and vibration problems.

This module solves problems in the general areas of acoustics, acoustic-structure interaction, aeroacoustics, thermoviscous acoustics, linear ultrasound, pressure and elastic waves in porous materials, vibrations, and geometrical acoustics. The physics interfaces included are fully multiphysics enabled, making it possible to couple them to any other physics interface in COMSOL Multiphysics. Explicit demonstrations of these capabilities are supplied with the product in a library (the Acoustics Module Application Library) of ready-to-run models and applications that make it quicker and easier to get introduced to discipline-specific problems. One example being a model of a loudspeaker involving both electromechanical and acoustic-structural couplings.

This chapter is an introduction to the capabilities of the Acoustics Module and gives a short introduction to the fundamentals of acoustics. The different study types available are also presented. A summary of the physics interfaces and where you can find documentation and model examples is also included. The last section is a brief overview with links to each chapter in this guide.

In this chapter:

Acoustics Module Capabilities

In this section:

What Can the Acoustics Module Do?

The Acoustics Module is a collection of physics interfaces for COMSOL Multiphysics adapted to a broad category of acoustics simulations in fluids and solids. This module is useful even if you are not familiar with computational techniques. It can serve equally well as an excellent tool for educational purposes.

The module supports time-harmonic (frequency domain), modal, and transient studies for fluid pressure as well as static, transient, eigenfrequency, and frequency-response analyses for structures. The available physics interfaces include the following functionality:

- *Pressure acoustics*: model the propagation of sound waves (pressure waves) in the frequency domain and in the time domain solving the Helmholtz equation or the scalar wave equation, respectively.
- *Acoustic-structure interaction*: combine pressure waves in the fluid with elastic waves in the solid. The physics interface provides predefined multiphysics couplings at the fluid-solid interface.
- *Boundary mode acoustics*: find propagating and evanescent modes in ducts and waveguides.
- *Thermoviscous acoustics*: model the detailed propagation of sound in geometries with small length scales. This is acoustics including thermal and viscous losses explicitly. Also known as visco-thermal acoustics, thermo acoustics, or linearized compressible Navier-Stokes.
- *Aeroacoustics*: model the influence a background mean flow has on the propagation of sound waves in the flow, so-called, flow borne noise/sound. Interfaces exist to solve the linearized potential flow, the linearized Euler equations, and the linearized Navier-Stokes equations in both time and frequency domain.

- *Compressible potential flow*: determine the flow of a compressible, irrotational, and inviscid fluid.
- *Solid mechanics and elastic waves*: solve structural mechanics problems and the propagation of elastic waves in solids.
- *Piezoelectricity*: model the behavior of piezoelectric materials in a multiphysics environment solving for the electric field and the coupling to the solid structure.
- *Poroelastic waves*: in porous materials model the coupled propagation of elastic waves in the solid porous matrix and the pressure waves in the saturation fluid. Biot's equations are solved here. Includes options to include both thermal and viscous losses.
- *Ultrasound*: in ultrasound problems transient propagation is important and it is also important to be able to solve models with many wavelengths. These interfaces are based in the discontinuous Galerkin or DG-FEM formulation.
- *Acoustic diffusion equation*: solve a diffusion equation for the acoustic energy density distribution for systems of coupled room in room acoustic applications.
- *Ray acoustics*: compute trajectories and intensity of acoustic rays in room acoustic as well as underwater acoustic applications.
- *Pipe acoustics*: use this physics interface to model the propagation of sound waves in pipe systems including the elastic properties of the pipe. The equations are formulated in 1D for fast computation and can include a stationary background flow. This functionality requires the addition of the Pipe Flow Module.

All the physics interfaces include a large number of boundary conditions. For the pressure acoustics applications you can choose to analyze the scattered wave in addition to the total wave. Impedance conditions can be used to mimic a specific acoustic behavior at a boundary, for example, the acoustic properties of the human ear or a mechanical system approximated by a simple RCL circuit. Perfectly matched layers (PMLs) provide accurate simulations of open pipes and other models with unbounded domains. The modeling domain includes support for several types of damping and losses that occur in porous materials (poroacoustics) or that are due to viscous and thermal losses (narrow region acoustics). For results evaluation of pressure acoustics models, you can compute the far field (phase and magnitude) and plot it in predefined far-field plots.

What are the Application Areas?

The Acoustics Module can be used in all areas of engineering and physics to model the propagation of sound waves in fluids. The module also includes several multiphysics interfaces because it is common for many application areas involving sound to also have interaction between fluid and solid structures, have electric fields in piezo materials, have heat generation, or require modeling of electro-acoustic transducers. Typical application areas for the Acoustics Module include:

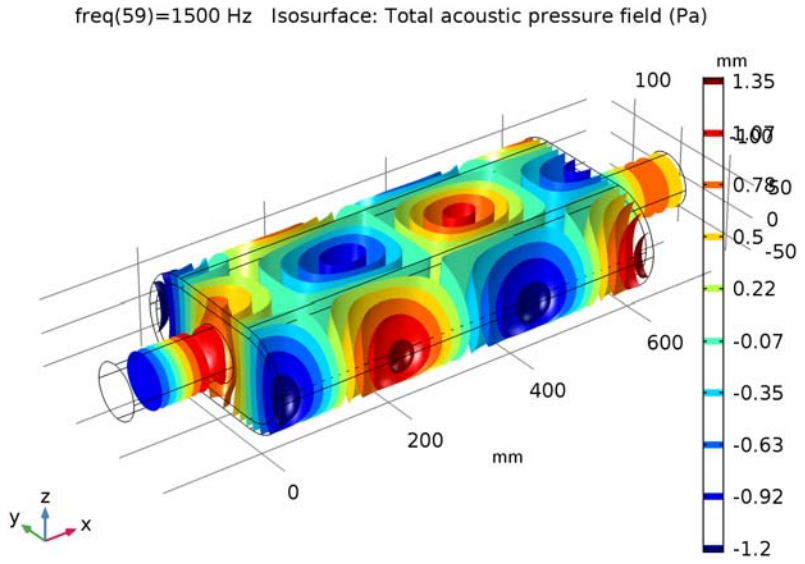

Figure 1-1: An application example is the modeling of mufflers. Here a pressure isosurface plot from the Absorptive Muffler model from the COMSOL Multiphysics Applications Libraries.

- Automotive applications such as mufflers, particulate filters, and car interiors.
- Sound scattering, absorption, and sound emission problems.
- Civil engineering applications such as characterization of sound insulation and sound scatterers. Vibration control and sound transmission problems. Pipe acoustics for HVAC type of systems.
- Modeling of loudspeakers, microphones, and other transducers. Transducers are devices for transformation of one form of energy to another (electrical, mechanical,

or acoustical). This type of problem is common in acoustics and is a true multiphysics problem involving electric, structural, and acoustic interfaces.

- Mobile applications such as feedback analysis, optimized transducer placement, and directivity assessment.
- Aeroacoustics for jet engine noise, muffler systems with non-isothermal flow, and flow meters.
- Ultrasound piezo transducers.
- Musical instruments.
- Bioacoustic applications with ultrasound and more.
- Underwater acoustics and sonar applications.

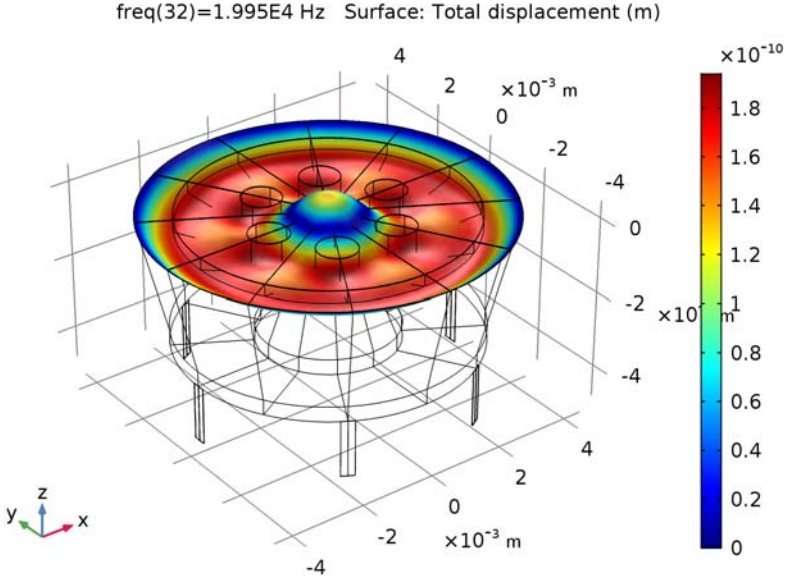

Figure 1-2: Modeling a transducer is a true multiphysics application, comprising thermoviscous acoustics, electrostatics, and a membrane. Here the displacement of the microphone diaphragm from the Brüel & Kjær 4134 Condenser Microphone model from the COMSOL Multiphysics Applications Libraries.

- Pressure waves in geophysics.

- Room acoustics using the ray tracing method or an acoustic diffusion equation approach.
- Advanced multiphysics applications such as photoacoustics, optoacoustics, thermoacoustic cooling, acoustofluidics, acoustic streaming and radiation, and combustion instabilities.

Using the full multiphysics couplings within the COMSOL Multiphysics environment, you can couple the acoustic waves to, for example, an electromagnetic analysis or a structural analysis for acoustic-structure interaction. The module smoothly integrates with all of the COMSOL Multiphysics functionality.

Which Problems Can You Solve?

The Acoustics Module interfaces handle acoustics in fluids (both quiescent and moving background flows) and solids. The physics interfaces for acoustics in fluids support transient, eigenfrequency, frequency domain, mode analysis, and boundary mode analysis in pressure acoustics and linearized potential flow. Thermoacoustic problems, that involve thermal and viscous losses, have support for eigenfrequency and frequency domain analysis. The study of elastic and poroelastic waves in solids also has support for eigenfrequency and frequency domain analysis. The physics interfaces for solids support static, transient, eigenfrequency, and frequency response analysis. Further, by using the predefined couplings between fluid and solid interfaces, you can solve problems involving acoustic-structure interaction including the coupling to piezoelectric materials.

All categories are available as 2D, 2D axisymmetric, and 3D models, with the following differences.

- The Acoustic-Shell Interaction interfaces are only supported in 3D and also require the addition of the Structural Mechanics Module.
- The Pipe Acoustics interfaces, which require the Pipe Flow Module, exist in edges in 2D and 3D.
- In 2D the module has in-plane physics interfaces for problems with a planar symmetry as well as axisymmetric physics interfaces for problems with a cylindrical symmetry.

- Use the fluid acoustics interfaces with 1D and 1D axisymmetric geometries.

	When using the axisymmetric models, the horizontal axis represents the *r* direction and the vertical axis the *z* direction. The geometry is in the right half plane; that is, the geometry must be created and is valid only for positive *r*.

Fundamental of Acoustics

This section includes a brief introduction to acoustics and provides a short introduction to the mathematical formulation of the governing equations. It also introduces some important concepts like damping and the use of artificial boundaries.

In this section:

Acoustics Explained

Acoustics is the physics of sound. Sound is the sensation, as detected by the ear, of very small rapid changes in the air pressure above and below a static value. This static value is the atmospheric pressure (about 100,000 pascals), which varies slowly. Associated with a sound pressure wave is a flow of energy — the intensity. Physically, sound in air is a longitudinal wave where the wave motion is in the direction of the movement of energy. The wave crests are the pressure maxima, while the troughs represent the pressure minima.

Sound results when the air is disturbed by some 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 propagating at the speed of sound.

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. The elastic sound waves in the solid are the counterpart to the pressure waves or compressible waves propagating in the fluid.

Standard acoustic problems involve solving for the small acoustic pressure variations p on top of the stationary background pressure p_0. Mathematically this represents a linearization (small parameter expansion) around the stationary quiescent values.

The governing equations for a compressible lossless (no thermal conduction and no viscosity) fluid flow problem are the momentum equation (Euler's equation) and the continuity equation. These are given by:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p$$

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

where ρ is the total density, p is the total pressure, and $\mathbf{u}$ is the velocity field. In classical pressure acoustics all thermodynamic processes are assumed reversible and adiabatic, known as an isentropic process. The small parameter expansion is performed on a stationary fluid of density ρ_0 (SI unit: kg/m^3) and at pressure p_0 (SI unit: Pa) such that:

$$\begin{aligned} p &= p_0 + p' \\ \rho &= \rho_0 + \rho' \\ \mathbf{u} &= \mathbf{0} + \mathbf{u}' \end{aligned} \qquad \text{with} \qquad \begin{aligned} p' &\ll p_0 \\ \rho' &\ll \rho_0 \end{aligned}$$

where the primed variables represent the small acoustic variations (sometimes denoted with a subscript 1 instead). Inserting these into the governing equations and only retaining terms linear in the primed variables yields

$$\frac{\partial \mathbf{u}'}{\partial t} = -\frac{1}{\rho_0}\nabla p'$$

$$\frac{\partial \rho'}{\partial t} + \rho_0(\nabla \cdot \mathbf{u}') = 0$$

One of the dependent variables, the density, is removed by expressing it in terms of the pressure using a Taylor expansion (linearization)

$$\rho' = \left.\frac{\partial \rho_0}{\partial p}\right|_s p' = \frac{1}{c_s^2} p'$$

where c_s is recognized as the (isentropic) speed of sound (SI unit: m/s) at constant entropy s. It should be noted that this equation is valid for constant valued (non-space

dependent) density and background pressure. The subscripts s and 0 are dropped in the following. From the above expression it also follows that another requirement for the perturbation to be valid is that

$$p' \ll \rho_0 c_s^2$$

Finally, rearranging the equations (divergence of momentum equation inserted into the continuity equation) and dropping the primes yields the wave equation for sound waves in a lossless medium

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p - \mathbf{q}_\mathrm{d})\right) = Q_\mathrm{m} \tag{1-1}$$

The speed of sound is related to the compressibility of the fluid where the waves are propagating. The combination ρc^2 is called the *bulk modulus*, commonly denoted K (SI unit: N $/\mathrm{m}^2$). The equation is further extended with two optional source terms:

- The *dipole domain source* $\mathbf{q}_\mathrm{d}$ (SI unit: $\mathrm{N/m}^3$).
- The *monopole domain source* Q_m (SI unit: $1/\mathrm{s}^2$).

A special case is a time-harmonic wave, for which the pressure varies with time as

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

where $\omega = 2\pi f$ (SI unit: rad/s) is the angular frequency and f (SI unit: Hz) is denoting the frequency. Assuming the same harmonic time-dependence for the source terms, the wave equation for acoustic waves reduces to an inhomogeneous Helmholtz equation:

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

With the two source terms removed, this equation can also be treated as an eigenvalue PDE to solve for eigenmodes and eigenfrequencies.

Typical boundary conditions for the wave equation and the Helmholtz equation are:

- Sound-hard boundaries (walls)
- Sound-soft boundaries
- Impedance boundary conditions
- Radiation boundary conditions

In lossy media, an additional term of first order in the time derivative needs to be introduced to model attenuation of the sound waves:

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} - d_a\frac{\partial p}{\partial t} + \nabla\cdot\left(-\frac{1}{\rho}(\nabla p - \mathbf{q}_d)\right) = Q_m$$

where d_a is the damping coefficient. Note also that even when the sound waves propagate in a lossless medium, attenuation frequently occurs by interaction with the surroundings at the boundaries of the system.

A detailed derivation of the governing equations is given in Theory Background for the Pressure Acoustics Branch. For the propagation of compressional (acoustic) waves in a viscous and thermally conducting fluid the theory is presented in Theory Background for the Thermoviscous Acoustics Branch and for acoustics in moving media (aeroacoustics) in Theory Background for the Aeroacoustics Branch.

Damping

Fluids with a dynamic viscosity in the same range as air or water — by far the most common media in acoustics simulations — exhibit practically no internal damping over the number of wavelengths that can be resolved on current computers. Instead, damping takes place through interaction with solids, either because of friction between the fluid and a porous material filling the domain, or because acoustic energy is transferred to a surrounding solid where it is absorbed. In systems with small length scales, significant losses can occur in the viscous and thermal acoustic boundary layer at walls.

POROUS ABSORBING MATERIALS

For frequency-domain modeling, the most convenient and compact description of a damping material (where material here refers to the homogenization of a fluid and a porous solid) is given by its complex wave number k and complex impedance Z, both functions of frequency. Knowing these properties, define a complex speed of sound as $c_c = \omega/k$ and a complex density as $\rho_c = kZ/\omega$. Defining ρ_c and c_c results in a so-called equivalent-fluid model or fluid model.

It is possible to directly measure the complex wave number and impedance in an impedance tube in order to produce curves of the real and imaginary parts (the resistance and reactance, respectively) as functions of frequency. These data can be used directly as input to COMSOL Multiphysics interpolation functions to define k and Z.

Sometimes acoustic properties cannot be obtained directly for a material you want to try in a model. In that case you must resort to knowledge about basic material properties independent of frequency. Several empirical or semi-empirical models exist in COMSOL Multiphysics and can estimate the complex wave number and impedance as function of material parameters. These models are defined in the Poroacoustics domain feature of the Pressure Acoustics interfaces — for example, the *Johnson-Champoux-Allard* model and the *Delany-Bazley-Miki* models; the latter uses frequency and flow resistivity as input.

The Acoustics Module includes a series of fluid models that are described in Pressure Acoustics and Theory for the Equivalent Fluid Models. In addition, The Poroelastic Waves Interface can be used for detailed modeling of the propagation of coupled pressure and elastic waves in porous materials.

BOUNDARY LAYER ABSORPTION (THERMOVISCOUS ACOUSTICS)

In systems of small dimensions (or at low frequencies) the size of the acoustic boundary layer (the viscous and thermal acoustic penetration depth) that exists at all walls can become comparable to the physical dimensions of the modeled system. In air the boundary layer thickness is 0.22 mm at 100 Hz. This is typically the case inside miniature transducers, condenser microphones, in MEMS systems, in tubing for hearing aids, or in narrow gaps of vibrating structures.

For such systems it is often necessary to use a more detailed model for the propagation of the acoustics waves. This model is implemented in the Thermoviscous Acoustics interface. In simple cases for sound propagating in long ducts of constant cross sections, the losses occurring at the boundaries can be smeared out on the fluid using one of the fluid models of the Narrow Region Acoustics domain feature.

More details on the detailed acoustic model for viscous and thermal losses are described in Thermoviscous Acoustics Interfaces. See the boundary layer absorption fluid models in Narrow Region Acoustics.

DAMPING AT BOUNDARIES

The losses associated with the acoustic field often stem from the interaction with boundaries. For example, when interacting with a rubber material. In this case it may be necessary to include the acoustic-structure interaction using the appropriate multiphysics coupling. Another way of including the losses is to use an impedance

boundary condition. The Acoustics Module provide a series of impedance models to model, for example, the human ear, human skin, or a simple mechanical lumped RCL system.

Artificial Boundaries

In most cases, the acoustic wave pattern that is to be simulated is not contained in a closed cavity. That is, there are boundaries in the model that do not represent a physical wall or limit of any kind. Instead, the boundary condition has to represent the interaction between the wave pattern inside the model and everything outside. Conditions of this kind are generically referred to as *artificial boundary conditions* (ABCs).

Such conditions should ideally contain complete information about the outside world, but this is not practical. After all, the artificial boundary was introduced to avoid spending degrees of freedom (DOFs) on modeling whatever is outside. The solution lies in trying to approximate the behavior of waves outside the domain using only information from the boundary itself. This is difficult in general for obvious reasons.

One particular case that occurs frequently in acoustics concerns boundaries that can be assumed to let wave energy propagate out from the domain without reflections. This leads to the introduction of a particular group of artificial boundary conditions known as *non-reflecting boundary conditions* (NRBCs), of which two kinds are available in this module: *matched boundary conditions* and *radiation boundary conditions*.

Another way to model an open non-reflecting boundary is to add a so-called perfectly matched layer (PML) domain. This domain dampens all outgoing waves with no or minimal reflections. See Perfectly Matched Layers (PMLs) for more information.

Acoustics Module Study Types

The Acoustics Module is primarily designed for frequency-domain simulations, including related eigenvalue and mode analysis problems. Transient analysis is possible but less efficient from the computational point of view. The Thermoviscous Acoustics interfaces only support the frequency-domain analysis type. The Compressible Potential Flow interface is tailored to model a stationary background flow to be used in a subsequent time-harmonic aeroacoustics simulation. The linearized aeroacoustic interfaces support both transient and frequency-domain analysis. In the Solid Mechanics interface, the static analysis type is also included and can be used to model the stationary state of prestressed systems subject to time-harmonic vibrations.

The analysis types require different solvers and equations. The following study types, briefly discussed in this section, can help you find good candidates for the application:

- Stationary Study
- Frequency Domain Study
- Eigenfrequency Study
- Mode Analysis Study
- Time Dependent Study
- Frequency Domain Modal and Time-Dependent Modal Studies
- Modal Reduced Order Model
- Additional Analysis Capabilities

Studies and Solvers and Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis in the *COMSOL Multiphysics Reference Manual*.

Prestressed Acoustic-Structure Interaction type of problems are also sported using a combination of a stationary solver and a frequency domain perturbation solver.

Stationary Study

A stationary analysis solves for stationary displacements or a steady-state condition. All loads and constraints are constant. For a stationary analysis, use a Stationary study type ( ). For all pure acoustic and vibration problems this type of analysis yields the

zero solution as, by definition, these represent and describe propagating varying fields — either time dependent or time harmonic in the frequency domain.

Stationary in the *COMSOL Multiphysics Reference Manual*

Frequency Domain Study

Wave propagation is modeled by equations from linearized fluid dynamics (pressure waves) and structural dynamics (elastic waves). The full equations are time dependent, but noting that a harmonic excitation of the field u has a time dependence of the form

$$u = ue^{i\omega t}$$

gives rise to an equally harmonic response with the same frequency; the time can be eliminated completely from the equations. Instead the angular frequency $\omega = 2\pi f$, enters as a parameter where f is the frequency.

This procedure is often referred to as working in the *frequency domain* or *Fourier domain* as opposed to the *time domain*. From the mathematical point of view, the time-harmonic equation is a Fourier transform of the original time-dependent equations and its solution as function of ω is the Fourier transform of a full transient solution. It is therefore possible to synthesize a time-dependent solution from a frequency-domain simulation by applying an inverse Fourier transform.

COMSOL Multiphysics and the Acoustics Module are based on the finite element method; a frequency domain simulation suits this method very well. Therefore, select the Frequency Domain study type ( ) over a Time Dependent study whenever possible. Certain important software features, notably PMLs and damping due to porous media or boundary layer absorption, are only present when using the frequency domain physics interfaces.

The result of a frequency domain analysis is a complex time-dependent field u, which can be interpreted as an amplitude $u_{\text{amp}} = \text{abs}(u)$ and a phase angle $u_{\text{phase}} = \arg(u)$. The actual displacement at any point in time is the real part of the solution:

$$u = u_{\text{amp}} \cos(2\pi f \cdot t + u_{\text{phase}})$$

Visualize the amplitudes and phases as well as the solution at a specific angle (time). When using the Solution data sets, the *solution at angle (phase)* parameter makes this task easy. When plotting the solution, COMSOL Multiphysics multiplies it by $e^{i\varphi}$,

where φ is the angle in radians that corresponds to the angle (specified in degrees) in the Solution at angle field. The plot shows the real part of the evaluated expression:

$$u = u_{\text{amp}} \cos(\varphi + u_{\text{phase}})$$

The angle φ is available as the variable `phase` (in radians) and is allowed in plot expressions. Both the frequency `freq` and angular frequency `omega` are available variables.

	In a frequency domain study almost everything is treated as harmonic — prescribed pressures and displacements, velocities, and accelerations — not only the forces and dependent fields. Notable exceptions are certain quantities, such as the sound pressure level, which by definition are time averages.

	Frequency Domain and Solution (data sets) in the *COMSOL Multiphysics Reference Manual*

Eigenfrequency Study

If all sources are removed from a frequency-domain equation, its solution becomes zero for all but a discrete set of angular frequencies ω, where the solution has a well-defined shape but undefined magnitude. These solutions are known as eigenmodes and the corresponding frequencies as eigenfrequencies.

The eigenmodes and eigenfrequencies have many interesting mathematical properties, but also direct physical significance because they identify the *resonance frequency* (or frequencies) of the structure. When approaching a resonance frequency in a harmonically-driven problem, a weaker and weaker source is needed to maintain a given response level. At the actual eigenfrequency, the time-harmonic problem breaks down and lacks solution for a nonzero excitation.

Select the **Eigenfrequency** study type () when you are interested in the resonance frequencies of the acoustic domain or the structure, whether you want to exploit them, as in a musical instrument, or avoid them, as in a reactive muffler or inside a hifi speaker system. To an engineer, the distribution of eigenfrequencies and the shape of eigenmodes can also give a good first impression about the behavior of a system.

An eigenfrequency analysis solves for the eigenfrequencies and the shape of the eigenmodes. When performing an eigenfrequency analysis, specify whether to look at the mathematically more fundamental eigenvalue λ (available as the variable `lambda`) or the eigenfrequency f which is more commonly used in an acoustics context:

$$f = \frac{-\lambda}{2\pi i}$$

Eigenfrequency in the *COMSOL Multiphysics Reference Manual*

Mode Analysis Study

The Mode Analysis study () is available with The Pressure Acoustics, Frequency Domain Interface, The Linearized Potential Flow, Frequency Domain Interface, and The Thermoviscous Acoustics, Frequency Domain Interface in plane 2D and axially symmetric 1D acoustics interfaces.

The Pressure Acoustics, Boundary Mode Interface and The Linearized Potential Flow, Boundary Mode Interface are special interfaces for more advanced Mode Analysis studies on boundaries in 3D and 2D axisymmetry. Acoustic waves can propagate over large distances in ducts and pipes, with a generic name referred to as *waveguides*. After some distance of propagation in a waveguide of uniform cross section, such guided waves can be described as a sum of just a few discrete *propagating modes*, each with its own shape and phase speed. The equation governing these modes can be obtained as a spatial Fourier transform of the time-harmonic equation in the waveguide axial z direction or by inserting the assumption that the mode is harmonic in space,

$$u = ue^{-ik_z z}$$

and eliminating all out-of-plane z dependence.

The axial wave number k_z is a parameter in the 2D acoustics interfaces.

Similar to the full time-harmonic equation, the transformed equation can be solved at a given frequency with a nonzero excitation for most axial wave numbers k_z. But at certain discrete values the equation breaks down. These values are the propagation

constants or wave numbers of the propagating or evanescent waveguide modes. The eigenvalue solver can solve for these propagation constants together with the corresponding mode shapes.

The propagating wave number is a function of the frequency. The relation between the two is commonly referred to as a *dispersion curve*.

The most common use for the Mode Analysis is to define sources for a subsequent time-harmonic simulation. If there is a component with one or more waveguide connections, its behavior can be described by simulating its response to the discrete set of propagating modes on the waveguide port cross sections. In thermoviscous acoustics a Mode Analysis study also provides information about the absorption coefficient for the propagating modes, which is the imaginary part of the wave number.

Mode Analysis in the *COMSOL Multiphysics Reference Manual*

Jet Pipe: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/jet_pipe**

Time Dependent Study

The complete equations behind the theory of acoustic wave propagation are time dependent, as discussed in the Frequency Domain Study section. Solving time-domain equations is more complicated from a numerical point of view and should therefore be avoided when possible. Short-term transient processes like step and impulse responses can benefit from modeling in the time domain, if not for efficiency so for convenience.

Some central modeling techniques, such as the use of PMLs, are not available for the **Time Dependent** study type. Be careful when defining your sources to avoid, as far as possible, exciting waves at frequencies that the mesh cannot resolve.

Time Dependent in the *COMSOL Multiphysics Reference Manual*

Frequency Domain Modal and Time-Dependent Modal Studies

The **Frequency Domain Modal** study type ( ) is used to do modal analysis in the frequency domain and the **Time-Dependent Modal** study ( ) is used to do time-dependent modal analysis.

Frequency-Domain Modal and Time-Dependent Modal in the *COMSOL Multiphysics Reference Manual*

Modal Reduced Order Model

The Modal Reduced Order Model study type ( ) is used to obtain the data necessary to construct reduced-order models from a COMSOL Multiphysics simulation. This study step is added after an existing Eigenvalue study step by right-clicking the **Study 1** node and selecting **Study Steps>Time Dependent>Modal Reduced Order Model**. After solving the model, right-click the **Derived Values** node (under **Results**) and select **System Matrix**. In the output section select the **Matrix** to display and the **Format**. Using the Matrix settings it is possible to access the stiffness, damping, and mass matrices of the system, for example.

In the *COMSOL Multiphysics Reference Manual:*

- Modal Reduced Order Model and Introduction to Solvers and Studies
- System Matrix and Results Analysis and Plots

Additional Analysis Capabilities

In a multiphysics interface you might want to use different analysis types for the different dependent variables. This can be done by adding an Empty Study ( ), and then adding different study steps to this study. Also perform parametric analyses by using the **Parametric Sweep** study node ( ). Typical parameters to vary include geometric properties, the frequency, and the out-of-plane or axial wave number.

Parametric Sweep in the *COMSOL Multiphysics Reference Manual*

Acoustics Module Physics Interface Guide

The Acoustics Module extends the functionality of the physics interfaces of the COMSOL Multiphysics base package. The details of the physics interfaces and study types for the Acoustics Module are listed in the table below.

In the *COMSOL Multiphysics Reference Manual:*

- Studies and Solvers
- The Physics Interfaces
- For a list of all the core physics interfaces included with a COMSOL Multiphysics license, see Physics Interface Guide.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Acoustics				
Pressure Acoustics				
Pressure Acoustics, Frequency Domain[1]		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; modal reduced order model; mode analysis (2D and 1D axisymmetric models only)
Pressure Acoustics, Boundary Mode		acbm	3D, 2D axisymmetric	mode analysis

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Acoustic-Structure Interaction				
Acoustic-Solid Interaction, Frequency Domain[4]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Acoustic-Solid Interaction, Transient[4]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal; time dependent; time-dependent modal; modal reduced order model
Acoustic-Shell Interaction, Frequency Domain[2,4]		—	3D	eigenfrequency; frequency domain; frequency-domain modal
Acoustic-Shell Interaction, Transient[2,4]		—	3D	eigenfrequency; frequency domain; frequency-domain modal; time dependent; time-dependent modal; modal reduced order model
Acoustic-Piezoelectric Interaction, Frequency Domain[4]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Acoustic-Piezoelectric Interaction, Transient[4]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal; time dependent; time-dependent modal; modal reduced order model
Solid Mechanics (Elastic Waves)		solid	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Poroelastic Waves		pelw	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Acoustic-Solid-Poroelastic Waves Interaction[4]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Acoustic-Poroelastic Waves Interaction[4]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal
Pipe Acoustics, Frequency Domain[3]		pafd	3D, 2D	eigenfrequency; frequency domain
Pipe Acoustics, Transient[3]		patd	3D, 2D	time dependent
Aeroacoustics				
Linearized Euler, Frequency Domain		lef	3D, 2D axisymmetric, 2D, and 1D	frequency domain; eigenfrequency
Linearized Euler, Transient		let	3D, 2D axisymmetric, 2D, and 1D	time dependent
Linearized Potential Flow, Frequency Domain		ae	all dimensions	frequency domain; mode analysis (2D and 1D axisymmetric models only)
Linearized Potential Flow, Transient		aetd	all dimensions	frequency domain; time dependent mode analysis (2D and 1D axisymmetric models only)
Linearized Potential Flow, Boundary Mode		aebm	3D, 2D axisymmetric	mode analysis
Compressible Potential Flow		cpf	all dimensions	stationary; time dependent
Linearized Navier-Stokes, Frequency Domain		lnsf	3D, 2D axisymmetric, 2D, and 1D	frequency domain; eigenfrequency
Linearized Navier-Stokes, Transient		lnst	3D, 2D axisymmetric, 2D, and 1D	time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Thermoviscous Acoustics				
Thermoviscous Acoustics, Frequency Domain		ta	all dimensions	eigenfrequency; frequency domain; frequency domain modal; mode analysis (2D and 1D axisymmetric models only)
Thermoviscous Acoustics, Transient		tatd	all dimensions	time dependent
Thermoviscous Acoustics, Boundary Mode		tabm	3D, 2D axisymmetric	mode analysis
Acoustic-Thermoviscous Acoustic Interaction, Frequency Domain[4]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain modal
Thermoviscous Acoustic-Solid Interaction, Frequency Domain[4]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain modal
Thermoviscous Acoustic-Shell Interaction, Frequency Domain[2,4]		—	3D	eigenfrequency; frequency domain; frequency domain modal
Ultrasound				
Convected Wave Equation, Time Explicit		cwe	3D, 2D, 2D axisymmetric	time dependent
Geometrical Acoustics				
Ray Acoustics		rac	3D, 2D, 2D axisymmetric	ray tracing; time dependent
Acoustic Diffusion Equation		ade	3D	eigenvalue; stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Structural Mechanics				
Solid Mechanics[1]		solid	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; prestressed analysis, eigenfrequency; mode analysis; time dependent; time-dependent modal; frequency domain; frequency-domain modal; prestressed analysis, frequency domain; modal reduced order model
Piezoelectric Devices[4]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; time-dependent modal; frequency domain; frequency-domain modal; small-signal analysis, frequency domain; prestressed analysis, eigenfrequency; prestressed analysis, frequency domain; modal reduced order model
Magnetostriction		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; prestressed analysis, eigenfrequency; prestressed analysis, frequency domain

[1] This physics interface is included with the core COMSOL package but has added functionality for this module.

[2] Requires both the Structural Mechanics Module and the Acoustics Module.

[3]Requires both the Pipe Flow Module and the Acoustics Module.

[4] This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.

Common Physics Interface and Feature Settings and Nodes

There are several common settings and sections available for the physics interfaces and feature nodes. Some of these sections also have similar settings or are implemented in the same way no matter the physics interface or feature being used. There are also some physics feature nodes that display in COMSOL Multiphysics.

In each module's documentation, only unique or extra information is included; standard information and procedures are centralized in the *COMSOL Multiphysics Reference Manual*.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Where Do I Access the Documentation and Application Libraries?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries are all accessed through the COMSOL Desktop.

If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules (as long as you have a license), application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes the core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to many of the features on the GUI. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context) help*.

Win

To open the **Help** window:

- In the **Model Builder**, **Application Builder**, or **Physics Builder** click a node or window and then press F1.
- On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1.
- From the **File** menu, click **Help** (?).
- In the upper-right corner of the COMSOL Desktop, click the **Help** (?) button.

Mac

Linux

To open the **Help** window:

- In the **Model Builder** or **Physics Builder** click a node or window and then press F1.
- On the main toolbar, click the **Help** (?) button.
- From the main menu, select **Help>Help**.

Opening the Documentation Window

Win

To open the **Documentation** window:

- Press Ctrl+F1.
- From the **File** menu select **Help>Documentation** ( ).

Mac

Linux

To open the **Documentation** window:

- Press Ctrl+F1.
- On the main toolbar, click the **Documentation** ( ) button.
- From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

Each application includes documentation with the theoretical background and step-by-step instructions to create a model application. The applications are available in COMSOL as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual applications as a template for your own modeling and applications. In most models, SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the application and its properties, including options to open it or a PDF document.

The Application Libraries Window in the *COMSOL Multiphysics Reference Manual.*

Opening the Application Libraries Window

To open the **Application Libraries** window ( ):

Win

- From the **Home** toolbar, **Windows** menu, click ( ) **Applications Libraries**.
- From the **File** menu select **Application Libraries**.

To include the latest versions of model examples, from the **File>Help** menu, select ( ) **Update COMSOL Application Library**.

Mac

Linux

Select **Application Libraries** from the main **File>** or **Windows>** menus.

To include the latest versions of model examples, from the **Help** menu select ( ) **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and a case number are sent to you by email.

COMSOL ONLINE RESOURCES

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

Overview of the User's Guide

The *Acoustics Module User's Guide* gets you started with modeling using COMSOL Multiphysics. The information in this guide is specific to this module. Instructions on how to use COMSOL in general are included with the *COMSOL Multiphysics Reference Manual*.

As detailed in the section Where Do I Access the Documentation and Application Libraries? this information can also be searched from the COMSOL Multiphysics software **Help** menu.

Good practices as well as some tips and tricks are located in dedicated modeling sections under each branch. They are:

- Modeling with the Pressure Acoustics Branch
- Modeling with the Acoustic-Structure Interaction Branch
- Modeling with the Aeroacoustics Branch
- Modeling with the Thermoviscous Acoustics Branch
- Modeling with the Convected Wave Equation Interface
- Modeling with the Ray Acoustics Interface
- Modeling with the Acoustic Diffusion Equation Interface
- Modeling with Multiphysics Couplings.

Dedicated theory sections also exist under each branch. Here the physics and mathematics of the governing equations and selected boundary conditions are discussed.

TABLE OF CONTENTS, GLOSSARY, AND INDEX

To help you navigate through this guide, see the Contents, Glossary, and Index.

THE PRESSURE ACOUSTICS BRANCH

Pressure Acoustics Interfaces chapter describes the following interfaces.

The Pressure Acoustics, Frequency Domain Interface is the core physics interface which models the sound waves in the frequency domain and The Pressure Acoustics, Transient Interface is the core physics interface which models the sound waves in the time domain. The Pressure Acoustics, Boundary Mode Interface solves for modes that propagate through a cross section of your geometry.

The physics interfaces solve for the acoustic variations in pressure. The Helmholtz equation is solved in the frequency domain and the scalar wave equation is solved in the time domain. Domain conditions exist for modeling losses in a homogenized way in porous materials as well as in narrow regions. Sources such as background fields are also available. Boundary conditions include sources, non-reflecting radiation conditions, impedance conditions, periodic conditions, far-field calculation conditions, as well as interior boundaries such as walls or perforated plates.

THE ACOUSTIC-STRUCTURE INTERACTION BRANCH

Acoustic-Structure Interaction Interfaces chapter describes these following interfaces.

The Acoustic-Solid Interaction, Frequency Domain Interface is a combination of pressure acoustics and solid mechanics with predefined couplings and The Acoustic-Solid Interaction, Transient Interface is a combination of transient pressure acoustics and solid mechanics with predefined couplings.

The Acoustic-Piezoelectric Interaction, Frequency Domain Interface is a combination of pressure acoustics and piezoelectric effects with a predefined coupling for the boundary between the acoustic domain and the piezoelectric device. The Acoustic-Piezoelectric Interaction, Transient Interface combines Pressure Acoustics, Transient, Solid Mechanics, Electrostatics, and the Piezoelectric Devices interface features.

The Solid Mechanics (Elastic Waves) Interface combines pressure acoustics and solid mechanics to connect the fluid pressure with the structural deformation in solids. It also features The Poroelastic Waves Interface, which can be seen as linear elastic waves coupled to pressure waves in porous elastic materials damped by a pore fluid.

The Acoustic-Solid-Poroelastic Waves Interaction Interface combines Pressure Acoustics, Frequency Domain and Elastic Waves together with the Acoustic-Structure Boundary and Acoustic-Porous Boundary multiphysics coupling feature. It also features The Acoustic-Poroelastic Waves Interaction Interface, another multiphysics interface that combines Pressure Acoustics, Frequency Domain and Poroelastic Waves together with the Acoustic-Structure Boundary and Acoustic-Porous Boundary multiphysics coupling feature.

The Acoustic-Shell Interaction, Frequency Domain Interface requires a Structural Mechanics Module license. It uses the features from the Pressure Acoustics, Frequency Domain and the Shell interfaces to connect the acoustics pressure waves in a fluid domain with the structural deformation in a shell. The physics interface is available for 3D geometry only.

The Acoustic-Shell Interaction, Transient Interface, which also requires a Structural Mechanics Module license, uses the features from the Pressure Acoustics, Transient and the Shell interfaces to connect the transient pressure acoustics in a fluid domain with the structural deformation of shell boundary. The physics interface is available for 3D geometry only.

The Pipe Acoustics Interfaces, which require both the Pipe Flow Module and the Acoustics Module, have the equations and boundary conditions for modeling the propagation of sound waves in flexible pipe systems. The equations are formulated in a general way to include the possibility of a stationary background flow. There are two interfaces, one for transient analysis and one for frequency domain studies.

THE AEROACOUSTICS BRANCH

The following interfaces are under Aeroacoustics Interfaces.

The Linearized Potential Flow, Frequency Domain Interface models acoustic waves in potential flow in the frequency domain and The Linearized Potential Flow, Transient Interface models acoustic waves in potential flow in the time domain. The Linearized Potential Flow, Boundary Mode Interface solves for modes that propagate through a cross section of your geometry.

The Compressible Potential Flow Interface models irrotational flow used as input for the background flow in the linearized potential flow interfaces.

The Linearized Euler, Frequency Domain Interface and The Linearized Euler, Transient Interface models the acoustic variations in density, velocity, and pressure in the presence of a stationary background mean-flow that is well approximated by an ideal gas flow. This physics interface is used for aeroacoustic simulations that can be described by the linearized Euler equations.

The Linearized Navier-Stokes, Frequency Domain Interface and The Linearized Navier-Stokes, Transient Interface models the acoustic variations in pressure, velocity, and temperature in the presence of any stationary isothermal or non-isothermal background mean-flow. This physics interface is used for aeroacoustic simulations that can be described by the linearized Navier-Stokes equations.

THE THERMOVISCOUS ACOUSTICS BRANCH

The Thermoviscous Acoustics Interfaces chapter describes The Thermoviscous Acoustics, Frequency Domain Interface, which is necessary when modeling acoustics accurately in geometries with small dimensions. Near walls, viscosity and thermal conduction become important because they create a viscous and a thermal boundary layer where losses are significant. The The Thermoviscous Acoustics, Boundary Mode Interface is used to identify propagating and non-propagating modes waveguides and ducts of small dimensions including thermal and viscous losses.

The Acoustic-Thermoviscous Acoustic Interaction, Frequency Domain Interface combines the Thermoviscous Acoustics, Frequency Domain and Pressure Acoustics, Frequency Domain interfaces together with Acoustic-Thermoviscous Acoustic Boundary coupling feature.

The Thermoviscous Acoustic-Solid Interaction, Frequency Domain Interface is also described here. This physics interface combines features from pressure acoustics, thermoviscous acoustics, and solid mechanics with predefined couplings between all three physics interfaces.

The Thermoviscous Acoustic-Shell Interaction, Frequency Domain Interface requires a Structural Mechanics Module license. The physics interface uses the features from the Thermoviscous Acoustics, Frequency Domain and the Shell interfaces to connect wave propagation in pressure acoustic domains and thermoviscous acoustic domains with the structural deformation of shell boundaries.

ULTRASOUND

The Ultrasound Interfaces includes The Convected Wave Equation, Time Explicit Interface, used to model the propagating of linear ultrasound waves in the time domain, including the effects of a stationary background flow.

THE GEOMETRICAL ACOUSTICS BRANCH

Geometrical Acoustics Interfaces includes The Ray Acoustics Interface, used to compute the trajectories, phase, and intensity of acoustic rays, and The Acoustic Diffusion Equation Interface, which solves a diffusion equation for the acoustic energy density. The theory is also discussed for both physics interfaces.

THE MULTIPHYSICS COUPLINGS

The Multiphysics Couplings chapter describes all the multiphysics couplings available with the Acoustics Module. These are the built in conditions that can couple the

physics together, for example, the Acoustic-Structure boundary couples pressure acoustics to any solid boundary.

STRUCTURAL MECHANICS WITH THE ACOUSTICS MODULE

The Structural Mechanics with the Acoustics Module chapter provides information about The Solid Mechanics Interface used for modeling, for example, the structural part of acoustic-structure interaction. This is an extension of the Solid Mechanics interface in COMSOL Multiphysics, and you find it under the **Structural Mechanics** branch.

The Piezoelectric Devices Interface interface is also shortly presented. It combines Solid Mechanics and Electrostatics together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled and the piezoelectric coupling can be formulated using either the strain-charge or stress-charge forms.

The theory for the solid mechanics interface as well as for the piezoelectric devices interface is found in the *Structural Mechanics Module User's Guide*.

2

Pressure Acoustics Interfaces

This chapter describes the Acoustics Module background theory and physics interfaces found under the **Pressure Acoustics** branch ( ).

The Pressure Acoustics, Frequency Domain Interface

The **Pressure Acoustics, Frequency Domain (acpr)** interface ( ), found under the **Pressure Acoustics** branch ( ) when adding a physics interface, is used to compute the pressure variations for the propagation of acoustic waves in fluids at quiescent background conditions. It is suited for all frequency-domain simulations with harmonic variations of the pressure field.

The physics interface can be used for linear acoustics described by a scalar pressure variable. It includes domain conditions to model losses in a homogenized way, so-called equivalent fluid models, for porous materials as well as losses in narrow regions (waveguides or slits). The plane wave attenuation behavior of the acoustic waves may also be entered as a user-defined quantity, or defined to be bulk viscous and/or thermal losses. Domain features also include background acoustic fields, as well as monopole and dipole domain sources.

The physics interface solves the Helmholtz equation in the frequency domain for given frequencies, or as an eigenfrequency or modal analysis study.

An acoustics model can be part of a larger multiphysics model that describes, for example, the interactions between structures and acoustic waves. This physics interface is suitable for modeling acoustics phenomena that do not involve fluid flow (convective effects).

The sound pressure p, which is solved for in pressure acoustics, represents the acoustic variations (or acoustic perturbations) to the ambient pressure. In the absence of flow, the ambient pressure p_A is simply the static absolute pressure.

The governing equations and boundary conditions are formulated using the total pressure p_t with a so-called scattered field formulation. In the presence of a Background Pressure Field defining a background pressure wave p_b (this could, for example, be a plane wave), the total acoustic pressure p_t is the sum of the pressure solved for p (which is then equal to the scattered pressure p_s) and the background

pressure wave: $p_t = p+p_b$. The equations then contain the information about both the scattered field and the background pressure field.

For good modeling strategies, meshing, solvers, postprocessing information, acoustics specific plots, as well as tips and tricks, see the Modeling with the Pressure Acoustics Branch section.

When the geometrical dimensions of the acoustic problems are reduced from 3D to 2D (planar symmetry or axisymmetric) or to 1D axisymmetric, it is possible to specify an out-of-plane wave number k_z and a circumferential mode number m, when applicable. In this case, the wave number used in the equations k_{eq} contains both the ordinary wave number k as well as the out-of-plane wave number and circumferential wave number $k_m = m/r$, when applicable.

The following table lists the names and SI units for the most important physical quantities in the Pressure Acoustics, Frequency Domain interface:

TABLE 2-1: PRESSURE ACOUSTICS, FREQUENCY DOMAIN INTERFACE PHYSICAL QUANTITIES

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Pressure	p	Pascal	Pa
Total pressure	p_t	Pascal	Pa
Background pressure	p_b	Pascal	Pa
Scattered pressure	p_s	Pascal	Pa
Density (quiescent)	ρ or ρ_c	kilogram/meter^3	kg/m^3
Frequency	f	Hertz	Hz
Wave number	k	1/meter	1/m
Dipole domain source	$\mathbf{q}_d$	newton/meter^3	N/m^3
Monopole domain source	Q_m	1/second^2	1/s^2
Speed of sound	c or c_c	meter/second	m/s
Specific acoustic impedance	Z	pascal-second/meter	Pa·s/m
Acoustic impedance	Z_{ac}	pascal-second/meter^3	Pa·s/m^3
Normal acceleration	a_n	meter/second^2	m/s^2
Normal velocity	v_n	meter/second	m/s
Source location	$\mathbf{x}_0$	meter	m
Wave direction	$\mathbf{e}_k$	(dimensionless)	1

In the following descriptions of the functionality in this physics interface, the subscript c in ρ_c and c_c (the density and speed of sound, respectively) denotes that these can be complex-valued quantities in models with damping.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Pressure Acoustics Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and point conditions. You can also right-click **Pressure Acoustics** to select physics features from the context menu.

Physics Nodes — Equation Section in the *COMSOL Multiphysics Reference Manual*

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `acpr`.

EQUATION

Expand the **Equation** section to see the equations solved for with the **Equation form** specified. The default selection is **Equation form** is set to **Study controlled**. The available studies are selected under **Show equations assuming**.

- For **Study controlled**, the scaling and nonreflecting boundary settings are optimized for the numerical performance of the different solvers and study types.
- For **Frequency domain** enter the settings as described in Scaling Factor and Nonreflecting Boundary Condition Approximation.

PRESSURE ACOUSTICS EQUATION SETTINGS

In this section you can add out-of-plane information defining an out-of-plane wave number k_z or a circumferential wave number $k_m = m/r$ through the mode number m. Add if applicable:

- For 1D axisymmetric components, the default **Out-of-plane wave number** k_z (SI unit: rad/m) is 0 rad/m. The default **Circumferential mode number** m (dimensionless) is 0. The pressure has the form:

$$p(r, \varphi, z) = p(r)e^{-i(k_z z + m\varphi)}$$

- For 2D axisymmetric components, the default **Circumferential mode number** m (dimensionless) is 0. The pressure has the form:

$$p(r, \varphi, z) = p(r, z)e^{-im\varphi}$$

- For 2D components, the default **Out-of-plane wave number** k_z (SI unit: rad/m) is 0 rad/m. The pressure has the form:

$$p(x, y, z) = p(x, y)e^{-ik_z z}$$

Scaling Factor and Nonreflecting Boundary Condition Approximation

For all component dimensions, and if required, click to expand the **Equation** section, then select **Frequency domain** as the **Equation form** and enter the settings as described below.

The default **Scaling factor** Δ is $1/\omega^2$ and **Non-reflecting boundary condition approximation** is **Second order**. These values correspond to the equations for a Frequency Domain study when the equations are study controlled.

To get the equations corresponding to an Eigenfrequency study, change the **Scaling factor** Δ to 1 and the **Non-reflecting boundary conditions approximation** to **First order**.

SOUND PRESSURE LEVEL SETTINGS

The zero level on the dB scale varies with the type of fluid. That value is a reference pressure that corresponds to 0 dB. This variable occurs in calculations of the sound pressure level L_p based on the root mean square (rms) pressure p_{rms}, such that

$$L_p = 20\log\left(\frac{p_{rms}}{p_{ref}}\right) \qquad \text{with} \qquad p_{rms} = \sqrt{\frac{1}{2}pp^*}$$

where p_{ref} is the reference pressure and the star (*) represents the complex conjugate. This is an expression valid for the case of harmonically time-varying acoustic pressure p.

Select a **Reference pressure for the sound pressure level** based on the fluid type:

- **Use reference pressure for air** to use a reference pressure of 20 μPa ($20 \cdot 10^{-6}$ Pa).
- **Use reference pressure for water** to use a reference pressure of 1 μPa ($1 \cdot 10^{-6}$ Pa).
- **User-defined reference pressure** to enter a reference pressure $p_{ref,\,SPL}$ (SI unit: Pa). The default value is the same as for air, 20 μPa.

TYPICAL WAVE SPEED

Enter a value or expression for the **Typical wave speed for perfectly matched layers** c_{ref} (SI unit m/s). The default is 343 m/s.

DEPENDENT VARIABLES

This physics interface defines one dependent variable (field), the **Pressure** p. If required, edit the name, which changes both the field name and the dependent variable name. If the new field name coincides with the name of another pressure field in the model, the interfaces share degrees of freedom and dependent variable name. The new field name must not coincide with the name of a field of another type, or with a component name belonging to some other field.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. From the list select the element order and type (Lagrange or serendipity) for the **Pressure**, the default is **Quadratic Lagrange**.

	Choosing between Lagrange and Serendipity Shape Functions has influence on the number of DOFs solved for and on stability for distorted mesh.

	• Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface • Theory Background for the Pressure Acoustics Branch

Eigenmodes of a Room: Application Library path **COMSOL_Multiphysics/Acoustics/eigenmodes_of_room**

This also requires the Particle Tracing Module — *Acoustic Levitator*: Application Library path **Acoustics_Module/Nonlinear_Acoustics/acoustic_levitator**

Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface

The Pressure Acoustics, Frequency Domain Interface has these domain, boundary, edge, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users). Continuity in the total pressure is the default condition on interior boundaries.

The Pressure Acoustics, Transient Interface also shares these nodes, with some additional features described in Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface.

The Pressure Acoustics, Boundary Mode Interface also shares these nodes, with one additional feature described in Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Boundary Mode Interface. For the Pressure Acoustics, Boundary Mode interface, apply the feature to boundaries instead of domains for 3D components.

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Axial Symmetry
- Background Pressure Field
- Circular Source (for 2D axisymmetric components)
- Continuity
- Cylindrical Wave Radiation
- Dipole Point Source
- Dipole Domain Source
- Far-Field Calculation
- Heat Source
- Impedance
- Incident Pressure Field
- Interior Normal Acceleration
- Interior Normal Displacement
- Interior Normal Velocity
- Interior Impedance/Pair Impedance
- Interior Perforated Plate/Pair Perforated Plate
- Interior Sound Hard Boundary (Wall)
- Initial Values
- Line Source
- Line Source on Axis
- Matched Boundary
- Monopole Point Source
- Monopole Domain Source
- Narrow Region Acoustics
- Normal Acceleration
- Normal Displacement
- Normal Velocity
- Quadrupole Point Source
- Periodic Condition
- Plane Wave Radiation
- Point Sources for 2D Components
- Poroacoustics
- Pressure Acoustics
- Pressure
- Sound Hard Boundary (Wall)
- Sound Soft Boundary
- Spherical Wave Radiation
- Symmetry

For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node to the component that is valid on the axial symmetry boundaries only.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Sound Hard Boundary (Wall)

The **Sound Hard Boundary (Wall)** adds a boundary condition for a sound hard boundary or wall, which is a boundary at which the normal component of the acceleration (and thus the velocity) is zero:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) = 0$$

For zero dipole domain source ($\mathbf{q}_d = \mathbf{0}$) and constant fluid density ρ_c, this means that the normal derivative of the pressure is zero at the boundary

$$\frac{\partial p_t}{\partial \mathbf{n}} = 0$$

Sound-hard boundaries are available for all study types. Note that mathematically this condition is identical to the Symmetry condition.

Initial Values

The **Initial Values** node adds and initial value for the pressure. In the time domain it also adds and initial value for the pressure time derivative. The initial values can serve as an initial guess for a nonlinear solver or describe the initial state that will then evolve in time. If more than one initial value is needed, from the **Physics** toolbar click to add more **Initial Values** nodes.

INITIAL VALUES

Enter a value or expression for the initial values for the **Pressure** p (SI unit: Pa) and, in the time domain, also for the **Pressure, first time derivative**, $\partial p/\partial t$ (SI unit: Pa/s).

Monopole Domain Source

Use the **Monopole Domain Source** node to add a the domain source term Q_m to the governing equation. A monopole domain source added to a domain has a uniform strength in all directions. In advanced models this source term can, for example, be used to represent a domain heat source causing pressure variations or a nonlinear contribution to the equations in the time domain. Add this node from the **More** submenu.

MONOPOLE SOURCE

Enter a **Monopole domain source** Q_m (SI unit: $1/s^2$).

In a transient model the Monopole Domain Source can be used to add nonlinearities to the governing equation. See the model *Nonlinear Acoustics—Modeling of the 1D Westervelt Equation* for such an example: Application Library path **Acoustics_Module/Nonlinear_Acoustics/nonlinear_acoustics_westervelt_1d**.

Dipole Domain Source

Use the **Dipole Domain Source** node to add the domain source term $\mathbf{q}_d$ to the governing equation. This source represents a domain volumetric force. This source is typically stronger in two opposite directions. In advanced models this term can, for example, be used to represent a uniform constant background flow convecting the sound field. Add this node from the **More** submenu.

DIPOLE SOURCE

Enter coordinates for the **Dipole domain source** $\mathbf{q}_d$ (SI unit: N/m^3). These are the individual components of the dipole source vector.

Heat Source

Use the **Heat Source** feature to add a domain heat source that generates sound. If a fluid is heated and cooled rapidly thermal expansion and contraction will generate acoustic waves. This could be a source representing a pulsating laser beam or a flame source in a combustion simulation. The feature adds a monopole domain source to the right hand side defined as

$$Q_m = \frac{\alpha_p}{\rho C_p} i\omega Q_{heat} \quad \text{(frequency domain)}$$

$$Q_m = \frac{\alpha_p}{\rho C_p} \frac{\partial Q_{heat}}{\partial t} \quad \text{(time domain)}$$

$$\alpha_p = \frac{1}{c}\sqrt{\frac{C_p(\gamma-1)}{T}}$$

where the (isobaric) coefficient of thermal expansion α_p is defined in terms of the speed of sound c, heat capacity an constant pressure C_p, ambient temperature T (this value is taken from the model inputs and can be space dependent), and ratio of specific heats γ.

DOMAIN HEAT SOURCE

Specify the **Heat source** strength Q_{heat} (SI unit: W/m^3).

FLUID PROPERTIES

Enter the additional fluid properties necessary to define the heat source. Enter the **Heat capacity at constant pressure** C_{p} (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) and the **Ratio of specific heats** γ (dimensionless). The default for both is **From material**, select **User defined** to enter a value or expression directly.

Normal Acceleration

The **Normal Acceleration** adds an inward normal acceleration a_{n}:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_{\text{c}}}(\nabla p_{\text{t}}-\mathbf{q}_{\text{d}})\right) = a_{\text{n}}$$

Alternatively, specify the acceleration $\mathbf{a}_0$ of the boundary. The part in the normal direction is used to define the boundary condition:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_{\text{c}}}(\nabla p_{\text{t}}-\mathbf{q}_{\text{d}})\right) = -\mathbf{n}\cdot\mathbf{a}_0$$

This feature represents an external source term. It can also be used to manually couple acoustics with a structural analysis for modeling acoustic-structure interaction.

NORMAL ACCELERATION

Select a **Type**: **Inward Acceleration** (the default) or **Acceleration**.

- For **Inward Acceleration** enter the value of the **Inward acceleration** a_{n} (SI unit: m/s^2). Use a positive value for inward acceleration or a negative value for outward acceleration.
- For **Acceleration** enter values for the components of the **Acceleration** $\mathbf{a}_0$ (SI unit: m/s^2).

Normal Velocity

The **Normal Velocity** adds an inward normal velocity v_{n}, which is assumed to be harmonically oscillating:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_{\text{c}}}(\nabla p_{\text{t}}-\mathbf{q}_{\text{d}})\right) = i\omega v_{\text{n}}$$

Alternatively, specify the acceleration $\mathbf{v}_0$ of the boundary. The part in the normal direction is used to define the boundary condition:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right) = -i\omega\mathbf{n}\cdot\mathbf{v}_0$$

This feature represents an external source term. It can also be used to manually couple acoustics with a structural analysis for modeling acoustic-structure interaction.

NORMAL VELOCITY

Select a **Type**: **Inward Velocity** (the default) or **Velocity**.

- For **Inward Velocity** enter the value of the **Inward velocity** v_n (SI unit: m/s). Use a positive value for inward velocity or a negative value for outward velocity.
- For **Velocity** enter values for the components of the **Velocity** $\mathbf{v}_0$ (SI unit: m/s).

Normal Displacement

The **Normal Displacement** adds an inward normal displacement d_n, which is assumed to be harmonically oscillating:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right) = (i\omega)^2 d_n$$

Alternatively, specify the acceleration $\mathbf{d}_0$ of the boundary. The part in the normal direction is used to define the boundary condition:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right) = -(i\omega)^2\mathbf{n}\cdot\mathbf{v}_0$$

This feature represents an external source term. It can also be used to manually couple acoustics with a structural analysis for modeling acoustic-structure interaction.

NORMAL DISPLACEMENT

Select a **Type**: **Inward Displacement** (the default) or **Displacement**.

- For **Inward Displacement** enter the value of the **Inward displacement** d_n (SI unit: m). Use a positive value for inward displacement or a negative value for outward displacement.

For **Displacement** enter values for the components of the **Displacement** $\mathbf{d}_0$ (SI unit: m).

Sound Soft Boundary

The **Sound Soft Boundary** adds a boundary condition for a sound soft boundary, where the acoustic pressure vanishes: $p_t = 0$. It is an appropriate approximation for a liquid-gas interface and in some cases for external waveguide ports.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Pressure

The **Pressure** node creates a boundary condition that acts as a pressure source at the boundary, which means that a constant acoustic pressure p_0 is specified and maintained at the boundary: $p_t = p_0$. In the frequency domain, p_0 is the amplitude of a harmonic pressure source. The node is also available from the **Pairs** submenu as an option at interfaces between parts in an assembly.

PRESSURE

Enter the value of the **Pressure** p_0 (SI unit: Pa).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Impedance

The **Impedance** node adds an impedance boundary condition with the option to select between several built in impedance models. The impedance condition is a generalization of the sound-hard and sound-soft boundary conditions:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) = -\frac{i\omega p_t}{Z_i}$$

In the Pressure Acoustics, Transient interface using a Time Dependent study, the impedance boundary condition is the following:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d)\right) = \frac{1}{Z_i}\frac{\partial p_t}{\partial t}$$

Here Z_i is the specific acoustic input impedance of the external domain and it has the SI unit Pa·s/m — a pressure divided by a velocity. From a physical point of view, the

acoustic input impedance is the ratio between the local pressure and local normal particle velocity. The specific acoustic impedance Z_i is related to the acoustic impedance Z_{ac} (ratio of pressure and flow rate) and the mechanical impedance Z_{mech} (ratio of force and velocity) via the area A of the boundary, according to

$$Z_{mech} = AZ_i = A^2 Z_{ac}$$

The Impedance boundary condition is a good approximation of a locally reacting surface — a surface for which the normal velocity at any point depends only on the pressure at that exact point.

Most impedance models only exist in the frequency domain. The only exception is the User defined impedance which can be applied also in the time domain.

In the two opposite limits $Z_i \to \infty$ and $Z_i \to 0$, this boundary condition is identical to the Sound Hard Boundary (Wall) condition and the Sound Soft Boundary condition, respectively.

IMPEDANCE

A number of different types of impedance boundary conditions are included to address standard situations in many typical applications of pressure acoustics.

- Simple RCL models to define lumped-parameter circuits describing, for example, the mechanical response of a microphone diaphragm.
- A set of physiological models enable easy modeling of the surface impedance of human skin and the acoustic loads of the human ear and eardrum.
- The waveguide end impedances define simplified models of the acoustic losses at the end of pipes and ducts.
- A porous layer backed by a sound-hard wall model can be defined.
- Simple wave-type specific characteristic impedance model exist to address open boundaries in a simplified way.

The ear impedance, skin impedance, and RCL models provide a tool for engineers to add realistic acoustic loads when, for example, developing and simulating headphones, hearing aids, head sets, and other mobile devices.

Choose an **Impedance model** — **User defined**, **RCL**, **Physiological**, **Waveguide end impedance**, **Porous layer**, or **Characteristic specific impedance**.

User defined

Allows the user to enter any expression and is the only impedance model which applies to time-dependent models. It is advantageous to enter complicated user-defined models as a variable under the **Definitions** node or use an interpolation function for measured data.

Enter the value of the **Impedance** Z_i (SI unit: Pa·s/m). The default value is set to the characteristic specific impedance of air 1.2 kg/m^3·343 m/s.

RCL

The RCL model includes all possible circuits involving a source of damping (a resistor R_{ac}), an acoustic mass or inertance (an inductor L_{ac}), and a source of acoustic compliance (a capacitor C_{ac}). The circuit elements are entered in acoustic units. These can be used as a simple model of, for example, the input impedance of a microphone, a loudspeaker cone, or other electromechanical applications. Other applications include general transmission line/circuit models with applications in materials with exotic acoustic properties. More advanced circuit models may be entered manually in the **User defined** option or by coupling to an Electric Circuit model (this requires the AC/DC Module).

Choose an option from the list: **Serial coupling RCL**, **Parallel coupling RCL**, **Parallel LC in series with R**, **Parallel RC in series with L**, **Parallel RL in series with C**, **Serial RC in parallel with L**, **Serial LC in parallel with R**, or **Serial RL in parallel with C**.

Notice the matching diagram and **Equation** section information for each choice. Then enter the following:

- **Equivalent acoustic resistance** R_{ac} (SI unit: kg/(m^4·s)).
- **Equivalent acoustic compliance** C_{ac} (SI unit: m^4·s^2/kg).
- **Equivalent acoustic inertance** L_{ac} (SI unit: kg/m^4).

Generic 711 Coupler—An Occluded Ear-Canal Simulator: Application Library path **Acoustics_Module/Electroacosutic_Transducers/generic_711_coupler**

Physiological

This is a set of simple models to address applications involving interactions of acoustics with the human body. The models comprise human skin, the impedance of the entire human ear including or excluding the pinna, the outward radiation impedance caused by the pinna, and the inward impedance experienced at the ear drum comprising the

drum and the entire inner ear. For the two models of the human ear (with/without pinna), the pressure at the ear drum is automatically calculated. The variable has the form `acpr.imp1.p_ear_drum` and can be plotted in postprocessing.

The whole ear models are based on the geometry of the ear canal and pinna of a specific ear (see Ref. 28-30), but person-to-person variations of these are to be expected. For applications where a specific ear canal geometry can be obtained, better results are expected by explicitly modeling this and applying the eardrum impedance at the end.

Choose an option from the list: **Human skin**, **Outward human ear radiation**, **Human ear drum**, **Human ear without pinna**, or **Human ear, full.** Then select either **From material** (the default) or **User defined** for the following, as required:

- **Ratio of specific heats** γ (SI unit: 1).
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)).
- **Thermal conductivity** k (SI unit: W/(m·K)).
- **Dynamic viscosity** μ (SI unit: Pa·s).

When the **From material** option is selected, remember to add a material under the **Materials** node and assign it to the specific boundary. The boundary will not automatically assume the physical properties of the domain.

Waveguide end impedance

This is a set of idealized models for the acoustic losses at the end of pipes opening into vast domains. The models consider both square and circular cross sections, as well as flanged and unflanged pipe ends. These models are based on a plane wave assumption (propagation below the cut-off frequency).

Choose an option from the list: **Flanged pipe, circular** (the default), **Flanged pipe, rectangular**, **Unflanged pipe, circular (low ka limit)**, or **Unflanged pipe, circular**. Then enter the following as required:

- **Inner radius** a (SI unit: m) or
- **Inner width** w_i (SI unit: m) and **Inner height** h_i (SI unit: m).

Open Pipe: Application Library path **Acoustics_Module/Verification_Examples/open_pipe**

Porous layer

This choice models the acoustic losses of a normally incident field on a porous layer of user-defined thickness d backed by a sound-hard wall. Use this boundary condition as

an alternative to modeling the porous layer explicitly using the Poroacoustics feature. All material models from Poroacoustics are implemented in this feature.

Enter the **Thickness of porous layer** d (Si unit: m) and select a **Poroacoustic model**. The rest of the settings are the same as for Poroacoustics.

Characteristic specific impedance

This is a set of models describing the characteristic impedance associated with three basic wave types: plane wave, cylindrical wave, and spherical wave. Although mostly of academic interest, these serve as good first-order and wave-type specific boundary-condition models of infinite domains (open boundaries). They can be applied to all cases where the tangential components of the acoustic field at the boundary may be ignored. Use the radiation conditions (Plane Wave Radiation, Spherical Wave Radiation, or Cylindrical Wave Radiation) if a nonreflecting open boundary is modeled.

Select a **Wave type**: **Plane wave** (the default), **Cylindrical wave**, or **Spherical wave**. Then enter the **Source location** $\mathbf{x}_0$ (SI unit: m) and for the Cylindrical wave, the **Source axis** $\mathbf{e}_{\text{sa}}$ (SI unit: m).

The theory for the impedance models is discussed in detail in Theory for the Boundary Impedance Models.

Symmetry

The **Symmetry** node adds a boundary condition where there is symmetry in the pressure. Use this condition to reduce the size of a model by cutting it in half where there are symmetries. In pressure acoustics this boundary condition is mathematically identical to the Sound Hard Boundary (Wall) condition.

Plane Wave Radiation

The **Plane Wave Radiation** node adds a radiation boundary condition for a plane wave. If required add an Incident Pressure Field to model an incoming wave. This radiation condition allows an outgoing plane wave to leave the modeling domain with minimal reflections, when the angle of incidence is near normal. Add this node from the **Radiation Conditions** submenu.

The *plane wave* type is suitable for both far-field boundaries and ports. Because many waveguide structures are only interesting in the plane-wave region, it is particularly

relevant for ports. When using the radiation condition on an open far-field boundary, it is recommended to construct the boundary such that the incidence angle is near to normal, this of course requires a priori knowledge of the problem and the solution. See the theory section Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions for details about the equations and the formulation of this nonreflecting boundary condition.

- An estimate of the reflection coefficient R_s, for the spurious waves reflecting off the plane wave radiation boundary, is, for incident plane waves at angle θ, given by the expression:

$$R_s = \left|\frac{\cos\theta - 1}{\cos\theta + 1}\right|^N$$

where N is the order of the boundary condition (here 1 or 2). So at normal incidence ($\theta = 0$) there are no spurious reflections, while, for example, at an incidence angle of 30^0 for $N = 2$ (plane wave radiation in the frequency domain) the amplitude of the spurious reflected wave is 0.5 % of the incident.

Automotive Muffler: Application Library path **COMSOL_Multiphysics/Acoustics/automotive_muffler**

Spherical Wave Radiation

The **Spherical Wave Radiation** node adds a radiation boundary condition for a spherical wave, for which you define the source location. If required, add an Incident Pressure Field to model an incoming wave. This radiation condition allows an outgoing spherical wave to leave the modeling domain with minimal reflections. The geometry of the modeling domain should be adapted to have the outgoing spherical waves coincide with the boundary, in order to minimize spurious reflections. Add this node from the **Radiation Conditions** submenu.

See Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions for details about the equations and the formulation of this nonreflecting boundary condition.

SPHERICAL WAVE RADIATION

Enter coordinates for the **Radiating field source location** r_0 (SI unit: m).

Bessel Panel: Application Library path **Acoustics_Module/Tutorials/bessel_panel**

Cylindrical Wave Radiation

The **Cylindrical Wave Radiation** node adds a radiation boundary condition for a cylindrical wave, for which you define the source location and the source axis direction If required, add an Incident Pressure Field to model an incoming wave. This radiation condition allows an outgoing cylindrical wave to leave the modeling domain with minimal reflections. The geometry of the modeling domain should be adapted to have the outgoing cylindrical waves coincide with the boundary, in order to minimize spurious reflections. Add this node from the **Radiation Conditions** submenu.

See Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions for details about the equations and the formulation of this nonreflecting boundary condition.

CYLINDRICAL WAVE RADIATION

Enter coordinates for the **Radiating field source location** $\mathbf{r}_0$ (SI unit: m) and the **Radiating field source axis** direction $\mathbf{r}_{\text{axis}}$ (dimensionless).

Acoustic Cloaking: Application Library path **Acoustics_Module/Tutorials/acoustic_cloaking**

Incident Pressure Field

The **Incident Pressure Field** node is a subnode to all nonreflecting boundary conditions (plane, cylindrical, spherical wave radiation, and matched boundary). From the **Physics** toolbar, add to Matched Boundary, Plane Wave Radiation, Spherical Wave Radiation, or Cylindrical Wave Radiation nodes. Four options exist for the **Pressure Field Type.** They are **Plane Wave**, **Cylindrical Wave**, **Spherical Wave**, and **User Defined** (the only option for transient simulations).

Plane Wave

If the incident pressure field p_i is a predefined plane wave, it is of the type:

$$p_i = p_0 e^{-i(\mathbf{k}\cdot\mathbf{x})} = p_0 e^{-ik_s\left(\frac{\mathbf{x}\cdot\mathbf{e}_k}{\|\mathbf{e}_k\|}\right)}$$

where p_0 is the wave amplitude, $\mathbf{k}$ is the wave vector with amplitude $k_s = \omega/c$ and wave direction vector $\mathbf{e}_k$, and $\mathbf{x}$ is the location on the boundary.

Cylindrical Wave

If the incident pressure field p_i is a predefined cylindrical wave, it is of the type:

$$p_i = p_0 H_0^{(2)}(k_s r_s) \qquad r_s = \frac{|(\mathbf{x}-\mathbf{x}_0)\times\mathbf{e}_{sa}|}{|\mathbf{e}_{sa}|}$$

where p_0 is the amplitude given at the reference distance $r_{ref} = 0.548/k_s$ (the distance where the Hankel function is one), $k_s = \omega/c$ is the wavenumber, $H_m^{(2)}$ is the Hankel function of the second kind (representing an outgoing cylindrical wave), r_s is the distance from the source axis, $\mathbf{e}_{sa}$ is the direction of the source axis, $\mathbf{x}_0$ is a point on the source axis, and $\mathbf{x}$ is the location on the boundary.

Spherical Wave

If the incident pressure field p_i is a predefined spherical wave (only for 2D axisymmetric and 3D), it is of the type:

$$p_i = p_0 \frac{r_{ref}}{r_s} e^{-ik_s r_s} \qquad r_{ref} = 1\ \text{m} \qquad r_s = |\mathbf{x}-\mathbf{x}_0|$$

where p_0 is the amplitude given at the reference distance of 1 m, $k_s = \omega/c$ is the wavenumber, r_s is the distance from the source, $\mathbf{x}_0$ is the source location of the spherical wave, and $\mathbf{x}$ is the location on the boundary.

For both the cylindrical and the spherical wave options the source and/or source axis should be located outside the computational domain where the radiation condition is applied. The fields should be incident on the radiation boundary.

Transient Interface User Defined

If you are in the Pressure Acoustics, Transient interface the incident pressure field is only of the user-defined type. In this case the incident pressure field needs to be defined as a traveling wave of the form

$$f(\omega t - \mathbf{k}\cdot\mathbf{x})$$

where ω is the angular frequency and $\mathbf{k}$ is the wave vector. The function f is any function, for example, a sine function. This is a requirement for the radiation boundary condition to work properly.

INCIDENT PRESSURE FIELD

From the **Incident pressure field type** list, select **Plane wave** (the default), **Cylindrical Wave**, **Spherical Wave**, or **User Defined** to define the incident pressure field type.

- For **Plane Wave** enter a **Pressure amplitude** p_0 (SI unit: Pa), the **Speed of Sound** c (SI unit: m/s) of the medium defining the incident wave, and the **Wave direction** $\mathbf{e}_k$ (SI unit: m).
- For **Cylindrical Wave** enter a **Pressure amplitude at reference distance** p_0 (SI unit: Pa), the **Speed of Sound** c (SI unit: m/s) of the medium defining the incident wave, a **Source Location** $\mathbf{x}_0$ (SI unit: m), and a **Source Axis** $\mathbf{e}_{sa}$ (the vector need no to be normalized).
- For **Spherical Wave** enter a **Pressure amplitude at reference distance** p_0 (SI unit: Pa), the **Speed of Sound** c (SI unit: m/s) of the medium defining the incident wave, and a **Source Location** $\mathbf{x}_0$ (SI unit: m).
- If **User defined** is selected enter the expression for the **Incident pressure field** p_i (SI unit: Pa) as a function of space.

When calculating, for example, a transfer function use the defined variable `acpr.p_i` for the incident field value. This variable contains phase information which can be difficult to asses otherwise.

Interior Sound Hard Boundary (Wall)

The **Interior Sound Hard Boundary (Wall)** node adds a boundary condition for a sound hard boundary or wall on interior boundaries. Add this node from the **Interior Conditions** submenu. A sound-hard boundary is a boundary at which the normal component of the acceleration is zero:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{up}=0 \qquad -\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{down}=0$$

applied to the two sides of the boundary (the up and down sides). For zero dipole source $\mathbf{q}_d$ and constant fluid density ρ_c, this means that the normal derivative of the

pressure is zero at the boundary. On an interior sound hard boundary the pressure is not continuous but is treated as a so-called slit feature.

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and physics interfaces being modeled. This feature works well for cases like opposing parallel boundaries. In other cases, use a **Destination Selection** subnode to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

PERIODICITY SETTINGS

Select a **Type of periodicity**: **Continuity** (the default), **Floquet periodicity** (Bloch periodicity), or **Cyclic symmetry**, or **Antiperiodicity**.

- For **Floquet periodicity**, also known as Bloch periodicity, enter a **k-vector for Floquet periodicity** $\mathbf{k}_F$ (SI unit: rad/m) for the **x**, **y**, and **z** coordinates (3D components), or the **r** and **z** coordinates (2D axisymmetric components), or **x** and **y** coordinates (2D components).

  This condition is used to model infinite periodic structures with non-normal incident pressure fields or excitations. Use it to model, for example, a large perforated plate with an oblique incident wave with wave vector $\mathbf{k}$ (and set $\mathbf{k}_F = \mathbf{k}$) by only analyzing one hole or one subset of holes that is periodic.

- For **Cyclic symmetry** select a **Sector angle**: **Automatic** (the default), or **User defined**. For **User defined** enter a value for θ_S (SI unit: rad). Enter an **Azimuthal mode number** m (dimensionless).

  This condition is used to model any geometry that has a cyclic periodic structure such as a microphone or a loudspeaker driver. Setting the azimuthal mode number determines what mode is analyzed. The response of the full system to an external excitation is in general a linear combination of many different modes.

In the time domain both the **Cyclic symmetry** and the **Floquet periodicity** boundary conditions reduce to the continuity condition.

To optimize the performance of the **Floquet periodicity** and the **Cyclic symmetry** conditions, it is recommended that the source and destination meshes are identical. This can be achieved by first meshing the source boundary or edge and then copy the mesh to the destination boundary or edge. When the **Periodic Condition** stretches across regions with a mix of default material models, PMLs, background pressure fields, or background acoustic fields, it is recommended to add one **Periodic Condition** for each such boundaries.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

ORIENTATION OF SOURCE

For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

The *Porous Absorber* model uses Floquet periodic boundary conditions to model an infinite porous absorber used for sound proofing. The Application Library path is **Acoustics_Module/Building_and_Room_Acoustics/porous_absorber**

Axial Symmetry

The **Axial Symmetry** node is a default node added for all 2D and 1D axisymmetric components. The boundary condition is active on all boundaries on the symmetry axis.

Continuity

Continuity is available as an option at interfaces between parts in a pair, from the **Pairs >** sub menu.

This condition gives continuity in total pressure and in the normal acceleration over the pair. Subscripts “src” and “dest” in the equation refer to the source and destination in the pair, that is, the two sides in the pair:

$$p_{t,src} = p_{t,dest}$$

$$-\mathbf{n} \cdot \left[-\left(\frac{1}{\rho_c} (\nabla p_t - \mathbf{q}_d) \right)_{src} - \left(-\frac{1}{\rho_c} (\nabla p_t - \mathbf{q}_d) \right)_{dest} \right] = 0$$

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Pressure Acoustics

The **Pressure Acoustics** node adds the equations for time-harmonic and eigenfrequency acoustics modeling in the frequency domain. In the **Settings** window, define the properties for the acoustics model and model inputs including the background pressure and temperature.

The advanced fluid models are defined using individual physics feature nodes: Poroacoustics and Narrow Region Acoustics

For details about the available fluid models, see Theory for the Equivalent Fluid Models.

For more information about using variables during the results analysis, see Postprocessing Variables.

MODEL INPUTS

If **Ideal gas** is selected as the **Fluid model**, enter a **Temperature** T and an **Absolute pressure** p_A. For **User defined** enter a value or an expression for the absolute pressure (SI unit: Pa) and the **Temperature** (SI unit: K) in the field. This input is always available.

In addition, select a temperature field defined by a Heat Transfer interface or a Non-Isothermal Flow interface (if any), for example.

Non-Isothermal Flow requires the addition of the Heat Transfer Module or CFD Module.

If applicable, select a pressure defined by a Fluid Flow interface present in the model. For example, select **Absolute pressure (spf)** to use the absolute pressure defined by a

Laminar Flow interface **spf**. This makes it possible to use a system-based (gauge) pressure, while automatically including the reference pressure in the absolute pressure.

The input to these fields influences the value of the material parameters in the model. Typically, the density ρ and the speed of sound c in the model depend on the absolute pressure and/or the temperature. Picking up any of those from another physics interface typically results in $\rho = \rho(\mathbf{x})$ and $c = c(\mathbf{x})$ to be specially varying.

Detailed aeroacoustic models that take into account the full background flow (including the movement of the fluid) are available under the Aeroacoustics Interfaces.

PRESSURE ACOUSTICS MODEL

To define the properties of the bulk fluid, select a **Fluid model** from the list: **Linear elastic** (the default), **Linear elastic with attenuation**, **Viscous**, **Thermally conducting**, **Thermally conducting and viscous**, or **Ideal gas**.

The fluid models represent different bulk loss or attenuation mechanisms for (applied in a homogenized way) or ways of defining the fluid properties of the fluid. Some of these models are sometimes referred to as an *equivalent fluid model*. The loss model can be a theoretical model or a model based on measurement data for the attenuation in the fluid.

Losses in porous materials are defined in Poroacoustics and viscothermal losses in narrow regions are defined in Narrow Region Acoustics.

The fluid models may be roughly divided into these categories:

- *General fluids*:
  - **Linear elastic**. Go to Defining a Linear Elastic Fluid Model.
  - **Linear elastic with attenuation**. Go to Defining a Linear Elastic with Attenuation Fluid Model.
  - **Ideal gas**. Go to Defining an Ideal Gas Fluid Model.

- *Fluids with bulk viscous and/or thermal losses (for plane propagating waves)*:
  - **Viscous**. Go to Defining a Viscous Fluid Model.
  - **Thermally conducting**. Go to Defining a Thermally Conducting Fluid Model.
  - **Thermally conducting and viscous**. Go to Defining a Thermally Conducting and Viscous Fluid Model.

The theory for the fluid models is in the section Theory for the Equivalent Fluid Models.

Defining a Linear Elastic Fluid Model

Linear elastic is the default. When the material parameters are real values this corresponds to a lossless compressible fluid. From the **Specify** list, select **Density and speed of sound** (the default), **Impedance and wave number**, or **Bulk modulus and density**. To add user defined losses, in a general way, specify the properties as complex-valued data.

For each of the following, the default values (when applicable) are taken **From material** or for **User defined** enter other values or expressions.

- For **Density and speed of sound**, define the **Speed of sound** c (SI unit: m/s) and **Density** ρ (SI unit: kg/m^3).
- For **Impedance and wave number**, define the **Characteristic acoustic impedance** Z (SI unit: Pa·s/m) and enter a **Wave number** k (SI unit: rad/m).
- For **Bulk modulus and density**, define the **Equivalent bulk modulus** K (SI unit: Pa) and **Density** ρ (SI unit: kg/m^3). Selecting **User defined** is well suited for entering the properties of a user defined porous material fluid model. Predefined porous models exist in the Poroacoustics domain feature.

Defining a Linear Elastic with Attenuation Fluid Model

The **Linear elastic with attenuation** model adds a user defined attenuation to the fluid; this data is typically based on experimental data for the attenuation coefficient α. Adding attenuation makes the wave number k complex valued. For example, a plane wave $p(x)$ moving in the x-direction is attenuated according to

$$k = \frac{\omega}{c} - i\alpha$$

$$p(x) = e^{-ikx} = e^{-i(\omega/c)x} e^{-\alpha x}$$

When the attenuation is defined in Np per unit length, the wave has a spatial exponential decay governed by the attenuation coefficient.

The default **Speed of sound** c (SI unit: m/s) and **Density** ρ (SI unit: kg/m^3) are taken **From material**. For **User defined** enter other values or expressions for one or both options.

Select an **Attenuation type**: **Attenuation coefficient, Np per unit length** (the default) to define an attenuation coefficient α in Np/m (nepers per meter), **Attenuation coefficient, dB per unit length** to define an attenuation coefficient α in dB/m (decibel per meter), or **Attenuation coefficient, dB per wavelength** to define an attenuation coefficient α in dB/λ (decibel per wavelength). For any selection, enter a value or expression in the **Attenuation coefficient** α edit field.

Linear Elastic with Attenuation Fluid Model

Defining an Ideal Gas Fluid Model

For **Ideal Gas** you can also edit the **Model Inputs** section. For each of the following, the default values are taken **From material**. For **User defined** enter other values or expressions for any or all options.

- Select a **Gas constant type**: **Specific gas constant** R_s (SI unit: J/(kg·K) (the default) or **Mean molar mass** M_n (SI unit: kg/mol). For **Mean molar mass** the molar gas constant (universal gas constant) $R = 8.314$ J/(mol·K), is used as the built-in physical constant.
- From the **Specify Cp or γ** list, select **Heat capacity at constant pressure** C_p (SI unit: J / (kg·K)) (the default) or **Ratio of specific heats** γ. For common diatomic gases such as air, $\gamma = 1.4$ is the standard value.

Defining a Viscous Fluid Model

The **Viscous** fluid model adds the attenuation due to bulk viscous losses. This type of model is relevant in highly viscous fluids or when acoustic waves are traveling over large distances (relative to the wavelength). The losses apply for plane propagating waves. This is not a model for viscous boundary layer losses in narrow regions, for this see Narrow Region Acoustics. For each of the following, the default values are taken **From material**. For **User defined** enter other values or expressions for any or all options.

- **Speed of sound** c (SI unit: m/s).
- **Density** ρ (SI unit: kg/m^3).

- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Bulk viscosity** μ_B (SI unit: Pa·s).

Viscous Fluid Model

Defining a Thermally Conducting Fluid Model

The **Thermally conducting** fluid model adds the attenuation due to thermal conduction effects in the bulk of the fluid. This type of model is relevant in fluids that have high thermal conduction or when acoustic waves are traveling over large distances (relative to the wavelength). The losses apply for plane propagating waves. This is not a model for thermal boundary layer losses in narrow regions, for this see Narrow Region Acoustics. For each of the following, the default values are taken **From material**. For **User defined** enter other values or expressions for any or all options.

- **Speed of sound** c (SI unit: m/s).
- **Density** ρ (SI unit: kg/m^3).
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)).
- **Ratio of specific heats** γ (dimensionless).
- **Thermal conductivity** k (SI unit: W/(m·K)).

Thermally Conducting Fluid Model

Defining a Thermally Conducting and Viscous Fluid Model

The **Thermally conducting and viscous** fluid model adds the bulk attenuation that is due to the combined effect of viscous losses and thermal conduction. This type of model is relevant in fluids that have high thermal conduction and viscosity or when acoustic waves are traveling over large distances (relative to the wavelength). The losses apply for plane propagating waves. This is not a model for thermoviscous boundary layer losses in narrow regions, for this see Narrow Region Acoustics. For each of the following, the default values are taken **From material**. For **User defined** enter other values or expressions for any or all options.

- **Speed of sound** c (SI unit: m/s).
- **Density** ρ (SI unit: kg/m^3).
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)).

- **Ratio of specific heats** γ (dimensionless).
- **Thermal conductivity** k (SI unit: W/(m·K)).
- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Bulk viscosity** μ_B (SI unit: Pa·s).

It is possible to assess the magnitude of the losses due to thermal conduction and viscosity, that is, the power dissipation density (SI unit: W/m^3). This is done during the analysis process by plotting the variables for:

- the viscous power dissipation density (`acpr.diss_visc`),
- the thermal power dissipation density (`acpr.diss_therm`), or
- the combined total power dissipation density (`acpr.diss_tot`).

Thermally Conducting and Viscous Fluid Model

Poroacoustics

The **Poroacoustics** node defines a fluid domain with a porous material modeled in a homogenized way using a, so-called, equivalent fluid model. Several models exist to define the attenuation and dispersion experienced by the pressure waves as they propagate in the porous domain. The different models are described below.

About the Poroacoustics Models

POROACOUSTICS MODEL

Select a **Poroacoustics model**: **Delany-Bazley-Miki** (the default), **Zwikker-Kosten**, **Attenborough**, **Wilson**, **Johnson-Champoux-Allard**, **Johnson-Champoux-Allard-Lafarge**, **Johnson-Champoux-Allard-Pride-Lafarge**, **Williams EDFM**, or **Wood**.

FLUID PROPERTIES

Enter the properties of the saturating fluid that is inside of the porous material. These settings are common to most porous models. By default the **Fluid material** uses the **Domain material**.

The following properties are available based on the **Poroacoustics model** selected above. The default values are taken **From material**. For **User defined** enter a different value or expression.

- **Speed of sound** c (SI unit: m/s) of the saturating fluid.
- **Density** ρ_f (SI unit: kg/m^3) of the saturating fluid (the fluid density).

If any other than the default **Delany-Bazley-Miki** is selected the following properties are also require, depending on the selection. These material parameters are necessary as the more advanced models include the losses associated with viscosity and thermal conduction in a more or less detailed way:

- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)).
- **Ratio of specific heats** γ (dimensionless).
- **Thermal conductivity** k (SI unit: W/(m·K)).
- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Bulk modulus** K_f (SI unit: Pa). This is the bulk modulus of the fluid.

The implemented poroacoustics models are all applicable for any fluid, except the empirically determined Delany-Bazley-Miki model, which applies only to air. See Ideal Gas and General Fluid Options for details.

POROUS MATRIX PROPERTIES

In this section, enter the properties that describe the porous material. By default the **Porous elastic material** uses the **Domain material** (the material defined for the domain). Select another material as needed. For example, create your own material that contains the properties of a given porous material and refer to it here. Here you also select if you want to use a *rigid* frame approximation or a *limp* frame approximation for the porous material (where applicable). The poroacoustic model defines complex-valued frequency-dependent expressions for both the bulk modulus and the equivalent density.

Based on the **Poroacoustics model** selected, enter the following settings for the porous matrix.

- Delany-Bazley-Miki
- Zwikker-Kosten
- Attenborough
- Wilson
- Johnson-Champoux-Allard
- Johnson-Champoux-Allard-Lafarge
- Johnson-Champoux-Allard-Pride-Lafarge
- Williams effective density fluid model (EDFM)
- Wood

Delany-Bazley-Miki

The Delany-Bazley-Miki model is an empirical model used to describe fibrous materials such as rockwool or glass fiber. The model can be used for materials with a porosity, ε_p, close to one. For **Delany-Bazley-Miki** the **Flow resistivity** R_f (SI unit: Pa·s/m^2) uses values **From material**. For **User defined** enter a value or expression.

Select an option from the **Constants** list: **Delany-Bazley** (the default), **Miki**, **Qunli**, **Mechel, glass fiber, low X**, **Mechel, glass fiber, high X**, **Mechel, rock fiber, low X**, **Mechel, rock fiber, high X**, **Komatsu**, **Modified Champoux and Allard**, or **User defined**. For **User defined** enter values in the **C1** to **C8** fields. The models are empirical and based on fitting to measured data. This means the models have different regions of applicability with respect to the flow resistivity R_f, the frequency f, the material type, and the parameter X defined as

$$X = \frac{f \cdot \rho_f}{R_f}$$

All the models are applicable for materials with a porosity ε_p close to 1. The model informations are listed in Table 2-2. See also Ref. 23 and the relevant section in About the Poroacoustics Models for further details.

TABLE 2-2: DELANY-BAZLEY-MIKI MODEL OPTIONS AND APPLICABILITY

MODEL NAME	APPLICABILITY
Delany-Bazley	Glass and rock wool with: $0.01 \le X \le 1$ $10^3 \le R_f \le 50 \cdot 10^3$ Pasm^{-2}
Miki	Glass and rock wool with: $0.01 \le X \le 1$ $10^3 \le R_f \le 50 \cdot 10^3$ Pasm^{-2}
Qunli	Porous plastic and open foams: $200 \le f \le 2000$ Hz $3 \cdot 10^3 \le R_f \le 24 \cdot 10^3$ Pasm^{-2}
Mechel, glass fiber, low X	Glass fiber: $X \le 0.025$
Mechel, glass fiber, high X	Glass fiber: $X \ge 0.025$
Mechel, rock fiber, low X	Rock fiber: $X \le 0.025$
Mechel, rock fiber, high X	Rock fiber: $X \ge 0.025$
Komatsu	Glass and rock wool: $6 \cdot 10^3 \le R_f \le 73 \cdot 10^3$ Pasm^{-2}
Modified Champoux and Allard	$45 \le f \le 11 \cdot 10^3$ Hz

Flow resistivity R_f is easy to measure and is independent of frequency.

Absorptive Muffler: Application Library path **Acoustics_Module/Automotive/absorptive_muffler**

Zwikker-Kosten

The Zwikker-Kosten model is a two parameter semi-empirical model. It is one of the earliest equivalent fluid models for porous materials. The model assumes that the pores are cylinder-like with an effective hydraulic radius H_r. See Ref. 15 and About the

Poroacoustics Models for further details.

For **Zwikker-Kosten** select a **Porous matrix approximation**: **Rigid** (the default) or **Limp**. Then based on your choice, the default value for each of the following parameters is taken **From material**. For **User defined** enter another value or expression.

- **Porosity** ε_p (dimensionless).
- **Hydraulic radius of pores** H_r (SI unit: m).
- **Drained density of porous material** ρ_d (SI unit: kg/m^3).

Attenborough

The Attenborough model is also based on the cylindrical-like pore assumption. It is a so-called four parameter semi-empirical model. The model is an extension of the Zwikker-Kosten model and has two additional input parameters. It accounts for the tortuosity (high frequency limit) τ_∞, which is related to the orientation of the pores relative to the propagation direction. The hydraulic diameter of the pores is replaced by an expression that included the flow resistivity R_f, and a fitting parameter, b (this parameter is related to the anisotropy of the pore distribution). See Ref. 9, Ref. 16, and About the Poroacoustics Models for details.

For **Attenborough** select a **Porous matrix approximation**: **Rigid** (the default) or **Limp**. Then based on your choice, the default value for most of the following parameters is taken **From material**. For **User defined** enter another value or expression.

- **Porosity** ε_p (dimensionless).
- **Flow resistivity** R_f (SI unit: $Pa{\cdot}s/m^2$).
- **Tortuosity factor** (high frequency limit) τ_∞ (dimensionless). The default is 1.
- **Fitting parameter** b (dimensionless). The default is 1.
- **Drained density of porous material** ρ_d (SI unit: kg/m^3).

Wilson

The Wilson model is a generalization of the semi-analytical models for porous materials with constant cross section and parallel pores. This model is intended to match the middle frequency behavior of a porous material. See Ref. 17, Ref. 9, Ref. 18, and About the Poroacoustics Models for further details.

For **Wilson** select a **Porous matrix approximation**: **Rigid** (the default) or **Limp**. Then based on your choice, the default value for the following parameters is taken **From material**. For **User defined** enter another value or expression.

- **Porosity** ε_p (dimensionless).

- **Bulk modulus infinite frequency limit** K_∞ (SI unit: Pa).
- **Density infinite frequency limit** ρ_∞ (SI unit: kg/m^3).
- **Entropy-mode relaxation time** τ_{ent} (SI unit: s).
- **Vorticity-mode relaxation time** τ_{vor} (SI unit: s).

Approximative expressions for the two relaxation times that are based on intrinsic material properties are given in the theory section for the Wilson model.

Johnson-Champoux-Allard

The Johnson-Champoux-Allard (or JCA model) is a five parameter semi-empirical model for describing a large range of porous materials with rigid (or limp) frames. As input the model requires the viscous L_v, and the thermal L_{th}, characteristic lengths (these are sometimes denoted Λ and Λ'). These replace the hydraulic radius used in simpler models and account for the thermal and viscous losses that appear at the acoustic boundary layer at pore walls. See About the Poroacoustics Models for details.

For **Johnson-Champoux-Allard** select a **Porous matrix approximation**: **Rigid** (the default) or **Limp**. Then based on your choice, the default value for most of the following parameters is taken **From material**. For **User defined** enter another value or expression.

- **Porosity** ε_p (dimensionless).
- **Flow resistivity** R_f (SI unit: Pa·s/m^2).
- From the **Specify** list:
  - Select **Viscous characteristic length** (the default) to directly enter an expression for L_v (SI unit: m). The default expression is `sqrt(acpr.mu*acpr.tau*8/(acpr.Rf*acpr.epsilon_p))`, which corresponds to $s = 1$.
  - Select **Viscous characteristic length parameter** to enter a value for s (dimensionless) (the default is 1).
- **Thermal characteristic length** L_{th} (SI unit: m). The default expression is `2*acpr.Lv`.
- **Tortuosity factor** (high frequency limit) τ_∞ (dimensionless). The default is 1.
- **Drained density of porous material** ρ_d (SI unit: kg/m^3).

The Johnson-Champoux-Allard (JCA) model with a rigid (motionless) frame (skeleton) is sometimes referred to as a Biot equivalent fluid model with a rigid porous matrix.

Johnson-Champoux-Allard-Lafarge

The Johnson-Champoux-Allard-Lafarge (or JCAL model) is an extension to the JCA model. It introduces corrections to the bulk modulus thermal behavior at low frequencies which is not captured by the JCA model. It introduces the static thermal permeability parameter κ'_0 and, thus, has one more parameter than the JCA model. It is a six parameters semi-empirical model. See Ref. 13, Ref. 9, and About the Poroacoustics Models for further details.

For **Johnson-Champoux-Allard-Lafarge** the settings are the same as for Johnson-Champoux-Allard with the addition of the **Static thermal permeability** κ'_0 (SI unit: m^2) setting.

Johnson-Champoux-Allard-Pride-Lafarge

The Johnson-Champoux-Allard-Pride-Lafarge (or JCAPL model) further extends the JCAL models by introducing a static viscous τ_0 and thermal τ'_0 tortuosity.

For **Johnson-Champoux-Allard-Pride-Lafarge** the settings are the same as for Johnson-Champoux-Allard plus these additional parameters:

- **Static viscous tortuosity** τ_0 (dimensionless). The default is 1.
- **Static thermal tortuosity** τ'_0 (dimensionless). The default is 1.
- **Static thermal permeability** κ'_0 (SI unit: m^2).

Williams effective density fluid model (EDFM)

Williams EDFM is and *effective density fluid model (EDFM)* used to model the propagation of acoustic waves in sediments. The model assumes that the bulk and shear moduli of the frame of the porous material are negligible, meaning that the porous frame is limp. See Ref. 24 and About the Poroacoustics Models for further details.

The **Porous Matrix Properties** (the properties for the sediment) for **Williams EDFM** have the default values taken **From material**. For **User defined** enter another value or expression:

- **Grain density** ρ_{gr} (SI unit: kg/m^3)..
- **Grain bulk modulus** K_{gr} (SI unit: Pa).
- **Porosity** ε_p (dimensionless).
- **Tortuosity factor** (high frequency limit) τ_∞ (dimensionless). The default is 1.
- **Permeability** κ_p (SI unit: m^2).
- **Hydraulic radius of pores** H_r (SI unit: m).

Wood

The **Wood** models is a model for a fluid mixture or a fluid suspension (solid inclusions completely surrounded by fluid). The Woods formula for the sound velocity is determined by using the effective bulk modulus of the suspension and the volume average density. As the Williams EDFM this model gives effective values for the mixture. This model is exact for low frequencies, when the wavelength is much larger than the size of the inclusions. See Ref. 25 and About the Poroacoustics Models for further details.

The **Porous Matrix Properties** (the properties for the inclusions) for the **Wood** model are entered in the **Inclusion properties** table. For each **Inclusion** (any number of inclusions can be added, the number is automatically incremented) enter the **Volume fraction**, the **Bulk modulus**, and the **Density**. Add a new row (inclusion) by clicking the plus sign below the table. The table may be saved or loaded from a file.

Narrow Region Acoustics

The **Narrow Region Acoustics** node is a fluid model for viscous and thermal boundary-layer induced losses in channels and ducts. The losses due to the viscous and thermal dissipation in the acoustic boundary layer are homogenized and smeared on the fluid. This equivalent-fluid model can be used in long tubes of constant cross section (or only in a slowly varying cross section) instead of a fully detailed thermoviscous acoustic model. This type of model has a low computational cost compared to a thermoviscous acoustic model. The models are applicable for all fluids, that is, both gases and liquids.

About the Narrow Region Acoustics Models

NARROW REGION ACOUSTICS

Select a **Duct type**: **Wide duct approximation** (the default), **Very narrow circular duct (isothermal)**, **Slit**, **Circular duct**, **Rectangular duct**, **Equilateral triangular duct**, or **User defined**.

The **Slit**, **Circular duct**, **Rectangular duct**, and **Equilateral triangular duct** models are applicable as long as the cross section dimension is much smaller then the wavelength and the boundary layer thickness is smaller than the wavelength. The cross section parameter can be a slowly varying function of space.

Based on your selection above, the default values for the following are taken **From material**. For **User defined** enter other values or expressions for any or all options.

- **Speed of sound** c (SI unit: m/s)
- **Density** ρ (SI unit: kg/m^3)
- **Ratio of specific heats** γ (SI unit: 1). In many liquids the value of γ is close to 1, the exact value can be derived from the expression $(1 - (\alpha_p^2 T)/(\beta_T \rho C_p))^{-1}$ where the (isobaric) coefficient of thermal expansion α_p and the isothermal compressibility β_T is used.
- **Dynamic viscosity** μ (SI unit: Pa·s)

The following are available for **Wide duct approximation**, **Slit**, **Circular duct**, **Rectangular duct**, and **Equilateral triangular duct**:

- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K))
- **Thermal conductivity** k (SI unit: W/(m·K))
- For **Wide duct approximation** enter a **Hydraulic diameter** H_d (SI unit: m).

  This model can be used in ducts of any cross section as long as the hydraulic diameter of the duct (four times the cross section area divided by circumference) is much larger than the viscous boundary layer thickness.
- For **Very narrow circular duct (isothermal)** enter the duct **Radius** a (SI unit: m).

  This model is only valid for very narrow circular ducts where isothermal conditions apply. The radius of the duct has to be much smaller than the thickness of the thermal boundary layer. For this model it is assumed that the compressibility (bulk modulus) of the fluid also takes the isothermal value.

- For **Slit** enter the slit **Height** h (SI unit: m).

  Use this model in narrow slit domains to include the damping and attenuation that occurs here because of the losses in the viscous and thermal boundary layer.
- For **Circular duct** enter the duct **Radius** a (SI unit: m). This model is useful for modeling the damping and attenuation that occurs when acoustic waves propagate in all tubing systems of small cross section dimensions.
- For **Rectangular duct** enter the duct **Side lengths** W and H (SI unit: m).

  Use this model for wave guides and ducts with a rectangular cross section. Also see Advanced Physics Options for additional settings.
- For **Equilateral triangular duct** enter the duct **Side length** d (SI unit: m).

  Use this model for wave guides and ducts with an equilateral triangular cross section.
- For **User defined** enter the value for the **Complex wave number** k_c (SI unit: rad/m) and (specific) **Complex acoustic impedance** Z_c (SI unit: Pa·s/m).

ADVANCED PHYSICS OPTIONS

To display this section, click the **Show** button ( ), select **Advanced Physics Options** and choose **Rectangular duct** as the **Duct type**.

The **Number of terms for the sum** N (used to describe this model) (dimensionless) can be set. The default is $N = 100$ and should cover most cases, see under Slits, Circular Ducts, Rectangular Ducts, and Equilateral triangular Ducts.

	Lumped Receiver Connected to Test Setup with an 0.4-cc Coupler: Application Library path **Acoustics_Module/Electroacoustic_Transducer/lumped_receiver_04cc**

Background Pressure Field

Add a **Background Pressure Field** node to model an incident pressure wave or to study the scattered pressure field p_s, which is defined as the difference between the total acoustic pressure p_t and the background pressure field p_b:

$$p_t = p_b + p_s$$

This feature sets up the equations in a so-called scattered field formulation where the dependent variable is the scattered field $p = p_s$. In a model where the background

pressure field is not defined on all acoustic domains (or it is different) continuity is automatically applied in the total field p_t on internal boundaries between domains.

The background pressure field can be a function of space and, for The Pressure Acoustics, Transient Interface using a Time Dependent study, a function of time.

For a Frequency Domain study type, the frequency of the background pressure field is the same as for the dependent variable p.

BACKGROUND PRESSURE FIELD

Select a **Background pressure field type**: **Plane wave** (the default), **Cylindrical Wave**, **Spherical Wave**, or **User defined**.

Plane Wave

The **Plane wave** option defines the background pressure field p_b of the type:

$$p_b = p_0 e^{-i(\mathbf{k}\cdot\mathbf{x})} = p_0 e^{-ik_s\left(\frac{\mathbf{x}\cdot\mathbf{e}_k}{\|\mathbf{e}_k\|}\right)}$$

where p_0 is the wave amplitude, $\mathbf{k}$ is the wave vector with amplitude $k_s = \omega/c$ and wave direction vector $\mathbf{e}_k$, and $\mathbf{x}$ is the location on the boundary.

For **Plane wave** enter values for the **Pressure amplitude** p_0 (SI unit: Pa) and **Wave direction** $\mathbf{e}_k$ (dimensionless). Select to define the **Speed of sound** c (SI unit: m/s), of the fluid defining the field, either **From material** or **User defined**.

Cylindrical Wave

The **Cylindrical Wave** option defines the background pressure field p_b as a predefined cylindrical wave:

$$p_b = p_0 H_0^{(2)}(k_s r_s) \qquad r_s = \frac{\left|(\mathbf{x}-\mathbf{x}_0)\times\mathbf{e}_{sa}\right|}{\left|\mathbf{e}_{sa}\right|}$$

where p_0 is the amplitude given at the reference distance $r_{ref} = 0.548/k_s$ (the distance where the Hankel function is one), $k_s = \omega/c$ is the wavenumber, $H_m^{(2)}$ is the Hankel function of the second kind (representing an outgoing cylindrical wave), r_s is the distance from the source axis, $\mathbf{e}_{sa}$ is the direction of the source axis, $\mathbf{x}_0$ is a point on the source axis, and $\mathbf{x}$ is the location on the boundary.

For **Cylindrical Wave** enter a **Pressure amplitude at reference distance** p_0 (SI unit: Pa), the **Speed of Sound** c (SI unit: m/s) of the medium defining the incident wave, a **Source**

Location $\mathbf{x}_0$ (SI unit: m), and a **Source Axis** $\mathbf{e}_{sa}$ (the vector does not need to be normalized).

Spherical Wave

The **Spherical Wave** option defines the background pressure field p_b as a predefined spherical wave (only for 2D axisymmetric and 3D):

$$p_b = p_0 \frac{r_{ref}}{r_s} e^{-ik_s r_s} \qquad r_{ref} = 1\ \text{m} \qquad r_s = |\mathbf{x} - \mathbf{x}_0|$$

where p_0 is the amplitude given at the reference distance of 1 m, $k_s = \omega/c$ is the wavenumber, r_s is the distance from the source, $\mathbf{x}_0$ is the source location of the spherical wave, and $\mathbf{x}$ is the location on the boundary.

For **Spherical Wave** enter a **Pressure amplitude at reference distance** p_0 (SI unit: Pa), the **Speed of Sound** c (SI unit: m/s) of the medium defining the incident wave, and a **Source Location** $\mathbf{x}_0$ (SI unit: m).

For both the cylindrical and the spherical wave options the source and/or source axis should be located outside the domain where the background pressure field feature is defined.

Use Defined

For the **User defined** option enter an expression for the **Background pressure field** p_b (SI unit: Pa). This is the only option in the transient interface.

- *Acoustic Cloaking*: Application Library path **Acoustics_Module/Tutorials/acoustic_cloaking**
- *Acoustic Scattering off an Ellipsoid*: Application Library path **Acoustics_Module/Tutorials/acoustic_scattering**

Matched Boundary

The **Matched Boundary** node adds a matched boundary condition. Like the radiation boundary conditions, it belongs to the class of *nonreflecting boundary conditions* (NRBCs). The Incident Pressure Field subnode is available if required.

Properly set up, the matched boundary condition allows one mode with wave number k_1 (set $k_2 = k_1$), or two modes with wave numbers k_1 and k_2, to leave the modeling domain with minimal reflections. The equation is given by

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p - \mathbf{q}_d)\right) + \frac{i\left(\left(\frac{\omega}{c_c}\right)^2 + k_1 k_2\right)p + i\Delta_T p}{\rho_c(k_1 + k_2)} = \frac{i\left(\left(\frac{\omega}{c_c}\right)^2 + k_1 k_2\right)p_i + i\Delta_T p_i}{\rho_c(k_1 + k_2)} + \mathbf{n}\cdot\frac{1}{\rho_c}\nabla p_i$$

Here Δ_T, for a given point on the boundary, refers to the Laplace operator in the tangential plane at that point, while p_i is the amplitude of an optional incoming plane wave with wave vector $\mathbf{k}$. In addition to p_i, specify the propagation direction $\mathbf{n}_k$, whereas the wave number is defined by $k_{eq} = \omega/c_c$ in 3D and in 2D.

$$k_{eq}^2 = \left(\frac{\omega}{c_c}\right)^2 - k_z^2$$

The matched boundary condition is particularly useful for modeling acoustic waves in ducts and waveguides at frequencies below the cutoff frequency for the second excited transverse mode. In such situations set $k_1 = \omega/c_c$ and $k_2 = \omega_1/c_c$, where $\omega_1 = 2\pi f_1$, and f_1 is the cutoff frequency for the first excited mode. The cutoff frequency or wave number may be found using a Boundary Mode Acoustics model. When $k_1 = k_2 = \omega/c_c$, the matched boundary condition reduces to the time-harmonic plane-wave radiation boundary condition.

MATCHED BOUNDARY

From the **Match** list, select **One mode** (the default) or **Two modes**. Then enter a **Wave number** (SI unit rad/m) based on the selection: k_1 for **One mode** and k_1 and k_2 for **Two modes**.

Far-Field Calculation

Use the **Far-Field Calculation** node to apply the source boundaries for the near-to-far-field transformation and to specify a name for the acoustic far-field variable. This feature allows the calculation of the pressure field outside the computational domain at any distance including phase. The far-field boundary needs to enclose all sources and scatterers.

FAR-FIELD CALCULATION

Enter a **Far-field variable name** for the far-field acoustic pressure field (the default is `pfar`).

Select a **Type of integral**: **Integral approximation for $r \to \infty$** (the default) to compute the value in The Far-Field Limit or **Full integral** to compute The Helmholtz-Kirchhoff Integral Representation.

If required, use symmetry planes in your model when calculating the far-field variable. The symmetry planes have to coincide with one of the Cartesian coordinate planes. For each of these planes, select the type of symmetry check boxes: **Symmetry in the x=0 plane**, **Symmetry in the y=0 plane**, or **Symmetry in the z=0 plane**.

This selection should match the boundary condition used for the symmetry boundary. With these settings, the parts of the geometry that are not in the model, for symmetry reasons, can be included in the far-field analysis.

ADVANCED SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

The option **Use polynomial-preserving recovery for the normal gradient** is selected per default on internal boundaries. This means that the far-field feature automatically uses the polynomial-preserving recovery operator `ppr()` to get an enhanced evaluation of the normal derivative of the pressure. This increases the precision of the far-field calculation. If you click to clear this check box this removes all instances of the operator from the equations.

The `ppr()` operator is not added when the far-field calculation is performed on an external boundary or a boundary adjacent to a perfectly matched layer (PML) domain. In the latter case, the `down()` or `up()` operator is added in order to retrieve values of variables from the physical domain only.

In these cases, use a single boundary layer mesh on the inside of the outer boundary or on the inside of the PML layer to enhance the precision of the far-field calculation.

- Far-Field Calculation
- Evaluating the Acoustic Field in the Far-Field Region
- Several dedicated acoustics plots rely on the far-field calculation, see Dedicated Acoustics Plots for Postprocessing
- ppr and pprint and up and down (operators) in the *COMSOL Multiphysics Reference Manual*

- *Acoustic Scattering off an Ellipsoid*: Application Library path **Acoustics_Module/Tutorials/acoustic_scattering**
- *Bessel Panel*: Application Library path **Acoustics_Module/Tutorials/bessel_panel**

Interior Normal Acceleration

The **Interior Normal Acceleration** node adds a normal acceleration on an interior boundary and ensures that the pressure is noncontinuous here. Add this node from the **Interior Conditions** submenu. The pressure has a so-called slit condition on this boundary. This boundary condition can be used to model sources as, for example, the movement of a speaker cone modeled as a boundary. The condition adds the normal part of an acceleration $\mathbf{a}_0$:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{up}=\mathbf{n}\cdot\mathbf{a}_0 \qquad -\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{down}=\mathbf{n}\cdot\mathbf{a}_0$$

Alternatively, specify the inward acceleration a_n. The normal of the boundary is interpreted as pointing outward.

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{up}=a_n \qquad -\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{down}=a_n$$

INTERIOR NORMAL ACCELERATION

Select a **Type**: **Acceleration** (the default) or **Inward acceleration**.

- For **Acceleration** enter values for the components of the **Acceleration** $\mathbf{a}_0$ (SI unit: $\mathrm{m/s^2}$).
- For **Inward acceleration** enter the value of the **Inward acceleration** a_n (SI unit: $\mathrm{m/s^2}$) in the normal direction. Use a positive value for inward acceleration or a negative

value for outward acceleration. The normal of the boundary points outward. The normal is depicted as a red arrow in the graphics window on the selected boundary.

- *Cylindrical Subwoofer*: Application Library path **Acoustics_Module/Tutorials/cylindrical_subwoofer**
- *Lumped Loudspeaker Driver*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver**

Interior Normal Velocity

The **Interior Normal Velocity** node adds a normal velocity on an interior boundary and ensures that the pressure is noncontinuous here. Add this node from the **Interior Conditions** submenu. The pressure has a so-called slit condition on this boundary. The condition is very similar to the Interior Normal Acceleration condition simply having the acceleration given by $i\omega$ times the velocity.

INTERIOR NORMAL VELOCITY

Select a **Type**: **Velocity** (the default) or **Inward velocity**.

- For **Velocity** enter values for the components of the **Velocity** $\mathbf{v}_0$ (SI unit: m/s^2).
- For **Inward velocity** enter the value of the **Inward velocity** v_n (SI unit: m/s^2) in the normal direction. Use a positive value for inward velocity or a negative value for outward velocity. The normal of the boundary points outward. The normal is depicted as a red arrow in the graphics window on the selected boundary.

Interior Normal Displacement

The **Interior Normal Displacement** node adds a normal displacement on an interior boundary and ensures that the pressure is noncontinuous here. Add this node from the **Interior Conditions** submenu. The pressure has a so-called slit condition on this boundary. The condition is very similar to the Interior Normal Acceleration condition simply having the acceleration given by $(i\omega)^2$ times the displacement.

INTERIOR NORMAL DISPLACEMENT

Select a **Type**: **Displacement** (the default) or **Inward displacement**.

- For **Displacement** enter values for the components of the **Displacement** $\mathbf{d}_0$ (SI unit: m/s^2).

- For **Inward displacement** enter the value of the **Inward displacement** d_n (SI unit: m/s^2) in the normal direction. Use a positive value for inward displacement or a negative value for outward displacement. The normal of the boundary points outward. The normal is depicted as a red arrow in the graphics window on the selected boundary.

Interior Impedance/Pair Impedance

The **Interior Impedance** and **Pair Impedance** nodes add an impedance boundary condition on interior boundaries or boundaries between the parts of pairs. Add the **Interior Impedance** node from the **Interior Conditions** submenu. This condition is a generalization of the sound-hard and sound-soft boundary conditions. The condition corresponds to a transfer impedance condition, relating the pressure drop across the boundary $p_{t,up} - p_{t,down}$ to the velocity at the boundary. In the frequency domain, it imposes the following equations:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right)_{up} = (p_{t,up} - p_{t,down})\frac{-i\omega}{Z_i}$$

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right)_{down} = (p_{t,up} - p_{t,down})\frac{-i\omega}{Z_i}$$

For a Time Dependent study (time domain), the boundary condition uses the following equations:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d)\right)_{up} = \frac{1}{Z_i}\frac{\partial}{\partial t}(p_{t,up} - p_{t,down})$$

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d)\right)_{down} = \frac{1}{Z_i}\frac{\partial}{\partial t}(p_{t,up} - p_{t,down})$$

On pair (interior) impedance conditions the up/down, that refers to the up and down side of an interior boundary, is replaced by a 1 and 2 index in the equation display. These refer to the two faces in the pair.

Z_i is the transfer impedance, which from a physical point is the ratio between pressure drop and normal particle velocity.

In the two opposite limits $Z_i \to \infty$ and $Z_i \to 0$, this boundary condition is identical to the Sound Hard boundary condition and the Sound Soft boundary condition, respectively. Additional information is found in Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual*.

INTERIOR IMPEDANCE/PAIR IMPEDANCE

Enter the value of the **Impedance** Z_i (SI unit: Pa·s/m). The default is 0 Pa·s/m.

Interior Perforated Plate/Pair Perforated Plate

The **Interior Perforated Plate** and **Pair Perforated Plate** nodes provide the possibility of specifying the characteristic properties for a perforated plate. Add the **Interior Perforated Plate** node from the **Interior Conditions** submenu.

COMSOL Multiphysics has three **Model types** for calculating the transfer impedance of a perforated plate. The following options are available:

- **Thin plate** (the default), the model where the losses due to heat conduction are negligible.
- **Thick plate**, the model that takes the thermal effects into account.
- **Asymptotic (legacy) model**, the model available in COMSOL version 5.2a and earlier.

A detailed description of the implemented models can be found in the Theory for the Interior Impedance Models section. The model properties are divided into two groups as shown below.

INTERIOR PERFORATED PLATE/PAIR PERFORATED PLATE

- **Hole diameter** d_h (SI unit: m). The default is 1 mm (10^{-3} m).
- **Plate thickness** t_p (SI unit: m). The default is 1.5 mm ($1.5 \cdot 10^{-3}$m).
- **Area porosity** σ, that is, the holes' fraction of the boundary surface area, a dimensionless number between 0 and 1. The default is 0.1, which means 10% of the plate's area consists of holes.
- **End correction** to the resistance δ_{resist} and the reactance δ_{reactt} (SI unit: m). The default built-in sets $\delta_{resist} = \delta_{resist} = 4d_h/3\pi$. Otherwise, select the user defined.

- **Hole-hole interaction** f_{int}, a dimensionless function that accounts for the influence of the porosity on the end correction. The default is the built-in Fok function (using eight terms):

$$f_{\text{int}} = \sum_{n=0}^{8} a_n(\sqrt{\sigma})^n .$$

- **Discharge coefficient (linear)** $C_{\text{D}}^{(\text{lin})}$, that is, a dimensionless coefficient related to the rate of the real flow through a hole to the theoretical flow. The default is 1.

Enable the **Include nonlinear effects** check box to include the nonlinear contribution to the resistance at high sound pressure levels. The following parameters become available:

- **Scaling factor** f_{nl}, a dimensionless. Default is 1.
- **Discharge coefficient (nonlinear)** $C_{\text{D}}^{(\text{nl})}$, which has the same meaning as $C_{\text{D}}^{(\text{lin})}$. The default is 0.76.

Enable **User-defined contribution** check box to, for example, include the effects of a mean flow on the impedance. The following parameters become available:

- **User-defined resistance** $\theta^{(\text{user})}$, a contribution to the resistive part of the impedance, dimensionless. The default is 0.
- **User-defined resistance** $\chi^{(\text{user})}$, a contribution to the reactive part of the impedance, dimensionless. The default is 0.

FLUID PROPERTIES

Select the **Fluid material** from **Boundary material** (the default) or select a material from the list. If **Boundary material** is selected, you need to add a material to the given boundary under the **Materials** node.

Select the material properties either **From material** (the default) or **User defined**:

- **Density** ρ (SI unit: kg/m^3).
- **Speed of sound** c (SI unit: m/s).
- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Ratio of specific heats** γ (SI unit: 1). Only for the **Thick plate** model.
- **Heat capacity at constant pressure** C_{p} (SI unit: J/(kg·K)). Only for the **Thick plate** model.
- **Thermal conductivity** k (SI unit: W/(m·K)). Only for the **Thick plate** model.

	The transfer impedance models from the list above are only valid for the perforates with round holes. For other types of perforates — with squared or slit-shaped holes — the values of some parameters can considerably differ from the of the suggested built in ones. It is recommended that you use the Interior Impedance/Pair Impedance condition to enter a user-defined model in such a case. The user-defined impedance can be obtained from a thermoviscous acoustic sub-model (see The Thermoviscous Acoustics, Frequency Domain Interface) as demonstrated in the Acoustic Muffler with Thermoviscous Acoustic Impedance Lumping model.

Line Source

Use the **Line Source** node to add a source on a line/edge in 3D components. This type of source corresponds a radially vibrating cylinder in the limit where its radius tends to zero. The line source adds a source term to the right-hand side of the governing Helmholtz equation such that:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2 p_t}{\rho_c} = \frac{4\pi}{\rho_c} S\delta(\mathbf{x} - \mathbf{x}_0)dl$$

where $\delta(\mathbf{x} - \mathbf{x}_0)$ is the delta function in 3D that adds the source on the edge where $\mathbf{x} = \mathbf{x}_0$ and dl is the line element along the edge (SI unit: m). The monopole amplitude S (SI unit: N/m^2) depends on the source type selected, as discussed below.

	• Frequency Domain Study • Solution (data sets) in the *COMSOL Multiphysics Reference Manual*

	For the Pressure Acoustics, Transient interface, only the **Flow** (no phase specification), **User defined,** and the **Gaussian pulse** source types are available. The **Gaussian pulse** source type has no effect in the frequency domain. See The Gaussian Pulse Source Type Settings.

LINE SOURCE

Select a **Type**: **Flow** (the default), **Intensity**, **Power**, or **User defined**.

For **User defined** enter a **Monopole amplitude**, $S = S_{\text{user}}$ (SI unit: N/m^2). Otherwise, enter details as follows.

Flow

Select **Flow** to add an edge source located at $\mathbf{r} = \mathbf{r}_0$ defined in terms of the volume flow rate per unit length out from source Q_S and the phase ϕ of the source. The flow edge source defines the following monopole amplitude:

$$S = e^{i\phi} \frac{i\omega\rho_c}{4\pi} Q_S$$

A flow edge source with the strength Q_S represents an area flow out from the source (the source is a very thin cylinder with a surface that pulsates).

- Enter a **Volume flow rate per unit length out from source**, Q_S (SI unit: m^2/s) for the source-strength amplitude in the field.
- Enter a **Phase** ϕ (SI unit: rad).

When defining a **Solution** data set and plotting the results, specify a nonzero phase ϕ to produce a nonzero result when visualizing the resulting pressure field using the default value (0) in the **Solution at angle (phase)**.

Intensity

Select **Intensity** to add an edge source located at $\mathbf{r} = \mathbf{r}_0$ defined in terms of the source intensity radiated I_{rms} and the phase ϕ of the source. Set a desired free space reference intensity (RMS) I_{rms} at a specified distance d_{src} from the source. In a homogeneous medium, the specified intensity is obtained when the edge is a straight line (this is the reference). With other objects and boundaries present, or if the edge is curved, the actual radiated intensity is different. This source type defines the following monopole amplitude:

$$S = e^{i\phi} \frac{d_{\text{src}}}{L_{\text{edge}}} \sqrt{2 I_{\text{ref}} \rho_c c_c}$$

where L_{edge} is the length of the source line (automatically determined), d_{src} is the distance from the source where free space reference intensity (RMS) I_{rms} is specified. Enter values or expressions for:

- **Free space reference intensity (RMS)**, I_{rms} (SI unit: W/m^2).
- **Distance from source center** d_{src} (SI unit: m).
- **Phase** ϕ (SI unit: rad).

Power

Select **Power** to add an edge source located at $\mathbf{r} = \mathbf{r}_0$ specified in terms of the source's reference RMS strength by stating the total power P_{rms} a straight line source would radiate into a homogeneous medium. This source type defines the following monopole amplitude:

$$S = \frac{e^{i\phi}}{L_{\text{edge}}}\sqrt{\frac{\rho_c c_c P_{\text{rms}}}{2\pi}}$$

where L_{edge} is the length of the source line (automatically determined) and P_{rms} denotes the free space reference RMS power (in the reference homogeneous case) per unit length measured in W/m. Enter values or expressions for:

- **Free space reference power (RMS)**, P_{rms} (SI unit: W).
- **Phase** ϕ (SI unit: rad).

Line Source on Axis

Use the **Line Source on Axis** node to add a line source on the axis of symmetry in 2D axisymmetric components. This type of source corresponds a radially vibrating cylinder in the limit where its radius tends to zero. The line source adds a source term to the right-hand side of the governing Helmholtz equation such that:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2 p_t}{\rho_c} = \frac{4\pi}{\rho_c} S\delta(z - z_0)dz$$

where $\delta(z - z_0)$ is the delta function in 3D that adds the source on the axis of symmetry where $z = z_0$ and $r = 0$, and dz is the line element along the z-axis

(SI unit: m). The monopole amplitude S (SI unit: $\mathrm{N/m^2}$) depends on the source type selected and is the same as discussed in the 3D case for a Line Source.

For the Pressure Acoustics, Transient interface, only the **Flow** (no phase specification), **User defined,** and the **Gaussian pulse** source types are available. The **Gaussian pulse** source type has no effect in the frequency domain. See The Gaussian Pulse Source Type Settings.

LINE SOURCE ON AXIS

Select a **Type**: **Flow** (the default), **Intensity**, **Power**, or **User defined**. See the options and expression for Line Source. The sources are the same but in 2D axisymmetric components they are only applicable on the symmetry axis at $r = 0$.

- Frequency Domain Study
- Solution (data sets) in the *COMSOL Multiphysics Reference Manual*

Monopole Point Source

Use the **Monopole Point Source** node to add a monopole point source in 3D components on any point and in 2D axisymmetric components on points on the axis of symmetry. This is a source that is uniform and equally strong in all directions. A monopole represents a radially pulsating sphere in the limit where the radius tends to zero. The monopole point source adds a point source term to the right-hand side of the governing Helmholtz equation such that:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_\mathrm{t} - \mathbf{q}_\mathrm{d})\right) - \frac{k_{\mathrm{eq}}^2 p_\mathrm{t}}{\rho_c} = \frac{4\pi}{\rho_c} S\delta(\mathbf{x} - \mathbf{x}_0)$$

where $\delta(\mathbf{x} - \mathbf{x}_0)$ is the delta function in three dimensions and adds the source at the point where $\mathbf{x} = \mathbf{x}_0$. The monopole amplitude S (SI unit: $\mathrm{N/m^2}$) depends on the source type selected, as discussed below.

For the Pressure Acoustics, Transient interface, only the **Flow** (no phase specification), **User defined,** and the **Gaussian pulse** source types are available. The **Gaussian pulse** source type has no effect in the frequency domain. See The Gaussian Pulse Source Type Settings.

MONOPOLE POINT SOURCE

Select a **Type**: **Flow** (the default), **Intensity**, **Power**, or **User defined**.

For **User defined** enter a **Monopole amplitude**, $S = S_{\text{user}}$ (SI unit: N/m).

Flow

Select **Flow** to add an monopole point source located at $\mathbf{x} = \mathbf{x}_0$ defined in terms of the volume flow rate out from source Q_S and the phase ϕ of the source. The source defines the following monopole amplitude:

$$S = e^{i\phi}\frac{i\omega\rho_c}{4\pi}Q_S$$

Enter values or expressions for:

- **Volume flow rate out from source**, Q_S (SI unit: m^3/s).
- **Phase** ϕ (SI unit: rad).

Intensity

Select **Intensity** to define the source in terms of the free space reference RMS intensity I_{rms} it radiates. In a homogeneous medium the specified intensity is obtained (the reference), but with other objects and boundaries present the actual intensity is different. The source defines the following monopole amplitude:

$$S = e^{i\phi}d_{\text{src}}\sqrt{2\rho_c c_c I_{\text{rms}}}$$

where d_{src} is the distance from the source where the intensity I_{rms} is specified and ϕ is the phase of the source. Enter values or expressions for:

- **Free space reference intensity (RMS)**, I_{rms} (SI unit: W/m^2).
- **Distance from source center** d_{src} (SI unit: m).
- **Phase** ϕ (SI unit: rad).

Power

When **Power** is selected, specify the source's reference RMS strength by stating the power it radiates. In a homogeneous medium the specified power is obtained (the reference), but with other objects and boundaries present the actual power is different. The source defines the following monopole amplitude:

$$S = e^{i\phi}\sqrt{\frac{\rho_c c_c P_{\text{rms}}}{2\pi}}$$

where P_{ref} denotes the radiated RMS power per unit length measured in W/m. Enter values or expressions for:

- **Free space reference power (RMS)**, P_{rms} (SI unit: W).
- **Phase** ϕ (SI unit: rad).

	*Bessel Panel*: Application Library path **Acoustics_Module/Tutorials/bessel_panel** *Hollow Cylinder*: Application Library path **Acoustics_Module/Vibrations_and_FSI/hollow_cylinder**

	Point sources in 2D components also exist. They actually correspond to infinite out-of-plane line sources, see Point Sources for 2D Components.

Dipole Point Source

Use the **Dipole Point Source** node to add a dipole point source. A dipole is mathematically a source that corresponds to two monopoles close to each other that are completely out of phase. Dipoles appear when there are fluctuating forces in the medium, for example, a small object that vibrates back and forth. A complex acoustic source may be expanded and approximated by a collection of point sources (Monopole Point Source, Dipole Point Source, and Quadrupole Point Source). The dipole point source adds a point source term to the right-hand side of the governing Helmholtz equation such that:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2 p_t}{\rho_c} = \frac{4\pi}{\rho_c}\mathbf{D} \cdot \nabla\delta(\mathbf{x} - \mathbf{x}_0)$$

where $\delta(\mathbf{x} - \mathbf{x}_0)$ is the delta function in three dimensions and adds the source at the point where $\mathbf{x} = \mathbf{x}_0$. The dipole moment vector $\mathbf{D}$ (SI unit: N) depends on the source type selected, as discussed below. In 2D axisymmetric models the dipole point source is only added to the z-axis, such that $\mathbf{x}_0 = (0,0,z)$, see Ref. 5 for details.

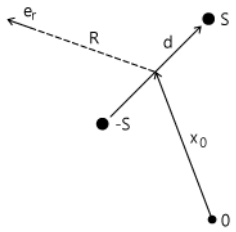

Figure 2-1: Schematic illustration of a dipole source.

DIPOLE POINT SOURCE

Select a **Type**: **Power** (the default) or **User defined**.

User Defined

For **User defined** enter a **Dipole moment vector** $\mathbf{D}$ (SI unit: N). In 2D axisymmetric components enter the z-component only.

Power

The **Power** option defines the following dipole moment vector in terms of the free space reference power, the dipole direction, and the source phase

$$\mathbf{D} = |\mathbf{D}|e^{i\phi}\frac{\mathbf{e}_\mathrm{D}}{|\mathbf{e}_\mathrm{D}|} \qquad |\mathbf{D}|^2 = \frac{3\rho_\mathrm{c}c_\mathrm{c}^3 P_\mathrm{rms}}{2\pi\omega^2}$$

Enter the following:

- **Free space reference power (RMS)**, P_rms (SI unit: W). In a homogeneous medium the specified power is radiated (the reference), but with other objects and boundaries present the actual power is different.
- The source **Phase** ϕ (SI unit: rad).
- **Dipole moment direction** $\mathbf{e}_\mathrm{D}$. Defines the direction of the dipole moment vector. In 2D axisymmetric components the direction is aligned along the z-axis such that $\mathbf{e}_\mathrm{D} = \mathbf{e}_\mathrm{z}$.

Point sources in 2D components also exist. They actually correspond to infinite out-of-plane line sources, see Point Sources for 2D Components.

Quadrupole Point Source

Use the **Quadrupole Point Source** node to enter a quadrupole point source. A quadrupole is mathematically a source that corresponds to two dipoles close to each other. A complex acoustic source may be expanded and approximated by a collection of point sources (Monopole Point Source, Dipole Point Source, and Quadrupole Point Source). The quadrupole point source adds a point source term to the right-hand side of the governing Helmholtz equation such that:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2 p_t}{\rho_c} = \frac{4\pi}{\rho_c}(\mathbf{D} \cdot \nabla)(\mathbf{d} \cdot \nabla)\delta(\mathbf{x} - \mathbf{x}_0)$$

where $\delta(\mathbf{x} - \mathbf{x}_0)$ is the delta function in three dimensions and adds the source at the point where $\mathbf{x} = \mathbf{x}_0$. The quadrupole D-vector $\mathbf{D}$ (SI unit: N) and the quadrupole d-vector $\mathbf{d}$ (SI unit: m) depend on the source type selected, as discussed below. In 2D axisymmetric models the quadrupole point source is only added to the z-axis, such that $\mathbf{x}_0 = (0,0,z)$ and only orientations along the axis are possible. Two types of predefined quadrupole sources exist, a **Power (longitudinal)** and a **Power (lateral)** configuration, respectively. In the longitudinal source the $\mathbf{D}$ and $\mathbf{d}$ vectors are parallel and point in the same direction. In the lateral configuration the $\mathbf{D}$ and $\mathbf{d}$ vectors are perpendicular to each other, see Ref. 5 for details.

QUADRUPOLE POINT SOURCE

Select a **Type**: **Power (longitudinal)** (the default), **Power (lateral)**, or **User defined**. The lateral options does not exist in 2D-axisymmetric models.

User Defined

For **User defined** enter the **Quadrupole d vector** (SI unit: m) and **Quadrupole D vector** (SI unit: N). In 2D axisymmetric models enter a **Quadrupole amplitude** Q (SI unit: N·m) as both the $\mathbf{D}$ and $\mathbf{d}$ vectors are pointed along the z-axis.

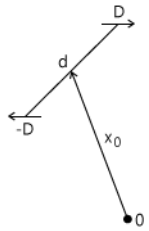

Figure 2-2: Schematic illustration of the user defined quadrupole point source.

Power (longitudinal)

The **Power (longitudinal)** defines the following values for the two quadrupole vectors

$$\mathbf{e}_\mathrm{d} = \mathbf{e}_\mathrm{D} \qquad Q = \mathbf{d} \cdot \mathbf{D}^\mathrm{T} = Qe^{i\phi}(\mathbf{e}_\mathrm{d} \cdot \mathbf{e}_\mathrm{D})$$

$$Q^2 = \frac{5\rho_\mathrm{c} c_\mathrm{c} P_\mathrm{rms}}{2\pi k^4}$$

Enter the following:

- **Free space reference power (RMS)**, P_rms (SI unit: W). In a homogeneous medium the specified power is radiated (the reference), but with other objects and boundaries present the actual power is different.
- The source **Phase** ϕ (SI unit: rad).
- **Quadrupole direction,** $\mathbf{e}'_\mathrm{D}$ (dimensionless). In a 2D axisymmetric model the direction is along the z-axis.

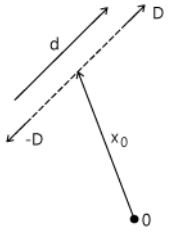

Figure 2-3: Schematic illustration of the longitudinal power quadrupole point source.

Power (lateral)

The **Power (lateral)**, not defined in 2D axisymmetric components, defines the following values for the two quadrupole vectors

$$\mathbf{e}_\mathrm{d} = \mathbf{n} \times \mathbf{e}_\mathrm{D} \qquad Q = \mathbf{d} \cdot \mathbf{D}^\mathrm{T} = Qe^{i\phi}(\mathbf{e}_\mathrm{d} \cdot \mathbf{e}_\mathrm{D})$$

$$Q^2 = \frac{15\rho_\mathrm{c} c_\mathrm{c} P_\mathrm{rms}}{2\pi k^4}$$

Enter the following:

- **Free space reference power (RMS)**, P_{rms} (SI unit: W). In a homogeneous medium the specified power is radiated (the reference), but with other objects and boundaries present the actual power is different.
- The source **Phase** ϕ (SI unit: rad).
- **Quadrupole normal**, $\mathbf{n}$ (dimensionless). This is the normal to the plane in which the quadrupole is located.
- **Quadrupole direction,** $\mathbf{e}'_{\mathrm{D}}$ (dimensionless). The projection of this vector onto the plane (defined by the normal $\mathbf{n}$) defines the direction of the dipole D-vector.

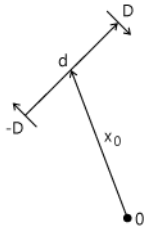

Figure 2-4: Schematic illustration of lateral quadrupole point source.

Point sources in 2D components also exist. They actually correspond to infinite out-of-plane line sources, see Point Sources for 2D Components.

Point Sources for 2D Components

For 2D components the monopole, dipole, and quadrupole point sources also exist. All the sources actually correspond to an infinite line source in the out-of-plane direction. The monopole is uniform and equally strong in all the in-plane directions — an infinitely long pulsating cylinder. The dipole source represents an infinitely long cylinder moving back and forth. The quadrupole represents two dipoles close to each other.

The point sources adds the following point source term to the right-hand side of the governing Helmholtz equation — monopole, dipole, and quadrupole, respectively:

$$\frac{4\pi}{\rho_c} S\delta(\mathbf{x}-\mathbf{x}_0) \qquad \frac{4\pi}{\rho_c}\mathbf{D}\cdot\nabla\delta(\mathbf{x}-\mathbf{x}_0) \qquad \frac{4\pi}{\rho_c}(\mathbf{D}\cdot\nabla)(\mathbf{d}\cdot\nabla)\delta(\mathbf{x}-\mathbf{x}_0)$$

where $\delta(\mathbf{x}-\mathbf{x}_0)$ is the delta function in 2D and adds the source at the point where $(x,y) = \mathbf{x} = \mathbf{x}_0$. The monopole amplitude S (SI unit: $\mathrm{N/m^2}$), the dipole moment vector $\mathbf{D}$ (SI unit: N/m), and the quadrupole $\mathbf{d}$ vector (SI unit: m) and $\mathbf{D}$ vectors (SI unit: N/m) depend on the source type selected.

MONOPOLE POINT SOURCE (2D COMPONENTS)

Select a **Type**: **Flow** (the default), **Intensity**, **Power**, or **User defined**. For **User defined** enter a **Monopole amplitude**, S_{user} (SI unit: N/m).

Flow

Select **Flow** to add an monopole point source located at $\mathbf{r} = \mathbf{r}_0$ defined in terms of the volume flow rate per unit length out from source Q_{S} and the phase ϕ of the source. The source defines the following monopole amplitude:

$$S = e^{i\phi}\frac{i\omega\rho_{\mathrm{c}}}{4\pi}Q_{\mathrm{S}}$$

Enter values or expressions for:

- **Volume flow rate out from source**, Q_{S} (SI unit: $\mathrm{m^3/s}$).
- **Phase** ϕ (SI unit: rad).

Intensity

Select **Intensity** to define the source in terms of the free space reference RMS intensity I_{rms} it radiates. In a homogeneous medium the specified intensity is obtained (the reference), but with other objects and boundaries present the actual intensity is different. The source defines the following monopole amplitude:

$$S = e^{i\phi}\sqrt{\frac{2\rho_{\mathrm{c}}\omega I_{\mathrm{rms}}d_{\mathrm{src}}}{2\pi}}$$

where d_{src} is the distance from the source where the intensity I_{rms} is specified and ϕ is the phase of the source. Enter values or expressions for:

- **Free space reference intensity (RMS)**, I_{rms} (SI unit: $\mathrm{W/m^2}$).
- **Distance from source center** d_{src} (SI unit: m).
- **Phase** ϕ (SI unit: rad).

Power

When **Power** is selected, specify the source's reference RMS strength by stating the power per unit length it radiates. In a homogeneous medium the specified power is

obtained (the reference), but with other objects and boundaries present the actual source power is different. The source defines the following monopole amplitude:

$$S = e^{i\phi}\sqrt{\frac{2\rho_c \omega P_{\mathrm{rms}}}{(2\pi)^2}}$$

where P_{rms} denotes the free space RMS reference power per unit length measured in W/m and ϕ is the phase of the source. Enter values or expressions for:

- **Free space reference power (RMS) per unit length**, P_{rms} (SI unit: W/m).
- **Phase** ϕ (SI unit: rad).

DIPOLE POINT SOURCE (2D COMPONENTS)

Enter a user defined **Dipole-moment vector,** D (SI unit: N/m).

QUADRUPOLE POINT SOURCE (2D COMPONENTS)

Enter a user defined **Quadrupole d vector**, $\mathbf{d}$ (SI unit: m), and a user defined **Quadrupole D vector**, $\mathbf{D}$ (SI unit: N/m).

Circular Source (for 2D axisymmetric components)

Use the **Circular Source** node to add a source in a 2D axisymmetric components on points off the axis of symmetry. Such points correspond to circular sources or ring sources. This type of source is, for example, used to mimic source terms from rotors. The circular source adds a point source term to the right-hand side of the governing Helmholtz equation such that:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2 p_t}{\rho_c} = \frac{4\pi}{\rho_c} S\delta(\mathbf{x}-\mathbf{x}_0) r d\varphi$$

where $\delta(\mathbf{x}-\mathbf{x}_0)$ is the delta function that adds the source at the point where $\mathbf{x}=\mathbf{x}_0$ and $rd\varphi$ is the line element around the circular source (SI unit: m). The monopole amplitude S (SI unit: N/m^2) depends on the source type selected, as discussed below

For the Pressure Acoustics, Transient interface the **Flow** (no phase specification), **User defined,** and the **Gaussian pulse** source types are available. The **Gaussian pulse** source type has no effect in the frequency domain. See The Gaussian Pulse Source Type Settings.

CIRCULAR SOURCE

Select a **Type**: **Flow** (the default) or **User defined**. For **User defined** enter a **Monopole amplitude**, $S = S_{\text{user}}$ (SI unit: N/m^2).

Flow

When **Flow** is selected the source is defined in terms of the volume flow rate Q_{S} per unit length out form the source. The flow circular source defines the following monopole amplitude:

$$S = e^{i\phi}\,\frac{i\omega\rho_{\text{c}}}{4\pi}Q_{\text{S}}$$

Enter values or expressions for:

- **Volume flow rate per unit length out from source**, Q_{S} (SI unit: m^2/s).
- **Phase** ϕ (SI unit: rad).

The Pressure Acoustics, Transient Interface

The **Pressure Acoustics, Transient (actd)** interface ( ), found under the **Acoustics>Pressure Acoustics** branch ( ) when adding a physics interface, is used to compute the pressure variation when modeling the propagation of acoustic waves in fluids at quiescent background conditions. It is suited for time-dependent simulations with arbitrary time-dependent fields and sources.

The physics interface can be used to model linear and nonlinear acoustics that can be well described by the scalar pressure variable. Domain conditions also include background incident acoustic fields. User-defined sources can be added to, for example, include certain nonlinear effects such as a square pressure dependency of the density variations. For open problems Perfectly Matched Layers (PMLs) can be applied, also in the time domain for Pressure Acoustics, as efficient non reflecting boundary conditions.

The physics interface solves the scalar wave equation in the time domain. Studies for performing time-dependent modal and modal reduced-order models also exist. The physics interface also solves in the frequency domain with the available boundary conditions.

For good modeling strategies, meshing, solvers, postprocessing information, acoustics specific plots, as well as tips and tricks, see the Modeling with the Pressure Acoustics Branch section.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Transient Pressure Acoustics Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and source. You can also right-click **Pressure Acoustics, Transient** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern

<name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is actd.

TRANSIENT SOLVER SETTINGS

Select the **Time stepping** (method) as **Manual** (default and recommended) or **Automatic/free** and then enter the **Maximum frequency to resolve** in the model. The default frequency is set to 1000[Hz] but should be changed to reflect the frequency content of the sources used in the model. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study. The generated settings will not be adequate for highly nonlinear problems. In this case manual tuning needs to be done.

Further details on Time Stepping in Transient Models in the Modeling with the Pressure Acoustics Branch section.

The remainder of the **Settings** window is shared with The Pressure Acoustics, Frequency Domain Interface.

- Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface
- Theory Background for the Pressure Acoustics Branch
- Theory for the Perfectly Matched Layers in the Time Domain

- *Transient Gaussian Explosion*: Application Library path **Acoustics_Module/Tutorials/gaussian_explosion**
- *Gaussian Pulse Absorption by Perfectly Matched Layers: Pressure Acoustics, Transient*: Application Library path **Acoustics_Module/Tutorials/gaussian_pulse_perfectly_matched_layers**

Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface

The Pressure Acoustics, Transient Interface shares most of its nodes with the Pressure Acoustics, Frequency Domain interface, except the following which differ:

- Transient Pressure Acoustics Model
- The Gaussian Pulse Source Type Settings
- Normal Acceleration
- Normal Velocity
- Normal Displacement
- Interior Normal Acceleration
- Interior Normal Velocity
- Interior Normal Displacement

Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface

Transient Pressure Acoustics Model

The **Transient Pressure Acoustics Model** node adds the equations for primarily time-dependent (transient) acoustics modeling. This is the scalar wave equation

$$\frac{1}{\rho c^2}\frac{\partial^2 p_t}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d)\right) = Q_m$$

where p_t is the total acoustic pressure, ρ is the fluid density, c is the speed of sound, $\mathbf{q}_d$ is the Dipole Domain Source, and Q_m is the Monopole Domain Source. In this formulation of the wave equation the speed of sound and density may in general be space dependent but only slowly varying in time, that is, at a time scale much slower than the variations in the acoustic signal.

In the **Settings** window, define the properties for the acoustics model and model inputs including temperature.

See Pressure Acoustics for details of the fluid model equations.

Select a **Fluid model**: **Linear elastic** (the default), **Viscous**, **Thermally conducting**, **Thermally conducting or viscous**, or **Ideal Gas**. Then see the descriptions for The Pressure Acoustics, Frequency Domain Interface:

- Defining a Linear Elastic Fluid Model
- Defining a Viscous Fluid Model
- Defining a Thermally Conducting Fluid Model
- Defining a Thermally Conducting and Viscous Fluid Model
- Defining an Ideal Gas Fluid Model

MODEL INPUTS (IDEAL GAS ONLY)

If **Ideal gas** is selected as the **Fluid model**, enter a **Temperature** T (which can be a constant temperature or a temperature field from a Heat Transfer interface) and an **Absolute pressure** p_A:

- Select **User defined** to enter a value or an expression for the absolute pressure (SI unit: Pa) in the field that appears. This input is always available.
- In addition, select an absolute pressure defined by a Fluid Flow interface present in the model (if any). For example, select **Absolute pressure (spf)** to use the absolute pressure defined by a Laminar Flow interface **spf**. This allows the use of a system-based (gauge) pressure, while automatically including the reference pressure in the absolute pressure.

The Gaussian Pulse Source Type Settings

For the Pressure Acoustics, Transient interface, you can select a **Gaussian pulse** as a source **Type** for the Line Source, Line Source on Axis, Monopole Point Source, Point Sources for 2D Components, and Circular Source (for 2D axisymmetric components) features. This type adds a source with a Gaussian time profile defined in terms of its amplitude A, its frequency bandwidth f_0, and the pulse peak time t_p. Using this source type results in solving a wave equation of the type:

$$\frac{1}{\rho c^2}\frac{\partial^2 p_t}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d)\right) = \frac{4\pi}{\rho} S\delta(\mathbf{x} - \mathbf{x}_0)$$

$$S = \frac{\rho}{4\pi}\frac{\partial}{\partial t}[Ae^{-\pi^2 f_0^2 (t - t_p)^2}] = -A\rho\frac{\pi}{2} f_0^2 (t - t_p) e^{-\pi^2 f_0^2 (t - t_p)^2}$$

where S is the source strength.

In 3D components (for Line Source), in 2D axisymmetric components (for Line Source on Axis and Circular Source (for 2D axisymmetric components)), and 2D components (for Point Sources for 2D Components), all of which are effectively line sources, enter the following values or expressions:

- The value of the pulse **Amplitude** A (SI unit: m^2/s).
- **Frequency bandwidth** f_0 (SI unit: Hz).
- **Pulse peak time** t_p (SI unit: s) for the duration of the pulse.

In 3D and 2D axisymmetric components for the Monopole Point Source, enter the following values or expressions:

- The value of the pulse **Amplitude** A (SI unit: m^3/s).
- **Frequency bandwidth** f_0 (SI unit: Hz).
- **Pulse peak time** t_p (SI unit: s) for the duration of the pulse.

Transient Gaussian Explosion: Application Library path **Acoustics_Module/Tutorials/gaussian_explosion**

Normal Acceleration

The **Normal Acceleration** adds an inward normal acceleration $a_n(t)$. Mathematically the condition adds the boundary condition:

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

Alternatively, specify the acceleration $\mathbf{a}_0$ of the boundary. The part in the normal direction is used to define the boundary condition:

$$-\mathbf{n} \cdot \left(-\frac{1}{\rho}(\nabla p - \mathbf{q}_d)\right) = -\mathbf{n} \cdot \mathbf{a}_0(t)$$

This feature represents an external source term. It can also be used to manually couple acoustics with a structural analysis for modeling acoustic-structure interaction.

NORMAL ACCELERATION

Select a **Type**: **Inward Acceleration** (the default) or **Acceleration**.

- For **Inward Acceleration** enter the value of the **Inward acceleration** a_n (SI unit: m/s^2). Use a positive value for inward acceleration or a negative value for outward acceleration.
- For **Acceleration** enter values for the components of the **Acceleration** $\mathbf{a}_0$ (SI unit: m/s^2).

In transient models all sources need to be functions of time `t` in order to represent an acoustic signal. This can, for example, be a harmonic sine wave defined as `sin(omega*t)` or any other signal.

Normal Velocity

The **Normal Velocity** adds an inward normal velocity $v_n(t)$ or specify the acceleration $\mathbf{v}_0(t)$ of the boundary. The part in the normal direction is used to define the boundary condition. The condition is very similar to the Normal Acceleration condition having the acceleration given by the time derivative of the velocity $a_n(t) = \partial v_n(t)/\partial t$. This feature represents an external source term. It can also be used to manually couple acoustics with a structural analysis for modeling acoustic-structure interaction.

NORMAL VELOCITY

Select a **Type**: **Inward Velocity** (the default) or **Velocity**.

- For **Inward Velocity** enter the value of the **Inward velocity** v_n (SI unit: m/s). Use a positive value for inward velocity or a negative value for outward velocity.
- For **Velocity** enter values for the components of the **Velocity** $\mathbf{v}_0$ (SI unit: m/s).

Normal Displacement

The **Normal Displacement** adds an inward normal displacement $d_n(t)$ or specify the acceleration $\mathbf{d}_0(t)$ of the boundary. The part in the normal direction is used to define the boundary condition. The condition is very similar to the Normal Acceleration condition having the acceleration given by the double time derivative of the displacement $a_n(t) = \partial^2 d_n(t)/\partial t^2$. This feature represents an external source term. It

can also be used to manually couple acoustics with a structural analysis for modeling acoustic-structure interaction.

NORMAL DISPLACEMENT

Select a **Type**: **Inward Displacement** (the default) or **Displacement**.

- For **Inward Displacement** enter the value of the **Inward displacement** d_n (SI unit: m). Use a positive value for inward displacement or a negative value for outward displacement.
- For **Displacement** enter values for the components of the **Displacement** $\mathbf{d}_0$ (SI unit: m).

Interior Normal Acceleration

The **Interior Normal Acceleration** node adds a normal acceleration on an interior boundary and ensures that the pressure is noncontinuous here. Add this node from the **Interior Conditions** submenu. The pressure has a so-called slit condition on this boundary. This boundary condition can be used to model sources as, for example, the movement of a speaker cone modeled as a boundary. The condition adds the normal part of an acceleration $\mathbf{a}_0(t)$:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{up}=\mathbf{n}\cdot\mathbf{a}_0(t)\qquad -\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{down}=\mathbf{n}\cdot\mathbf{a}_0(t)$$

Alternatively, specify the inward acceleration $a_n(t)$. The normal of the boundary is interpreted as pointing outward.

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{up}=a_n(t)\qquad -\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{down}=a_n(t)$$

INTERIOR NORMAL ACCELERATION

Select a **Type**: **Acceleration** (the default) or **Inward acceleration**.

- For **Acceleration** enter values for the components of the **Acceleration** $\mathbf{a}_0$ (SI unit: m/s^2).
- For **Inward acceleration** enter the value of the **Inward acceleration** a_n (SI unit: m/s^2) in the normal direction. Use a positive value for inward acceleration or a negative

value for outward acceleration. The normal of the boundary points outward. The normal is depicted as a red arrow in the graphics window on the selected boundary.

In transient models all sources need to be functions of time `t` in order to represent an acoustic signal. This can, for example, be a harmonic sine wave defined as `sin(omega*t)` or any other signal.

Interior Normal Velocity

The **Interior Normal Velocity** node adds a normal velocity on an interior boundary and ensures that the pressure is noncontinuous here. Add this node from the **Interior Conditions** submenu. The pressure has a so-called slit condition on this boundary. The condition is very similar to the Interior Normal Acceleration condition having the acceleration given by the time derivative of the velocity.

INTERIOR NORMAL VELOCITY

Select a **Type**: **Velocity** (the default) or **Inward velocity**.

- For **Velocity** enter values for the components of the **Velocity** $\mathbf{v}_0$ (SI unit: $\mathrm{m/s^2}$).
- For **Inward velocity** enter the value of the **Inward velocity** v_n (SI unit: $\mathrm{m/s^2}$) in the normal direction. Use a positive value for inward velocity or a negative value for outward velocity. The normal of the boundary points outward. The normal is depicted as a red arrow in the graphics window on the selected boundary.

Interior Normal Displacement

The **Interior Normal Displacement** node adds a normal displacement on an interior boundary and ensures that the pressure is noncontinuous here. Add this node from the **Interior Conditions** submenu. The pressure has a so-called slit condition on this boundary. The condition is very similar to the Interior Normal Acceleration condition having the acceleration given by the double time derivative of the displacement.

INTERIOR NORMAL DISPLACEMENT

Select a **Type**: **Displacement** (the default) or **Inward displacement**.

- For **Displacement** enter values for the components of the **Displacement** $\mathbf{d}_0$ (SI unit: $\mathrm{m/s^2}$).

For **Inward displacement** enter the value of the **Inward displacement** d_n (SI unit: $\mathrm{m/s^2}$) in the normal direction. Use a positive value for inward displacement or a negative

value for outward displacement. The normal of the boundary points outward. The normal is depicted as a red arrow in the graphics window on the selected boundary.

The Pressure Acoustics, Boundary Mode Interface

The **Pressure Acoustics, Boundary Mode (acbm)** interface ( ), found under the **Acoustics>Pressure Acoustics** branch ( ) when adding a physics interface, is used to compute and identify propagating and nonpropagating modes in waveguides and ducts by performing a boundary mode analysis on a given boundary. The study is useful, for example, when specifying sources at inlets or analyzing transverse acoustic modes in ducts. It is available for 3D and 2D axisymmetric component models.

The physics interface solves the Helmholtz eigenvalue equation on boundaries, searching for the out-of-plane wave numbers at a given frequency.

When this physics interface is added, these default nodes are also added to the **Model Builder**: **Pressure Acoustics Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Pressure Acoustics, Boundary Mode** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `acbm`.

EQUATION

For 2D axisymmetric components, the **Circumferential mode number** m is by default 0. It is an integer entering the axisymmetric expression for the pressure:

$$p(r,z,\varphi) = p(r)\,e^{-i(k_z z + m\varphi)}$$

Change the value as needed. Also see Physics Nodes — Equation Section in the *COMSOL Multiphysics Reference Manual*.

SOUND PRESSURE LEVEL SETTINGS

See the settings for Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface.

DEPENDENT VARIABLES

This physics interface defines one dependent variable (field), the **Pressure** p. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**.

- Initial Values
- Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Boundary Mode Interface
- Pressure Acoustics, Boundary Mode Equations

Initial Values

The **Initial Values** node adds initial values for the sound pressure. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the **Pressure** p (SI unit: Pa) initial value.

Special postprocessing variables exist for the Boundary Mode Acoustics interface. They are described in Pressure Acoustics, Boundary Mode Variables.

Eigenmodes in a Muffler: Application Library path **Acoustics_Module/Automotive/eigenmodes_in_muffler**

Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Boundary Mode Interface

Except for Initial Values, The Pressure Acoustics, Boundary Mode Interface shares all of its feature nodes with the Pressure Acoustics, Frequency Domain interface. See Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface.

Also, for the Pressure Acoustics, Boundary Mode interface, apply the features to boundaries instead of domains for 3D components.

Modeling with the Pressure Acoustics Branch

In this section:

Meshing (Resolving the Waves)

Solutions to acoustic problems are wavelike. The waves are characterized by a wavelength λ in space, whose value depends on the frequency and speed of sound c in the medium according to $\lambda = c/f$. This wavelength has to be resolved by the mesh.

To represent a wave on a discrete grid (the mesh), it is obvious that the mesh elements must be smaller than the wavelength in order to resolve the wave. That is, there needs to be several *degrees of freedom* (DOFs) per wavelength in the direction of propagation. In reality, the lower limit for a fully reliable solution lies at about ten to twelve degrees of freedom per wavelength.

Because the direction of propagation is generally not known beforehand, it is good practice to aim for an isotropic mesh with about twelve DOFs per wavelength on average, independently of the direction. Therefore the number of DOFs in a sufficiently resolved mesh is about:

- $1728 = 12^3$ times the model volume measured in wavelengths cubed in 3D.

- $144 = 12^2$ times the model area measured in wavelengths squared in 2D.
- 12 times the model length measured in wavelengths in 1D.

Before starting a new model, try to estimate the required number of DOFs using these guidelines. The maximum number of DOFs that can be solved for differs between computer systems. See Solving Large Acoustics Problems Using Iterative Solvers for solver suggestions.

When creating a mesh set the maximum element size $h_{\max}$ to about $\lambda/5$ or smaller. This will create a good initial mesh with the required 12 DOFs per wavelength. The mesh should also resolve important geometric features. Always check the sensitivity of your solution on the mesh size. Make sure the solution does not change significantly when refining the mesh.

USING LAGRANGE AND SERENDIPITY ELEMENTS

When creating an unstructured mesh for use with the default 2nd-order Lagrange elements or 2nd-order serendipity elements (see Lagrange and Serendipity Shape Functions), set the maximum element size $h_{\max}$ to about $\lambda/5$ or smaller. Because all elements in the constructed mesh are smaller than $h_{\max}$, the limit is set larger than the actual required element size. The mesh should also resolve important geometric features and possible gradients in the material parameters and model inputs. After meshing the model, check the total number of DOFs against the model volume and the above guidelines. If the mesh turns out, on average, to be too coarse or too fine, try to change $h_{\max}$ accordingly.

Unstructured meshes are generally better than structured meshes for wave problems where the direction of wave propagation is not known everywhere in advance. The reason is that in a structured mesh, the average resolution typically differs significantly between directions parallel to the grid lines and directions rotated 45 degrees about one of the axes.

Meshing in the *COMSOL Multiphysics Reference Manual*

MESHING THE PERFECTLY MATCHED LAYER (PML)

When using a perfectly matched layer (PML) to truncate the computational domain it is good practice to use a structured mesh inside the PML region. In 3D models use a **Swept** mesh inside the PML and in 2D models use a **Mapped** mesh. Use at least 5 elements in the thickness when using rational PML scaling and 8 elements when using the default polynomial scaling in the PML. Again make sure the check for mesh convergence by adding more layers.

When creating the geometry for your model use the **Layers** option to create the geometry of your PML layer/domain. This will ensure that it is suited for proper meshing using a structured mesh.

	Acoustic Scattering off an Ellipsoid: Application Library path **Acoustics_Module/Tutorials/acoustic_scattering**

	Structured Meshes in the *COMSOL Multiphysics Reference Manual*

Lagrange and Serendipity Shape Functions

In most of the physics interfaces in the Acoustics Module and specifically in the Pressure Acoustics interfaces, you can choose between two families of shape functions: *Lagrange* and *serendipity*. The current default is to use Lagrange shape functions. To display the **Discretization** section, click the **Show** button ( ) and select **Discretization**.

When using a structured mesh it may be advantageous to switch to the serendipity elements as they generate significantly fewer degrees of freedom (DOFs). The accuracy is in most cases almost as good as for the Lagrange elements. The Lagrange elements are however less sensitive to strong mesh distortions.

The serendipity shape functions differs from the Lagrange shape functions only for the following element shapes:

- 2D: Quadrilateral elements of discretization order higher than 1.
- 3D: Hexahedral, prism, and pyramid elements of discretization order higher than 1.

In the *COMSOL Multiphysics Reference Manual*:

- The Lagrange Element
- The Nodal Serendipity Element

When coupling two physics interfaces that have the same DOFs like, for example, displacement, the same type of shape functions should be used in both interfaces to ensure conformity. Since there is no difference between the two families of shape functions in 1D, this is not an issue when connecting edges.

Time Stepping in Transient Models

When solving transient models, first decide on the maximal frequency you want to resolve, say $f_{\max}$. This frequency translates to a minimal wavelength $\lambda_{\min} = c/f_{\max}$ and in turn to a maximum element size $h_{\max} < \lambda_{\min}/5$ as discussed in Meshing (Resolving the Waves).

The value of this maximum frequency should also be entered into the **Maximum frequency to resolve** field in the Transient Solver Settings section at the top physics level. Here it is also possible to select the **Time stepping** (method) as either **Manual** or **Automatic/free**. It is recommended to use the **Manual** method as this method is best suited for wave propagation problems. Using these settings the generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model, see Meshing (Resolving the Waves). The auto generated suggestion is good for all linear and weakly nonlinear problems. If the model studied exhibits high nonlinearities the solver may need manual set up and tuned.

The logic for the automatic choice made is as follows. The mesh resolution imposes a restriction on the time-step size Δt taken by the solver. The relationship between mesh size and time-step size is closely related to the CFL number (Ref. 33), which is defined as

$$\mathrm{CFL} = \frac{c\Delta t}{h}$$

where c is the speed of sound and h is the mesh size. This nondimensional number can be interpreted as the fraction of an element the wave travels in a single time step. A CFL number around 1 would correspond to the same resolution in space and time if the discretization errors were of the same size; however, that is normally not the case.

By default, COMSOL Multiphysics uses the implicit second-order accurate method generalized-α to solve transient acoustics problems. In space, the default is 2nd-order Lagrange elements. Generalized-α introduces some numerical damping of high frequencies but much less than the BDF method.

Time Dependent in the *COMSOL Multiphysics Reference Manual*

The temporal discretization errors for generalized-α are larger than the spatial discretization errors when 2nd-order elements are used in space. The limiting step size, where the errors are of roughly the same size, can be found somewhere at CFL < 0.2. You can get away with a longer time step if the forcing does not make full use of the mesh resolution; that is, if high frequencies are absent from the outset.

When the excitation contains all the frequencies the mesh can resolve, there is no point in using an automatic time-step control which can be provided by the time-dependent solver. The tolerances in the automatic error control are difficult to tune when there is weak but important high-frequency content. Instead, you can use your knowledge of the typical mesh size, speed of sound, and CFL number to calculate and prescribe a fixed time step. This is exactly the default behavior when the **Manual** method is chosen in the **Transient Solver Settings** section. The **Automatic/free** option corresponds to the automatic time-step control but with some tighter controls of the allowed time-steps. This latter option is still not recommended as the manual option typically yields much better results (and is faster).

The internal time step generated by the **Manual** option and the entered **Maximal frequency to resolve** is set by assuming that the user has generated a mesh that properly resolves the same maximal frequency (minimal wavelength). The following step is generated

$$\Delta t = \frac{h_{\max}\mathrm{CFL}}{c} = \frac{\mathrm{CFL}}{f_{\max}N} \approx \frac{1}{60f_{\max}}$$

Assuming that N is between 5 and 6 and the CFL number is roughly 0.1. These values give a good margin of safety. To check that the accuracy is acceptable, it is

recommended that you run a short sequence of typical excitations with progressively smaller time steps (larger $f_{\max}$)and check the convergence.

	*Transient Gaussian Explosion*: Application Library path **Acoustics_Module/Tutorials/gaussian_explosion**

	When several physics are present in a model the logic for the time step is to use the minimal time step dictated by the contributing physics. Some physics, like electrostatics, suggest a BDF time stepping method to be used. If this physics is present BDF will be used but the maximal time steps will be taken from the acoustics suggestions.

Frequency-Domain Modal and AWE

When solving models in the frequency domain and performing a large frequency sweep it can sometimes be useful to use the **Frequency-Domain Modal** study or the **Asymptotic waveform evaluation** (AWE) option in the normal **Frequency Domain** study. Both approaches can significantly speed up the solution time when sweeping over many frequencies.

For the **Frequency-Domain Modal** consider:

- The modal solver requires that the eigenfrequencies and modes can be easily found for up to twice the maximal study frequency. If a model contains many resonances in the desired frequency range, this may not be a good idea.
- Note that since the modal solver assumes linear perturbation all sources should be declared using the `linper()` operator.

For the **Asymptotic waveform evaluation** consider:

- When using the AWE option under **Study Extensions** in the **Frequency Domain** study, it is necessary to enter an **AWE expression**. This is an expression that represents the response of the system. It needs to be a global expression. This can for example be

the sound pressure level (SPL) evaluated using an integration or average operator. Integration in a point will give the SPL in that point.

Both solvers are used and discussed in the tutorial: *Helmholtz Resonator Analyzed with Different Frequency Domain Solvers*: Application Library path **Acoustics_Module/Tutorials/helmholtz_resonator_solvers**

See also the sections Frequency Domain Study and Frequency Domain Modal and Time-Dependent Modal Studies in the Acoustics Module Study Types section.

Solving Large Acoustics Problems Using Iterative Solvers

This section has some guidance for solving large acoustics problems. For smaller problems using a direct solver like MUMPS is often the best choice. For larger problems, especially in 3D, the only option is often to use an iterative method such as multigrid.

In large models with structured mesh you can often save DOFs by changing the default Lagrange elements to serendipity elements. For more information see Lagrange and Serendipity Shape Functions.

AUTOMATICALLY GENERATED SUGGESTIONS

If the direct solver runs out of memory a simple first approach is to enable and use one of the auto generated iterative solver suggestions. A good starting point for this is to right-click the study node and select **Show Default Solver**, then expand the **Solver Configuration** tree under **Stationary Solver** or **Time-Dependent Solver**. Predefined iterative solver suggestions are automatically generated. Per default a direct solver is used and two iterative solvers are suggested and disabled (grayed out). To turn on one of these right-click the solver and select **Enable** (or press F4). The first suggestion *(GMRES with GMG)* uses the GMRES iterative solver with a geometric multigrid (GMG) preconditioner. This method is typically faster than the direct solver and uses less memory for large 3D models. The second suggestion *(FGMRES with GMG)* uses the FGMRES iterative solver with a geometric multigrid (GMG) preconditioner. This method is more robust than the GMRES especially for problems that exhibit sharp

resonances. If the GMRES suggestion does not converge try the FGMRES suggestion instead. Both suggestions along with more details are described below.

For an example that solves a pressure acoustics model using an iterative solver see: *Test Bench Car Interior*: Application Library path **Acoustics_Module/Automotive/test_bench_car_interior**

If PMLs are present in the model solved with an iterative method it is recommended to use the **Polynomial** scaling option (the default) and the recommended 8 mesh layers. This option will ensure proper convergence of the iterative methods. See the Perfectly Matched Layers (PMLs) section for further details.

MANUAL SUGGESTIONS AND THEORY

The underlying equation for many of the problems within acoustics is the Helmholtz equation. For high frequencies (or wave numbers) the matrix resulting from a finite-element discretization becomes highly indefinite. In such situations, it can be problematic to use geometric multigrid (GMG) with simple smoothers such as Jacobi or SOR (the default smoother). Fortunately, there exist robust and memory-efficient approaches that circumvent many of the difficulties associated with solving the Helmholtz equation using geometric multigrid.

When using a geometric multigrid as a linear system solver together with simple smoothers, the Nyquist criterion must be fulfilled on the coarsest mesh. If the Nyquist criterion is not satisfied, the geometric multigrid solver might not converge. One way to get around this problem is to use GMRES or FGMRES as a linear system solver with geometric multigrid as a preconditioner.

As a good starting point for modifying the solver select **Show Default Solver** on the main study node and expand the tree. Go to the **Stationary Solver** and add an **Iterative** solver node, per default it uses the GMRES method. The default preconditioner is the incomplete LU (see the sub-node to the Iterative node), right-click the **Iterative** solver node and select **Multigrid**. Even if the Nyquist criterion is not fulfilled for the coarse meshes of the multigrid preconditioner, such a scheme is more likely to converge. For problems with high frequencies this approach might, however, lead to a large number of iterations. Then it might be advantageous to use either:

- Geometric multigrid as a linear system solver (set the **Solver** selection to **Use preconditioner**) with GMRES as a smoother. Under the **Multigrid** node right-click

the **Presmoother** and **Postsmoother** nodes and select the **Krylov Preconditioner** with the **Solver** selection to **GMRES**.

- FGMRES as a linear system solver (set the **Solver** selection to **FGMRES**) with geometric multigrid as a preconditioner (where GMRES is used as a smoother, as above).

Using GMRES or FGMRES as an outer iteration and smoother removes the requirements on the coarsest mesh. When GMRES is used as a smoother for the multigrid preconditioner, FGMRES must be used for the outer iterations because such a preconditioner is not constant (see Ref. 34).

Use GMRES as a smoother only if necessary because GMRES smoothing is very time- and memory-consuming on fine meshes, especially for many smoothing steps.

When solving large acoustics problems, the following options, in increasing order of robustness and memory requirements, can be of use:

- If the Nyquist criterion is fulfilled on the coarsest mesh, try to use geometric multigrid as a linear system solver (set **Multigrid** as preconditioner and set the linear system solver to **Use preconditioner**) with default smoothers. The default smoothers are fast and have small memory requirements.
- An option more robust than the first point is to use GMRES as a linear system solver with geometric multigrid as a preconditioner (where default SOR smoothers are used). GMRES requires memory for storing search vectors. This option can sometimes be used successfully even when the Nyquist criterion is not fulfilled on coarser meshes. Because GMRES is not used as a smoother, this option might find a solution faster than the next two options even if a large number of outer iterations are needed for convergence.
- If the above suggestion does not work, try to use geometric multigrid as a linear system solver with GMRES as a smoother.
- If the solver still has problems converging, try to use FGMRES as a linear system solver with geometric multigrid as a preconditioner (where GMRES is used as a smoother).

- Try to use as many multigrid levels as needed to produce a coarse mesh for which a direct method can solve the problem without using a substantial amount of memory.
- If the coarse mesh is still too fine for a direct solver, try using an iterative solver with 5–10 iterations as coarse solver.

Studies and Solvers and Multigrid in the *COMSOL Multiphysics Reference Manual*

Perfectly Matched Layers (PMLs)

The perfectly matched layer (PML) is a domain or layer (sometimes called sponge layer) that is added to an acoustic model to mimic an open and nonreflecting infinite domain. It sets up a perfectly absorbing domain as an alternative to nonreflecting boundary conditions. The PML works with all types of waves, not only plane waves. It is also efficient at very oblique angles of incidence. In the frequency domain the PML imposes a complex-valued coordinate transformation to the selected domain that effectively makes it absorbing at a maintained wave impedance, and thus eliminating reflections at the interface. In the time domain additional equitations are solved in the PML for the inverse Laplace transformed equations.

A **Perfectly Matched Layers** node is added to the model from the **Model>Definitions** node. In the frequency domain the PMLs can be used for the Pressure Acoustics, Acoustic-Structure Interaction, Aeroacoustics, and Thermoviscous Acoustics interfaces. In the time domain the PMLs only exist for the Pressure Acoustics, Transient interface.

In this section:

- Meshing the PMLs
- Coordinate Stretching, Scaling, and Curvature
- Geometry Type (User Defined)
- Limitations of the Perfectly Matched Layers
- Time Domain Perfectly Matched Layers

MESHING THE PMLS

Optimal behavior of the PML is achieved when the mesh inside the PML domain is structured. Use a mapped mesh in 2D models and a swept mesh in 3D models. Use at

least 8 layers when using the default polynomial stretching option. As a good starting point for the rational stretching use 5 or 6 mesh layers inside the PML. When creating the geometry for your model, it is also advantageous to use the layer feature to create the PML domains. This ensures that the geometry is suited for a structured mesh.

The following is a good tutorial model on the use of perfectly matched layers (PMLs): *Acoustic Scattering off an Ellipsoid*: Application Library path **Acoustics_Module/Tutorials/acoustic_scattering**

COORDINATE STRETCHING, SCALING, AND CURVATURE

The choice of the **Coordinate stretching type** and the **PML scaling factor** and the **PML curvature parameter** depends on the problem at hand. A detailed description is given in the PML Implementation section of the *COMSOL Multiphysics Reference Manual*. In general, the **Rational** stretching option is used for open radiation problems for propagating waves (it is efficient for many angles of incidence). The **Polynomial** stretching option is good for systems with a mix of different wave types (propagating and evanescent), for example, for terminating a waveguide or in multiphysics problems. Note that when solving a model using an iterative solver the **Polynomial** scaling should be used to ensure convergence.

There is also a **User Defined** coordinate stretching type which allow users to define advanced stretching functions to handle special cases. The stretching can in this way be optimized to a special problem.

To ensure that the PML is working optimally, it is good practice to make a mesh convergence test by refining (or most probably adding more layers to) the mesh in the PML domain. This is especially important at low frequencies, where evanescent waves may interact with the PML and give erroneous solutions.

The behavior of the PMLs at low frequencies is discussed in the following model. Plotting the total radiated power can be a good indicator of possible issues. *Lumped Loudspeaker Driver*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver**

Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*.

The PMLs damp a certain wavelength existing in the system. The wavelength is deducted from the frequency and a reference wave speed c_{ref}. The wave speed is defined in the Typical Wave Speed section. Set c_{ref} equal to the speed of sound of the material in the PML.

When multiple physics and materials are involved, be cautious. Depending on the configuration, use several PMLs and PML domains or a single PML. For details, see Configuration of Perfectly Matched Layers (PMLs) for Acoustic-Structure Interaction Models.

GEOMETRY TYPE (USER DEFINED)

When setting up a PML, you also select the geometry type of the layer. Typically, the predefined options **Cartesian**, **Cylindrical**, or **Spherical** can apply in most situations. Using these COMSOL will automatically detect the layer thickness and define the local coordinates inside the PML. In some cases the automatic detection can fail (this can, for example, happen for certain imported CAD geometries). The automatic detection also fails if the domain is not the outer most entity in the geometry. A workaround is then to use the **User defined** geometry type. This advanced option makes is possible to define the local *Distance functions* and layer *Thickness* manually. For example, for a spherical PML geometry the typical distance function is `sqrt(x^2+y^2+z^2)-r0`, where `r0` is the radius of the inner domain. The user-defined option can also be used for spacial layer shapes.

Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*.

LIMITATIONS OF THE PERFECTLY MATCHED LAYERS

When a model contains a Background Pressure Field and PMLs, certain configurations will create incompatibilities that lead to erroneous behavior. The problem arises if a domain with a background pressure field is next to a domain without the feature (for example when setting up absorption problems) and the two domains have a common PML attached to them. Meaning that the PML next to the background pressure field touches the PML next to the domain without the background pressure field. In this case, there is an incompatibility at the common edge of the PMLs. In one PML domain the pressure DOF is interpreted as a scattered field, while it is the total field in

the other. Note that you can set up models that contain this feature configuration as long as the PMLs do not touch.

TIME DOMAIN PERFECTLY MATCHED LAYERS

In the time domain the PML does not include a real stretching component. This means that the geometric thickness needs to be set adequately. Typically the layer should have a thickness which is between $\lambda/8$ and $\lambda/4$, where λ is the lowest wavelength in the signal. The PML also needs to be places roughly $\lambda/4$ away from the sources.

When meshing the PMLs for time domain simulations use a structured mesh in the same way as in the frequency domain. Use at least 8 mesh layers for the rational scaling and 6 for the polynomial scaling.

See Theory for the Perfectly Matched Layers in the Time Domain for details about the implementation.

Postprocessing Variables

Several specialized variables specific to acoustics are predefined in the Acoustics Module and can be used when analyzing the results of an acoustic simulation. The variables are available from the expression selection menus when plotting.

In this section:

- Intensity Variables
- Power Dissipation Variables
- Pressure Acoustics, Boundary Mode Variables

In the *COMSOL Multiphysics Reference Manual:*

- Results Analysis and Plots
- Operators, Functions, and Constants

INTENSITY VARIABLES

The propagation of an acoustic wave is associated with a flow of energy in the direction of the wave motion, the intensity vector $\mathbf{I}$. The sound intensity in a specific direction

(through a specific boundary) is defined as the time average of (sound) power per unit area in the direction of the normal to that area.

Knowledge of the intensity is important when characterizing the strength of a sound source — that is, the power emitted by the source. The power is given by the integral of $\mathbf{n}\cdot\mathbf{I}$ on a surface surrounding the source, where $\mathbf{n}$ is the surface normal. The intensity is also important when characterizing transmission phenomena, for example, when determining transmission loss or insertion loss curves.

The acoustic intensity vector $\mathbf{I}$ (SI unit: $\mathrm{W/m^2}$) is defined as the time average of the instantaneous rate of energy transfer per unit area (sound power) $p\mathbf{u}$, such that

$$\mathbf{I} = \frac{1}{T}\int_0^T p\mathbf{u}\,dt$$

where p is the pressure and $\mathbf{u}$ the particle velocity. In the frequency domain (harmonic time dependence), the integral reduces to

$$\mathbf{I} = \frac{1}{2}Re(p\mathbf{u}^*) = \frac{1}{4}(p\mathbf{u}^* + p^*\mathbf{u}) \qquad (2\text{-}1)$$

where * denotes complex conjugation. In the frequency domain the velocity is readily expressed in terms of the pressure as $\mathbf{u} = -1/(i\omega\rho)\nabla p$. Using the characteristic specific impedance for plane waves, the intensity can also be expressed in terms of the RMS pressure as

$$\mathbf{I} = \frac{1pp^*}{2\ \rho c}$$

In the Pressure Acoustics, Frequency Domain interface the more general formulation given in Equation 2-1 is used to define the intensity.

For time-dependent problems, the equivalent quantity is the instantaneous value of the intensity, defined as

$$\mathbf{i}_{\mathrm{inst}} = p\mathbf{u}$$

This expression is difficult to recover in Pressure Acoustics and would require the solution of an additional PDE to calculate the velocity from the pressure dependent variable. Only the intensity $\mathbf{I}$ (time averaged) is available as results and analysis variable in the frequency domain and can be selected from the expressions menus when plotting. The instantaneous intensity does exists as a postprocessing variable in

transient interfaces such as Linearized Euler and Linearized Navier-Stokes where the velocity is solved for explicitly, see Modeling with the Aeroacoustics Branch for details.

The variables are defined in Table 2-3, Table 2-4, and Table 2-5. In the variable names, *phys_id* represents the interface name, for example, acpr for a Pressure Acoustics, Frequency Domain interface.

TABLE 2-3: INTENSITY VARIABLES IN 3D

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ix	x-component of the intensity vector (frequency domain only)
phys_id.Iy	y-component of the intensity vector (frequency domain only)
phys_id.Iz	z-component of the intensity vector (frequency domain only)

TABLE 2-4: INTENSITY VARIABLES IN 2D AXISYMMETRIC

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ir	r-component of the intensity vector (frequency domain only)
phys_id.Iz	z-component of the intensity vector (frequency domain only)

TABLE 2-5: INTENSITY VARIABLES IN 2D

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ix	x-component of the intensity vector (frequency domain only)
phys_id.Iy	y-component of the intensity vector (frequency domain only)

In the *COMSOL Multiphysics Reference Manual:*

- Results Analysis and Plots
- Expressions and Predefined Quantities

POWER DISSIPATION VARIABLES

Common to all the Pressure Acoustics fluid models (also porous materials) and The Thermoviscous Acoustics, Frequency Domain Interface is that all the interfaces model some energy dissipation process, which stem from viscous and thermal dissipation processes. The amount of dissipated energy can be of interest as a results analysis variable or as a source term for a multiphysics problem. An example could be to

determine the amount of heating in the human tissue when using ultrasound. In the Acoustics Module special variables exist for the dissipation.

- For the case of a plane wave propagating in the bulk of a fluid (the general thermal and viscous fluid models described in Thermally Conducting and Viscous Fluid Model) the dissipation is

$$\Delta = \frac{1}{(\rho c^2)^2}\left[\mu\left(\frac{4}{3}+\frac{\mu_b}{\mu}\right)+\frac{k(\gamma-1)}{C_p}\right]\left(\frac{\partial p}{\partial t}\right)^2$$

  and in the frequency domain after averaging over one period

$$\Delta = \frac{1}{(\rho c^2)^2}\left[\mu\left(\frac{4}{3}+\frac{\mu_b}{\mu}\right)+\frac{k(\gamma-1)}{C_p}\right]\frac{\omega^2}{2}pp^* \tag{2-2}$$

  where * in Equation 2-2 is the complex conjugate operator.

- In addition, an approximate expression for the dissipated energy density from a propagating plane wave exists for the Narrow Region Acoustics, the Poroacoustics models and attenuation in Pressure Acoustics. This total dissipated power density Q_{pw} is defined by

$$Q_{\mathrm{pw}} = -2|\mathbf{I}|\mathrm{imag}(k)$$

  where $|\mathbf{I}|$ is the magnitude of the intensity vector $\mathbf{I}$, and k is the wave number. This expression is an approximation and is only valid for traveling plane waves (or waves that are close to plane); however, it has many uses as a first estimate of the dissipation since it is easy to calculate in many different situations. The expression is, for example, not valid for standing waves in resonant systems. When the above expression is not valid, the dissipated energy should be calculated using an energy balance approach.

The power dissipation variables are defined in Table 2-6. In the variable names, *phys_id* represents the name (acpr, for example, for a pressure acoustics interface).

TABLE 2-6: POWER DISSIPATION VARIABLES

VARIABLE	DESCRIPTION
phys_id.diss_therm	Thermal power dissipation density
phys_id.diss_visc	Viscous power dissipation density
phys_id.diss_tot	Total thermal and viscous power dissipation density
phys_id.Q_pw	Plane-wave total dissipated power density

PRESSURE ACOUSTICS, BOUNDARY MODE VARIABLES

A series of special variables exist for postprocessing after solving a boundary mode acoustics problem. They include in-plane and out-of-plane components of the velocity **v** and acceleration **a**.

The in-plane (ip) and out-of-plane (op) components to the acceleration and velocity are defined as

$$\begin{aligned}\mathbf{a}_{\text{ip}} &= \mathbf{a} - (\mathbf{a}\cdot\mathbf{n})\mathbf{n}\\ \mathbf{v}_{\text{ip}} &= \mathbf{v} - (\mathbf{v}\cdot\mathbf{n})\mathbf{n}\\ \mathbf{a}_{\text{op}} &= (\mathbf{a}\cdot\mathbf{n})\mathbf{n}\\ \mathbf{v}_{\text{op}} &= (\mathbf{v}\cdot\mathbf{n})\mathbf{n}\end{aligned}$$

where **n** is the normal to the surface being modeled. The velocity and acceleration are defined in terms of the gradient of the pressure p as follows

$$\nabla p = \nabla_{||} p - ik_{\text{n}} p\mathbf{n} \qquad \text{in 3D}$$

$$\nabla p = \nabla_{||} p - ik_{\text{n}} p(n_r, 0, n_z) + p\left(0, -i\frac{m}{r}, 0\right) \qquad \text{in 2D axisymmetry}$$

and

$$\mathbf{v} = \frac{i}{\omega\rho}\nabla p \qquad \text{and} \qquad \mathbf{a} = i\omega\mathbf{v}$$

where k_{n} is the out-of-plane wave number solved for, m is a possible radial wave mode number, and $\nabla_{||}$ is the tangential derivative along the boundary.

The boundary mode acoustics variables are defined in Table 2-7. In the variable names, *phys_id* represents the name (`acbm`, for example, for a Boundary Mode Acoustics interface).

TABLE 2-7: BOUNDARY MODE ACOUSTICS VARIABLES IN 3D

VARIABLE	DESCRIPTION
phys_id.vipx	In-plane velocity, x-component
phys_id.vipy	In-plane velocity, y-component
phys_id.vipz	In-plane velocity, z-component
phys_id.vip_rms	In-plane velocity RMS value
phys_id.aipx	In-plane acceleration, x-component
phys_id.aipy	In-plane acceleration, y-component

TABLE 2-7: BOUNDARY MODE ACOUSTICS VARIABLES IN 3D

VARIABLE	DESCRIPTION
phys_id.aipz	In-plane acceleration, z-component
phys_id.aip_rms	In-plane acceleration RMS value
phys_id.vopx	Out-of-plane velocity, x-component
phys_id.vopy	Out-of-plane velocity, y-component
phys_id.vopz	Out-of-plane velocity, z-component
phys_id.vop_rms	Out-of-plane velocity RMS value
phys_id.aopx	Out-of-plane acceleration, x-component
phys_id.aopy	Out-of-plane acceleration, y-component
phys_id.aopz	Out-of-plane acceleration, z-component
phys_id.aop_rms	Out-of-plane acceleration RMS value

Evaluating the Acoustic Field in the Far-Field Region

The Acoustics Module has functionality to evaluate the acoustic pressure field in the far-field region outside of the computational domain. This is the Far-Field Calculation feature available for pressure acoustics problems. This section gives some general advice for analyzing the far field.

THE NEAR-FIELD AND FAR-FIELD REGIONS

The solution domain for a scattering or radiation problem can be divided into two zones, reflecting the behavior of the solution at various distances from objects and sources. In the *far-field* region, scattered or emitted waves are locally planar, velocity and pressure are in phase with each other, and the ratio between pressure and velocity approaches the free-space impedance of a plane wave.

Moving closer to the sources into the *near-field* region, pressure and velocity gradually slide out of phase. This means that the acoustic field contains energy that does not travel outward or radiate. These evanescent wave components are effectively trapped close to the source. Looking at the sound pressure level, local maxima and minima are apparent in the near-field region.

Naturally, the boundary between the near-field and far-field regions is not sharp. A general guideline is that the far-field region is that beyond the last local energy maximum, that is, the region where the pressure amplitude drops monotonously at a rate inversely proportional to the distance from any source or object R.

A similar definition of the far-field region is the region where the radiation pattern — the locations of local minima and maxima in space — is independent of the distance to the wave source. This is equivalent to the criterion for Fraunhofer diffraction in optics, which occurs for Fresnel numbers, $F = a^2/\lambda R$, much smaller than 1. For engineering purposes, this definition of the far-field region can be applied:

$$R > \frac{8a^2}{\lambda} = \frac{8}{2\pi} ka^2 \tag{2-3}$$

In Equation 2-3, a is the radius of a sphere enclosing all objects and sources, λ is the wavelength, and k is the wave number. Another way to write the expression leads to the useful observation that the size of the near-field region expressed in source-radius units is proportional to the dimensionless number ka, with a prefactor slightly larger than one. This relation is known as the Rayleigh radius $R_0 = S/\lambda$, where S is the source area, for example for a piston.

Knowing the extent of the near-field region is useful when applying radiation boundary conditions because these are accurate only in the far-field region. PMLs, on the other hand, can be used to truncate a domain already inside the near-field region.

THE HELMHOLTZ-KIRCHHOFF INTEGRAL REPRESENTATION

In many cases, solving the acoustic Helmholtz equation everywhere in the domain where results are requested is neither practical nor necessary. For homogeneous media, the solution anywhere outside a closed surface containing all sources and scatterers can be written as a boundary integral in terms of quantities evaluated on the surface. To evaluate this *Helmholtz-Kirchhoff* integral, it is necessary to know both Dirichlet and Neumann values on the surface. Applied to acoustics, this means that if the pressure *and* its normal derivative is known on a closed surface, the acoustic field can be calculated at any point outside, including amplitude and phase. This functionality is included in the Far-Field Calculation feature. The feature has two options for the evaluation, one full integral and one that only looks in the extreme far field. See the section Theory for the Far-Field Calculation: The Helmholtz-Kirchhoff Integral for further details.

FULL INTEGRAL

To evaluate the full Helmholtz-Kirchhoff integral use the **Full integral** option in the settings for the far-field variables. The full Helmholtz-Kirchhoff integral gives the pressure (including phase) at any point at a finite distance from the source surface, but the numerical integration tends to lose accuracy at large distances. See Far-Field Calculation.

THE FAR-FIELD LIMIT

In many applications, the quantity of interest is the far-field radiation pattern, which can be defined as the limit of $r|p|$ when r goes to infinity in a given direction. To evaluate the pressure in the far-field limit use the **Integral approximation at $r \to \infty$** option in the settings for the far-field variables See Far-Field Calculation.

The far field pressure is evaluated using the far-field operator (the name is defined in the **Far-field variable name** input field, the default is `pfar`).

To evaluate the pressure in a point `(x0,y0,z0)`, simply write `pfar(x0,y0,z0)`. To evaluate the sound pressure level in the same point, it is advantageous to use the `subst()` operator and write, for example, `subst(acpr.ffc1.Lp_pfar,x,x0,y,y0,z,z0)`.

An example of this is given in the *Loudspeaker Driver* model form the Acoustics Application Libraries.

Dedicated Acoustics Plots for Postprocessing

Dedicated acoustics postprocessing functionality exist to plot responses, transfer functions, directivity patterns and more. They require the Acoustics Module license.

In this section:

- Far-Field Plots (Spatial Response)
- Grid and Parametrized Data Sets
- Octave Band Plots
- Directivity Plots

FAR-FIELD PLOTS (SPATIAL RESPONSE)

Evaluating and plotting the acoustic pressure around radiating objects and sources is essential for the development of several acoustic devices. This is known as the spatial response. Application areas range from underwater acoustic transducers and loudspeakers, to determining the spatial sensitivity of microphone systems (for example, using reciprocity). Using the Far-Field Calculation feature can help to reduce the computational domain while still being able to determine the pressure and thus the spatial response in the near-field to far-field. The *Far Field plots* are specially designed for easy evaluation of the far-field variables, that is, the acoustics far-field pressure and the far-field sound pressure level.

In the **Far Field** plots the far-field variable (pressure or sound pressure level) is represented in a polar plot for a selected number of angles. Data is retrieved on an evaluation circle in 2D, 2D axial symmetry, or 3D. The angle interval and the number of angles can be manually specified. The evaluation circle origin, orientation, and radius can be specified as well as the reference direction. The evaluation circle can be visualized using a **Preview Evaluation Plane** functionality. There is also a built in option to calculate the **Beam Width** of the plotted data. A 3D polar plot also exists where the data is evaluated on a sphere. For 3D far-field plots you also specify an expression for the surface color.

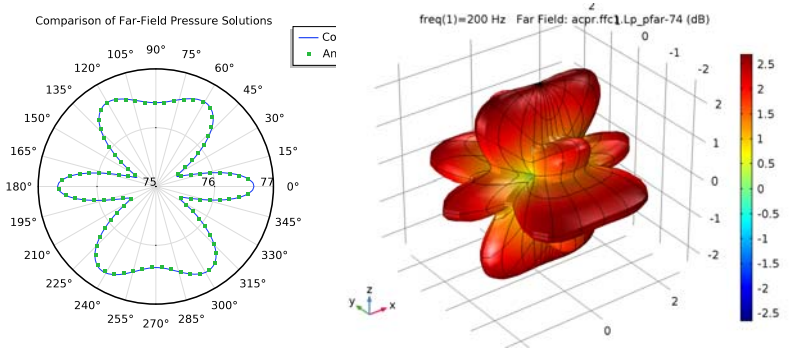

Figure 2-5: Example of a 2D and 3D far field plot from the Bessel Panel tutorial model.

The main advantage with the far-field plot, as compared to making a line graph, is that the unit circle/sphere that you use for defining the plot directions, is not part of your geometry for the solution. Thus, the number of plotting directions is decoupled from the discretization of the solution domain.

	Default **Far Field** plots are automatically added to any model that uses far-field calculations.

	• For a 3D example, see *Bessel Panel*: Application Library path **Acoustics_Module/Tutorials/bessel_panel**. • For a 2D axisymmetric example, see *Cylindrical Subwoofer*: Application Library path **Acoustics_Module/Tutorials/cylindrical_subwoofer**.

- Evaluating the Acoustic Field in the Far-Field Region
- Far Field and Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

GRID AND PARAMETRIZED DATA SETS

Another way of evaluating and depicting the far field is by using either the **Grid 2D**, the **Grid 3D**, the **Parametrized Curve**, or the **Parametrized Surface** data sets. Both types of data sets allow the evaluation of global quantities, like the far-field variables, outside of the computational domain (outside of the mesh).

The far field is depicted in several ways in the model *Piezoelectric Tonpilz Transducer*: Application Library path **Acoustics_Module/ Piezoelectric_Devices/tonpilz_transducer**.

OCTAVE BAND PLOTS

A dedicated **Octave Band** ( ) plot exists to plot frequency response, transfer functions, transmission loss, and insertion loss curves. The plot has several built in acoustics specific features like predefined weighting (Z, A, C, and user defined) as well as the possibility to plot the response in octaves, 1/3 octaves, or as a continuous response.

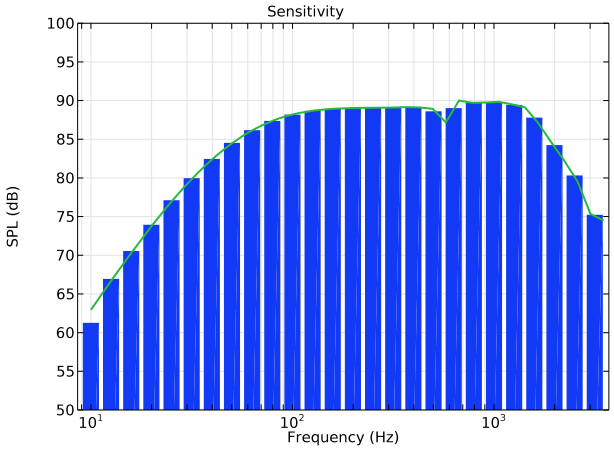

Figure 2-6: Sensitivity curve plotted as 1/3 octave bands and a continuous curve. From the Loudspeaker Driver model in the Application Library.

For details see Octave Band in the Results Analysis and Plots chapter in the *COMSOL Multiphysics Reference Manual*.

For examples that use an Octave Band plot, see:

- *Absorptive Muffler*: Application Library path **Acoustics_Module/Automotive/absorptive_muffler**
- *The Brüel & Kjær 4134 Condenser Microphone*: Application Library path **Acoustics_Module/Electroacoustic_Transducer/bk_4134_microphone**.

DIRECTIVITY PLOTS

Another acoustics specific plot, especially used for loudspeakers, is the dedicated **Directivity** ( ) plot. The plot allows audio engineers to depict the spatial response of a loudspeaker as function of both frequency and spatial angle in a contour like plot. Representing the spatial response in this manner is a very common the loudspeaker industry. Measured data is often also represented in the same manner. The plot includes many options to format the plot to achieve maximal insight into the modeled data, for example, easy switch of the x- and y-axis (frequency and polar angle axis) as

well as options to normalize the data with respect to a given angle or the maximal value.

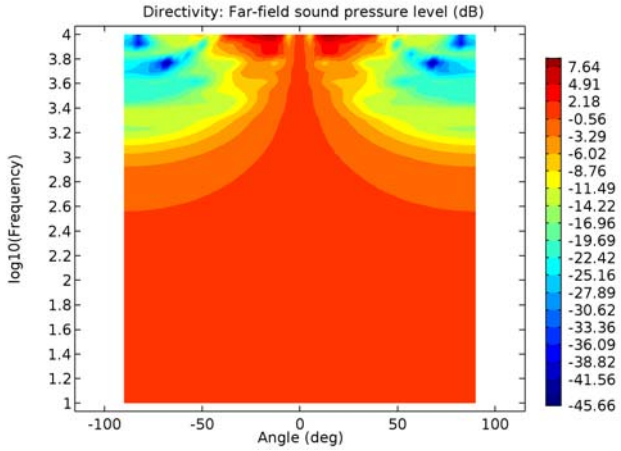

Figure 2-7: Example of the directivity plot from the Lumped Loudspeaker driver model in the Application Library

For details see Directivity in the Results Analysis and Plots chapter in the *COMSOL Multiphysics Reference Manual*.

Lumped Loudspeaker Driver. Application Library path: **Acoustics_Module/Electroacoustic_Transducer/lumped_loudspeaker_driver**

About the Material Databases for the Acoustics Module

The Acoustics Module includes two material databases: Liquids and Gases, with temperature-dependent fluid dynamic and thermal properties, and a Piezoelectric Materials database with over 20 common piezoelectric materials.

For detailed information about Materials, the Liquids and Gases Materials Database, and the Piezoelectric Materials Database see the *COMSOL Multiphysics Reference Manual*.

THE AIR MATERIAL

The built in Air material, located both in the Built-In and the Liquids and Gases library, is commonly used in applications and models. That material defines common material parameters and their dependency on the ambient pressure p_A and temperature T. The relations are simplified and not all thermodynamic dependencies are taken into account. The speed of sound c and the density ρ are defined through the ideal gas law (assuming adiabatic behavior) following

$$c^2 = \gamma RT \qquad \rho = \frac{p_A}{R_s T}$$

with the ratio of specific heats $\gamma = 1.4$ and the specific gas constant $R_s = 287$ J/(kg·K). This is an idealization of air valid in many cases, see Ref. 5. This means that the speed of sound is not a function of the ambient pressure for this build-in Air material. More detailed relations can of course be entered by defining a user defined material. The dynamic viscosity, the heat capacity at constant pressure, and the coefficient of thermal conduction are all given by polynomial fit curves and only include the dependency on the ambient temperature T.

Specifying Frequencies: Logarithmic and ISO Preferred

When solving a model in the frequency domain, you need to specify the **Frequencies** to solve for in the **Frequency Domain** study step. Several built in options exist when selecting the **Range** ( ). Specifically for acoustics modeling the **Entry method** has the **Logarithmic** and the **ISO preferred frequencies**. The latter option is only available with the Acoustics Module and is activated by clicking the **Show** button ( ) and select **Advanced Study Options**. For **ISO preferred frequencies**, select a start and stop frequency and the interval (Octave, 1/3 Octave, 1/6 Octave, 1/12 Octave, and 1/24 Octave). The 1/3 octave preferred frequencies are defined based on ISO 266. They are extended to frequencies based on the preferred numbers of ISO 3 (series R20, R40, and R80).

Theory Background for the Pressure Acoustics Branch

This section describes the governing equations and the mathematical formulation of the governing equations as used in the Pressure Acoustics branch of the Acoustics Module. Details are also given regarding some of the boundary conditions, including the radiation boundary conditions as well as the far-field calculation feature. The section starts with a general introduction to the governing equations used in pressure acoustics.

In this sections:

- The Governing Equations
- Pressure Acoustics, Frequency Domain Equations
- Pressure Acoustics, Transient Equations
- Pressure Acoustics, Boundary Mode Equations
- Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions
- Theory for the Far-Field Calculation: The Helmholtz-Kirchhoff Integral

The Governing Equations

Pressure acoustic problems involve solving for the small acoustic pressure variations p (also named p' or p_1) on top of the stationary background pressure p_0. Mathematically, this represents a linearization (small parameter expansion) of the dependent variables around the stationary quiescent values.

The governing equations for a compressible lossless (no thermal conduction and no viscosity) fluid flow problem are the mass conservation equation (continuity equation), the momentum conservation equation (Euler's equation), and the energy equation (entropy equation). These are given by:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = M$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \mathbf{F}$$

$$\frac{\partial s}{\partial t} + \nabla \cdot (s\mathbf{u}) = 0$$

where ρ is the total density, p is the total pressure, $\mathbf{u}$ is the velocity field, s is the entropy, M and $\mathbf{F}$ represent possible source terms. In classical pressure acoustics, all thermodynamic processes are assumed to be reversible and adiabatic, that is, isentropic processes. The small parameter expansion is performed on a stationary fluid ($\mathbf{u}_0 = \mathbf{0}$) of density ρ_0 (SI unit: kg/m^3) and at pressure p_0 (SI unit: Pa) such that:

$$\begin{aligned} p &= p_0 + p_1 \\ \rho &= \rho_0 + \rho_1 \\ \mathbf{u} &= \mathbf{0} + \mathbf{u}_1 \\ s &= s_0 + s_1 \end{aligned} \quad \text{with} \quad \begin{aligned} p_1 &\ll p_0 \\ \rho_1 &\ll \rho_0 \\ |\mathbf{u}_1| &\ll c \end{aligned}$$

where the variables with subscript 1 represent the small acoustic variations (the 1st order expansion). Assuming the initial entropy to be zero, then it will remain and $s_0 = 0$. Inserting these into the governing equations and only retaining terms linear in the acoustic variables yields

$$\begin{aligned} \frac{\partial \rho_1}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u}_1) &= M \\ \frac{\partial \mathbf{u}_1}{\partial t} &= -\frac{1}{\rho_0} \nabla p_1 + \mathbf{F} \\ \frac{\partial p_1}{\partial t} &= c_s^2 \left(\frac{\partial \rho_1}{\partial t} + \mathbf{u}_1 \cdot \nabla \rho_0 \right) \end{aligned} \tag{2-4}$$

where c_s is recognized as the (isentropic) speed of sound (SI unit: m/s) at constant entropy s. The last equation for the pressure time differential is derived from the entropy equation, for more details see Theory Background for the Thermoviscous Acoustics Branch. For constant material parameters the last equation reduces to the usual relation

$$p_1 = c_s^2 \rho_1$$

This expression gives a useful condition that needs to be fulfilled for the linear acoustic equations to hold:

$$|p_1| \ll \rho_0 c_s^2$$

The subscript s is dropped in the following along with the subscript 0 on the background density ρ_0. Finally, rearranging Equation 2-4, renaming the source terms,

and dropping the subscript 1 yields the wave equation for pressure waves in a lossless medium

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p - \mathbf{q}_d)\right) = Q_m . \tag{2-5}$$

Here ρ (SI unit: kg/m^3) refers to the density, and c (SI unit: m/s) denotes the speed of sound. In this general formulation of the scalar wave equation, the speed of sound and density may in general be space dependent, for example, through their dependency on the background temperature. The equation includes two optional source terms:

- The *monopole domain source* Q_m (SI unit: $1/s^2$). This source corresponds to a mass source on the right-hand side of the continuity equation.
- The *dipole domain source* $\mathbf{q}_d$ (SI unit: N/m^3). This source corresponds to a domain force source on the right-hand side of the momentum equation.

The combination ρc^2 is called the *adiabatic bulk modulus*, commonly denoted K_s (SI unit: Pa). The bulk modulus is equal to the one over the adiabatic compressibility coefficient $\beta_s = 1/K_s = 1/\rho c^2$ (SI unit: $1/Pa$).

In Equation 2-5 both the speed of sound $c = c(\mathbf{x})$ and the density $\rho = \rho(\mathbf{x})$ may be dependent on the spatial coordinates $\mathbf{x}$ while they are independent of time, or only slowly varying in time (that is, at a time scale much slower than the variations in the acoustic signal). If both domain sources are set to zero and the density is constant in space, you can recover the standard wave equation

$$\frac{\partial^2 p}{\partial t^2} = c^2 \nabla^2 p$$

Some classical references on acoustics for further reading are found in Ref. 4, Ref. 5, Ref. 6, Ref. 7, and Ref. 8.

An important special case is a time-harmonic wave, for which the pressure varies with time as

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

where $\omega = 2\pi f$ (rad/s) is the angular frequency and f (SI unit: Hz) is the frequency. Assuming the same harmonic time dependence for the source terms, the wave equation for acoustic waves reduces to an inhomogeneous Helmholtz equation

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p - \mathbf{q}_d)\right) - \frac{\omega^2 p}{\rho_c c_c^2} = Q_m . \tag{2-6}$$

In this equation, the subscript "c" on the density and the speed of sound refers to that they may be complex valued. Lossy media, like porous materials or highly viscous fluids, can be modeled by using the complex-valued speed of sound and density. A selection of such fluid models is available in The Pressure Acoustics, Frequency Domain Interface. The attenuation in these fluid models is frequency dependent in different ways, depending on the physical origin of the damping. A description of the different fluid models is given in Theory for the Equivalent Fluid Models.

In the time domain, only certain frequency dependencies can be modeled, which limits the number of fluid models that can be used in The Pressure Acoustics, Transient Interface. One way to model damping in the time domain is to introduce an additional term of first order in the time derivative to account for attenuation of the sound waves

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} - d_a \frac{\partial p}{\partial t} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p - \mathbf{q}_d)\right) = Q_m \tag{2-7}$$

The damping term in Equation 2-7 is absent from the standard PDE formulations in the Pressure Acoustics, Transient interface, but it corresponds to a monopole domain source proportional to the time derivative of the pressure. This approach is, however, not used in the viscous and thermally conducting fluid models that contain damping for transient acoustics (see Viscous Fluid Model, Thermally Conducting Fluid Model, and Thermally Conducting and Viscous Fluid Model). The physical origin of the damping corresponds here to a dipole-like source.

Even when sound waves propagate in a lossless medium, attenuation can occur by interaction with the surroundings at the system boundaries. In particular, this applies to the impedance boundary conditions.

Alternatively, treat the Helmholtz Equation 2-6 as an eigenvalue PDE to solve for eigenmodes and eigenfrequencies, see the Eigenfrequency and Mode Analysis in 2D and 1D axisymmetric sections below.

In order to solve the governing equations, boundary conditions are necessary. Typical boundary conditions used in acoustics are:

- Sound Hard Boundary (Wall)

- Sound Soft Boundary (zero acoustic pressure)
- Specified acoustic Pressure
- Specified Normal Acceleration
- Impedance boundary conditions (see also Theory for the Boundary Impedance Models)
- Plane Wave Radiation
- Spherical Wave Radiation
- Cylindrical Wave Radiation

Pressure Acoustics, Frequency Domain Equations

The Pressure Acoustics, Frequency Domain Interface exists for several types of studies. Here the equations are presented for the frequency domain, eigenfrequency, and modal studies. All the interfaces solve for the acoustic pressure p. It is available in all space dimensions — for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

FREQUENCY DOMAIN

The frequency domain, or time-harmonic, formulation uses the inhomogeneous Helmholtz equation:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2 p_t}{\rho_c} = Q_m \tag{2-8}$$

This is Equation 2-6 repeated with the introduction of the wave number k_{eq} used in the equations. It contains both the ordinary wave number k as well as out-of-plane and circumferential contributions, when applicable. Note also that the pressure is here the total pressure p_t which is the sum of a possible Background Pressure Field p_b and the scattered field p_s. This enables for a so-called scattered field formulation of the equations. If no background field is present $p_t = p_s = p$.

In this equation, $p = p\,(\mathbf{x},\omega) = p(\mathbf{x})e^{i\omega t}$ (the dependence on ω is henceforth not explicitly indicated). Compute the frequency response by doing a parametric sweep over a frequency range using harmonic loads and sources.

When there is damping, ρ_c and c_c are complex-valued quantities. The available damping models and how to apply them is described in the sections Pressure Acoustics and Theory for the Equivalent Fluid Models.

Equation 2-8 is the equation that the software solves for 3D geometries. In lower-dimensional and axisymmetric cases, restrictions on the coordinate dependence mean that the equations differ from case to case. Here is a brief summary of the situation.

2D

In 2D, the pressure is of the form

$$p(\mathbf{x}) = p(x, y)e^{-ik_z z}$$

which inserted in Equation 2-8 gives

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2}{\rho_c} p_t = Q_m \qquad (2\text{-}9)$$
$$k_{eq}^2 = \frac{\omega^2}{c_c^2} - k_z^2$$

The *out-of-plane wave number* k_z can be set on the Pressure Acoustics page. By default its value is 0. In the mode analysis type $-ik_z$ is used as the eigenvalue λ.

2D Axisymmetry

For 2D axisymmetric geometries the independent variables are the radial coordinate r and the axial coordinate z. The only dependence allowed on the azimuthal coordinate φ is through a phase factor,

$$p(r, \phi, z) = p(r, z)e^{-im\varphi} \qquad (2\text{-}10)$$

where m denotes the *circumferential mode number*. The mode number defines a *circumferential wave number* $k_m = m/r$. Because the azimuthal coordinate is periodic m must be an integer. Just like k_z in the 2D case, m can be set on the Settings window for Pressure Acoustics.

As a result of Equation 2-10, the equation to solve for the acoustic pressure in 2D axisymmetric geometries becomes

$$\frac{\partial}{\partial r}\left[-\frac{r}{\rho_c}\left(\frac{\partial p}{\partial r} - q_r\right)\right] + r\frac{\partial}{\partial z}\left[-\frac{1}{\rho_c}\left(\frac{\partial p}{\partial z} - q_z\right)\right] - \frac{k_{eq}^2}{\rho_c} rp = rQ_m$$
$$k_{eq}^2 = \left(\frac{\omega}{c_c}\right)^2 - k_m^2 \qquad k_m = \frac{m}{r}$$

1D Axisymmetry

In 1D axisymmetric geometries,

$$p(r, \phi, z) = p(r)e^{-i(k_z z + m\varphi)}$$

leading to the radial equation

$$\frac{\partial}{\partial r}\left[-\frac{r}{\rho_c}\left(\frac{\partial p}{\partial r} - q_r\right)\right] - \frac{k_{eq}^2}{\rho_c} rp = rQ_m$$

$$k_{eq}^2 = \left(\frac{\omega}{c_c}\right)^2 - k_m^2 - k_z^2 \qquad k_m = \frac{m}{r}$$

where both the *circumferential wave number* k_m and the *axial wave number* k_z, appear as parameters.

1D

The equation for the 1D case is obtained by letting the pressure depend on a single Cartesian coordinate x:

$$\frac{d}{dx}\left(-\frac{1}{\rho_c}\left(\frac{dp}{dx} - q_d\right)\right) - \frac{k_{eq}^2}{\rho_c} p = Q_m$$

$$k_{eq}^2 = \left(\frac{\omega}{c_c}\right)^2$$

EIGENFREQUENCY

In the eigenfrequency formulation, the source terms are absent; the eigenmodes and eigenfrequencies are solved for:

$$\nabla \cdot \left(-\frac{1}{\rho_c}\nabla p\right) + \frac{\lambda^2 p}{\rho_c c_c^2} = 0 \tag{2-11}$$

The eigenvalue λ introduced in this equation is related to the eigenfrequency f, and the angular frequency ω, through $\lambda = i2\pi f = i\omega$. Because they are independent of the pressure, the solver ignores any dipole and monopole sources unless a coupled eigenvalue problem is being solved.

Equation 2-11 applies to the 3D case. The equations solved in eigenfrequency studies in lower dimensions and for axisymmetric geometries are obtained from their time-harmonic counterparts, given in the previous subsection, by the substitution $\omega^2 \rightarrow -\lambda^2$.

Switch between specifying the eigenvalues, the eigenfrequencies, and the angular frequencies by selecting from the Eigenvalue transformation list in the solver sequence's Settings window for Eigenvalue.

Vibrations of a Disk Backed by an Air-Filled Cylinder: Application Library path **Acoustics_Module/Verification_Examples/coupled_vibrations**

MODE ANALYSIS IN 2D AND 1D AXISYMMETRIC

See Mode Analysis Study in the Pressure Acoustics, Boundary Mode Equations section. The mode analysis study type is only available for the Pressure Acoustics, Frequency Domain interface in 2D and 1D axisymmetric components. Where the solver solves for the eigenvalues $\lambda = -ik_z$ for a given frequency. Here k_z is the out-of-plane wave number of a given mode and the resulting pressure field p represents the mode on the cross section of an infinite wave guide or duct.

Pressure Acoustics, Transient Equations

Use the Time Dependent study type to model transient acoustic phenomena in a stationary fluid and to solve the wave equation

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p - \mathbf{q}_\mathrm{d})\right) = Q_\mathrm{m}$$

for the acoustic pressure, $p = p(\mathbf{x}, t)$. Here c is the speed of sound and ρ denotes the equilibrium density, while $\mathbf{q}_\mathrm{d}$ and Q_m are dipole and monopole sources, respectively. The density and speed of sound can both be non constant in space. In contrast, they are assumed to vary with time on scales much larger than the period for the acoustic waves and are therefore considered time independent in the previous equation. This physics interface is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

Pressure Acoustics, Boundary Mode Equations

When an acoustic wave of a given angular frequency ω is fed into a waveguide or a duct, only a finite set of shapes, or modes, for the transverse pressure field can propagate over long distances inside the structure. The higher the frequency, the higher the number of sustainable modes.

Take, as an example, a uniform straight duct whose axis is in the z-direction. The acoustic field in such a duct can be written as a sum of the form

$$p(\mathbf{x}) = \sum_{j=0}^{N} p_j(x,y)e^{-ik_{zj}z}$$

The constant k_{zj} is the axial wave number of the jth propagating transverse mode, $p_j(x, y)$. These transverse modes and their associated axial wave numbers are solutions to an eigenvalue problem defined on the duct's cross section. The mode analysis capabilities in The Pressure Acoustics, Boundary Mode Interface makes it possible to solve such eigenvalue problems. The physics interface is available for 3D Cartesian and 2D axisymmetric geometries and solves for the transverse eigenmodes for the acoustic pressure p and the associated propagation constants k_z. The Mode Analysis Study is briefly discussed.

MODE ANALYSIS STUDY

The eigenvalue solver computes a specified number of solutions $\{p_j, \lambda_j\}$ to the equation

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p - \mathbf{q}_d)\right) - \left(\frac{\omega^2}{\rho_c c_c^2} - \frac{k_n^2}{\rho_c}\right)p = Q_m \qquad (2\text{-}12)$$

defined on a 2D boundary of the modeling domain (in 3D) or on the 2D domain itself, with $\lambda = -ik_n$ as the eigenvalue. In this equation, p is the in-plane pressure, ρ_c is the density, c_c is the speed of sound, ω is the angular frequency, and k_n is the propagation constant in the direction normal to the surface, in this context also referred to as the out-of-plane wave number.

	The out-of-plane wave number is denoted k_n, and is in the normal direction to the two-dimensional surface on which Equation 2-12. As for a mode analysis study in the frequency domain the propagation direction does not necessarily have to be normal to the z-axis for 3D geometries.

	Special postprocessing variables exist for the Pressure Acoustics, Boundary Mode interface. They are described in Pressure Acoustics, Boundary Mode Variables.

Notice that the above equation is identical to the time-harmonic equation for pressure acoustics, except that k_n is interpreted as an eigenvalue and not as a parameter.

For axisymmetric geometries, the relevant eigenvalue equation to solve for the radial pressure modes and the eigenvalues λ is

$$\frac{d}{dr}\left(\frac{r}{\rho_c}\frac{dp}{dr}\right) + \left[\left(\frac{\omega}{c_c}\right)^2 + \lambda^2 - \left(\frac{m}{r}\right)^2\right]\frac{rp}{\rho_c} = 0$$

Here m, the *circumferential mode number*, is an integer-valued parameter. The equation is defined on the interval $r_1 < r < r_2$. The eigenvalue λ is defined in terms of the axial wave number k_z through the equation $\lambda = -ik_z$

Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions

Specify a Plane Wave Radiation, Spherical Wave Radiation, or Cylindrical Wave Radiation boundary condition to allow an outgoing wave to leave the modeling domain with minimal reflections. The condition can be adapted to the geometry of the modeling domain. The *plane wave* type is suitable for both far-field boundaries and ports. Because many waveguide structures are only interesting in the plane-wave region, it is particularly relevant for ports.

Radiation boundary conditions are available for all types of studies. For the frequency-domain study, Givoli and Neta's reformulation of the Higdon conditions (Ref. 1) for plane waves has been implemented to the second order. For cylindrical and spherical waves, COMSOL Multiphysics uses the corresponding 2nd-order expressions from Bayliss, Gunzburger, and Turkel (Ref. 2). The Transient, Mode analysis, and Eigenfrequency studies implement the same expansions to the first order.

The first-order radiation boundary conditions in the frequency domain read

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) + (ik + \kappa(r))\frac{p}{\rho_c} = (ik + \kappa(r))\frac{p_i}{\rho_c} + \mathbf{n}\cdot\left(\frac{\nabla p_i}{\rho_c}\right)$$

where k is the wave number and $\kappa(r)$ is a function whose form depends on the wave type:

- Plane wave: $\kappa(r) = 0$
- Cylindrical wave: $\kappa(r) = 1/(2r)$
- Spherical wave: $\kappa(r) = 1/r$

In the cylindrical and spherical wave cases, r is the shortest distance from the point $\mathbf{r} = (x, y, z)$ on the boundary to the source. The right-hand side of the equation represents an optional incoming pressure field p_i (see Incident Pressure Field).

The second-order radiation boundary conditions in the frequency domain are defined below. In these equations, Δ_T at a given point on the boundary denotes the Laplace operator in the tangent plane at that particular point.

PLANE WAVE

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) + i\frac{k}{\rho_c}p + \frac{i}{2k\rho_c}\Delta_\mathrm{T} p = \frac{i}{2k\rho_c}\Delta_\mathrm{T} p_i + i\frac{k}{\rho_c}p_i + \mathbf{n}\cdot\frac{1}{\rho_c}\nabla p_i$$

In the notation of Givoli and Neta (Ref. 1), the above expressions correspond to the parameter choices $C_0 = C_1 = C_2 = \omega/k$. For normally incident waves, this gives a vanishing reflection coefficient.

CYLINDRICAL WAVE

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}\nabla p_t - \mathbf{q}_d\right) =$$

$$\left(ik_\mathrm{eq} + \frac{1}{2r} - \frac{1}{8r(1+ik_\mathrm{eq}r)}\right)\frac{(p_i - p)}{\rho_c} + \mathbf{n}\cdot\frac{1}{\rho_c}\nabla p_i + \frac{(r\Delta_\mathrm{T} p_i - r\Delta_\mathrm{T} p)}{2(1+ik_\mathrm{eq}r)\rho_c}$$

The *cylindrical wave* boundary condition is based on a series expansion of the outgoing wave in cylindrical coordinates (Ref. 2), and it assumes that the field is independent of the axial coordinate. Specify the axis of this coordinate system by giving an orientation (n_x, n_y, n_z) and a point (x_0, y_0, z_0) on the axis. In axisymmetric geometries, the symmetry axis is the natural and only choice.

SPHERICAL WAVE

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) + \left(ik_\mathrm{eq} + \frac{1}{r}\right)\frac{p}{\rho_c} - \frac{r\Delta_\mathrm{T} p}{2\rho_c(ik_\mathrm{eq}r + 1)}$$

$$= -\frac{r\Delta_\mathrm{T} p_\mathrm{i}}{2\rho_{0c}(ik_\mathrm{eq}r + 1)} + \left(ik_\mathrm{eq} + \frac{1}{r}\right)\frac{p_\mathrm{i}}{\rho_c} + \mathbf{n}\cdot\frac{1}{\rho_c}\nabla p_\mathrm{i}$$

Use a *spherical wave* to allow a radiated or scattered wave — emanating from an object centered at the point (x_0, y_0, z_0) that is specified — to leave the modeling domain without reflections. The boundary condition is based on an expansion in spherical coordinates from Bayliss, Gunzburger, and Turkel (Ref. 2), implemented to the second order.

TRANSIENT ANALYSIS

The transient radiation boundary condition is the first-order expression

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d)\right) + \frac{1}{\rho}\left(\frac{1}{c}\frac{\partial p}{\partial t} + \kappa(r)\,p\right) = \frac{1}{\rho}\left(\frac{1}{c}\frac{\partial p_\mathrm{i}}{\partial t} + \kappa(r)\,p_\mathrm{i} + \mathbf{n}\cdot\nabla p_\mathrm{i}\right)$$

where $\kappa(r)$ is the same wave-type dependent function as for the eigenfrequency case and p_i the optional Incident Pressure Field.

An estimate of the reflection coefficient R_s for spurious waves off the plane wave radiation boundary, for incident plane waves at angle θ is given by the expression:

$$R_\mathrm{s} = \left|\frac{\cos\theta - 1}{\cos\theta + 1}\right|^N$$

where N is the order of the boundary condition (here 1 or 2). So at normal incidence ($\theta = 0$) there are no spurious reflections, while, for example, at an incidence angle of 30^o for $N = 2$ (plane wave radiation in the frequency domain) the amplitude of the spurious reflected wave is 0.5 % of the incident.

Theory for the Far-Field Calculation: The Helmholtz-Kirchhoff Integral

The Acoustics Module has functionality to evaluate the acoustic pressure field in the far-field region. This section gives the relevant definitions and mathematical background as well as some general advice for analyzing the far field. Details about how to use the far-field functionality is described in Far-Field Calculation and in the modeling section Evaluating the Acoustic Field in the Far-Field Region.

THE HELMHOLTZ-KIRCHHOFF INTEGRAL REPRESENTATION

In many cases, solving the acoustic Helmholtz equation everywhere in the domain where results are requested is neither practical nor necessary. For homogeneous media, the solution anywhere outside a closed surface containing all sources and scatterers can be written as a boundary integral in terms of quantities evaluated on the surface. To evaluate this *Helmholtz-Kirchhoff* integral, it is necessary to know both Dirichlet and Neumann values on the surface. Applied to acoustics, this means that if the pressure

and its normal derivative (which is related to the normal velocity) is known on a closed surface, the acoustic field can be calculated at any point outside.

In general, the solution p to Helmholtz' equation

$$-\nabla \cdot \nabla p - k^2 p = 0$$

in the homogeneous domain exterior to a closed surface, S, can be explicitly expressed in terms of the values of p and its normal derivative on S:

$$p(\mathbf{R}) = \int_S (G(\mathbf{R}, \mathbf{r})\nabla p(\mathbf{r}) - \nabla G(\mathbf{R}, \mathbf{r})p(\mathbf{r})) \cdot \mathbf{n}\, dS$$

Here the coordinate vector $\mathbf{r}$ parameterizes S. The unit vector $\mathbf{n}$ is the outward normal to the exterior infinite domain; thus, $\mathbf{n}$ points *into* the domain that S encloses. The function $G(\mathbf{R}, \mathbf{r})$ is a Green's function satisfying

$$-\nabla \cdot \nabla G(\mathbf{R}, \mathbf{r}) - k^2 G(\mathbf{R}, \mathbf{r}) = \delta^{(3)}(\mathbf{R} - \mathbf{r})$$

This essentially means that the Green's function, seen as a function of $\mathbf{r}$, is an outgoing traveling wave excited by a simple source at $\mathbf{R}$. In 3D, the Green's function is therefore:

$$G(\mathbf{R}, \mathbf{r}) = \frac{e^{-ik|\mathbf{r}-\mathbf{R}|}}{4\pi|\mathbf{r}-\mathbf{R}|}$$

In 2D, the Green's function contains a Hankel function instead of the exponential:

$$G(\mathbf{R}, \mathbf{r}) = \frac{i}{4} H_0^{(2)}(k|\mathbf{r}-\mathbf{R}|)$$

Inserting the 3D Green's function in the general representation formula gives:

$$p(\mathbf{R}) = \frac{1}{4\pi}\int_S \frac{e^{-ik|\mathbf{r}-\mathbf{R}|}}{|\mathbf{r}-\mathbf{R}|}\left(\nabla p(\mathbf{r}) + p(\mathbf{r})\frac{(1+ik|\mathbf{r}-\mathbf{R}|)}{|\mathbf{r}-\mathbf{R}|^2}(\mathbf{r}-\mathbf{R})\right) \cdot \mathbf{n}\, dS \qquad (2\text{-}13)$$

while in 2D, the Hankel function leads to a slightly different expression:

$$p(\mathbf{R}) = -\frac{i}{4}\int_S \left(H_0^{(2)}(k|\mathbf{r}-\mathbf{R}|)\nabla p(\mathbf{r}) + kp(\mathbf{r})\frac{H_1^{(2)}(k|\mathbf{r}-\mathbf{R}|)}{|\mathbf{r}-\mathbf{R}|}(\mathbf{r}-\mathbf{R})\right) \cdot \mathbf{n}\, dS \quad (2\text{-}14)$$

For axially symmetric geometries, the full 3D integral must be evaluated. The Acoustics Module uses an adaptive numerical quadrature in the azimuthal direction on

a fictitious revolved geometry in addition to the standard mesh-based quadrature in the *rz*-plane.

To evaluate the full Helmholtz-Kirchhoff integral in Equation 2-13 and Equation 2-14, use the Full integral option in the settings for the far-field variables. See Far-Field Calculation.

THE FAR-FIELD LIMIT

The full Helmholtz-Kirchhoff integral gives the pressure at any point at a finite distance from the source surface, but the numerical integration tends to lose accuracy at large distances. At the same time, in many applications, the quantity of interest is the far-field radiation pattern, which can be defined as the limit of $r|p|$ when r goes to infinity in a given direction.

Taking the limit of Equation 2-13 when $|\mathbf{R}|$ goes to infinity and ignoring the rapidly oscillating phase factor, the far field, p_{far} is defined as

$$p_{\text{far}}(\mathbf{R}) = -\frac{1}{4\pi}\int_S e^{ik\frac{\mathbf{r}\cdot\mathbf{R}}{|\mathbf{R}|}}\left(\nabla p(\mathbf{r}) - ikp(\mathbf{r})\frac{\mathbf{R}}{|\mathbf{R}|}\right)\cdot\mathbf{n}\,dS$$

The relevant quantity is $|p_{\text{far}}|$ rather than p_{far} because the phase of the latter is undefined. For the same reason, only the direction of $\mathbf{R}$ is important, not its magnitude.

Because Hankel functions asymptotically approach exponential, the limiting 2D integral is remarkably similar to that in the 3D case:

$$p_{\text{far}}(\mathbf{R}) = \frac{1-i}{4\sqrt{\pi k}}\int_S e^{ik\frac{\mathbf{r}\cdot\mathbf{R}}{|\mathbf{R}|}}\left(\nabla p(\mathbf{r}) - ikp(\mathbf{r})\frac{\mathbf{R}}{|\mathbf{R}|}\right)\cdot\mathbf{n}\,dS$$

For axially symmetric geometries, the azimuthal integral of the limiting 3D case can be handled analytically, which leads to a rather complicated expression but avoids the numerical quadrature required in the general case. For zero circumferential mode number $m = 0$, the expression is:

$$p_{\text{far}}(\mathbf{R}) \cong -\frac{1}{2}\int_S re^{ik\frac{zZ}{|\mathbf{R}|}}\left[J_0\left(\frac{krR}{|\mathbf{R}|}\right)\nabla p(\mathbf{r})\cdot\mathbf{n} - \frac{ikp(\mathbf{r})}{|\mathbf{R}|}\left(in_r RJ_1\left(\frac{krR}{|\mathbf{R}|}\right) + n_z ZJ_0\left(\frac{krR}{|\mathbf{R}|}\right)\right)\right]dS \tag{2-15}$$

In this integral, r and z are the radial and axial components of $\mathbf{r}$, while R and Z are the radial and axial components of $\mathbf{R}$.

To evaluate the pressure in the far-field limit according to the equations in this section, use the **Integral approximation at $r \to \infty$** option in the **Settings** window for the far-field variables See Far-Field Calculation.

THE ELKERNEL ELEMENT

These integrals can be implemented as integration coupling variables in COMSOL Multiphysics. However, such an approach is very inefficient because then the simple structure of the integration kernels cannot be exploited. In the Acoustics Module, convolution integrals of this type are therefore evaluated in optimized codes that hides all details from the user.

Theory for the Boundary Impedance Models

In this section:

Impedance Conditions

An impedance boundary condition relates the acoustic pressure and the acoustic velocity to each other at a given point. This relationship is provided by knowledge about the conditions on the boundary, for example, a specific material or an acoustically active boundary. Therefore, an impedance boundary condition can be used to impose the properties of the boundary without modeling it explicitly. Impedance boundary conditions thus generalize the sound-hard and sound-soft boundary conditions to address a large number of cases between these two extremes.

Mathematically, a specific acoustic impedance Z_i is defined on some cross-section as the ratio between the acoustic pressure p and the acoustic velocity perpendicular to the area $v_\perp$ (the normal velocity)

$$Z_\mathrm{i} = \frac{p}{v_\perp}.$$

In Pressure Acoustics, Frequency Domain, this boundary condition is imposed as

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_\mathrm{t} - \mathbf{q}_\mathrm{d})\right) = -\frac{i\omega p_\mathrm{t}}{Z_\mathrm{i}} \tag{2-16}$$

while in the Pressure Acoustics, Transient interface using a Time Dependent study, the impedance boundary condition is the following:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p - \mathbf{q}_d)\right) = \frac{1}{Z_\mathrm{i}}\frac{\partial p_\mathrm{t}}{\partial t} \tag{2-17}$$

Here Z_i is the acoustic input impedance of the external domain and it has the unit of a specific acoustic impedance. The specific acoustic impedance Z_i (SI unit: Pa·s/m) is related to the acoustic impedance Z_ac (ratio of pressure and flow rate, SI unit: Pa·s/m^3) and the mechanical impedance Z_mech (ratio of force and velocity, SI unit: N·s/m) via the area A of the boundary, according to

$$Z_\mathrm{mech} = AZ_\mathrm{i} = A^2 Z_\mathrm{ac} \tag{2-18}$$

Impedance boundary conditions only relate the normal velocity (velocity perpendicular to the boundary) to the pressure, but do not consider the tangential velocity (component parallel to the boundary). This is due to the mathematical construction of the governing equation and the fact that pressure acoustics solves only for the scalar pressure. Put differently, the impedance boundary condition only applies to the normal component of the incident field. Thus, by applying an impedance boundary condition this tangential velocity component is ignored altogether. For this reason, impedance boundary conditions are in most cases low-order approximations to the actual boundary properties. In cases where this is unacceptable, it is consequently better to either model the boundary explicitly or use a higher-order model, such as for instance the Plane Wave Radiation at an open boundary.

RCL Models

Circuit models are used as low-order models of acoustically active boundaries specifying the input impedance of the external domain. For example, modeling mechanical properties of a loudspeakers cone, a microphone diaphragm, or biological tissue, as experienced by the incident wave. For such boundaries, the pressure and velocity are modulated in response to the behavior of the boundary.

An RCL model is intended to provide a simple model to represent the frequency-dependent mechanical properties of a system that typically has some loss, a mass, and a compliance. The option includes all combinations of a three-element circuit consisting of an acoustic damper (acoustic resistance, a resistor R_ac), an acoustic mass (acoustic inertance, an inductor L_ac) and an acoustic compliance (a capacitor C_ac), which are the simplest resonating acoustic circuits. While full circuit models tend to be more sophisticated (see for example Ref. 26), one can typically identify a simple core circuit responsible for the basic response which consists of such three elements.

The acoustic impedance Z_{ac} is related to the specific impedance Z_i implemented in the weak equations Equation 2-16 and Equation 2-17 by the area of the boundary A as

$$Z_i = AZ_{ac}.$$

CIRCUIT MODEL OPTIONS

A schematic illustration is provided for each circuit in the Equation Display window.

Serial coupling of R_{ac}, C_{ac} and L_{ac}

$$Z_{ac} = R_{ac} + i\omega L_{ac} + \frac{1}{i\omega C_{ac}}$$

Parallel coupling of R_{ac}, C_{ac} and L_{ac}

$$Z_{ac} = \left[i\omega C_{ac} + \frac{1}{R_{ac}} + \frac{1}{i\omega L_{ac}}\right]^{-1}$$

Parallel coupled L_{ac} and C_{ac} in series with R_{ac}

$$Z_{ac} = R_{ac} + \left[i\omega C_{ac} + \frac{1}{i\omega L_{ac}}\right]^{-1}$$

Parallel coupled R_{ac} and C_{ac} in series with L_{ac}

$$Z_{ac} = i\omega L_{ac} + \left[\frac{1}{R_{ac}} + i\omega C_{ac}\right]^{-1}$$

Parallel coupled R_{ac} and L_{ac} in series with C_{ac}

$$Z_{ac} = \frac{1}{i\omega C_{ac}} + \left[\frac{1}{R_{ac}} + \frac{1}{i\omega L_{ac}}\right]^{-1}$$

Serial coupled R_{ac} and C_{ac} in parallel with L_{ac}

$$Z_{ac} = \left[\frac{1}{i\omega L_{ac}} + \left(\frac{1}{i\omega C_{ac}} + R_{ac}\right)^{-1}\right]^{-1}$$

Serial coupled L_{ac} and C_{ac} in parallel with R_{ac}

$$Z_{ac} = \left[\frac{1}{R_{ac}} + \left(i\omega L_{ac} + \frac{1}{i\omega C_{ac}}\right)^{-1}\right]^{-1}$$

Serial coupled R_{ac} and L_{ac} in parallel with C_{ac}

$$Z_{ac} = [i\omega C_{ac} + (i\omega L_{ac} + R_{ac})^{-1}]^{-1}$$

LIMITATIONS

A circuit model is one-dimensional and considers only changes in the direction normal to the boundary; any variations across the boundary are ignored. Therefore, circuit models provide good results at low frequencies with wavelengths substantially larger than the dimensions of the boundary.

ACOUSTICAL OR MECHANICAL CIRCUITS

It is sometimes advantageous to use circuit models in mechanical units rather than acoustical units. In these cases, the overall structure of the impedance boundary condition equation does not change, but the numerical value of the parameters do. The mechanical impedance Z_{mech} (the ratio of force F and velocity v) and the acoustic impedance Z_{ac} are related via the boundary surface area A by the expression

$$Z_{\text{mech}} = A^2 Z_{\text{ac}} .$$

Physiological Models

The physiological models are simple equivalent circuit models for parts of the human body which typically are of importance in acoustical applications, namely the skin, the full human ear, the eardrum and inner ear, and the acoustic impedance from the ear's pinna. These models are good approximations to the active, acoustical properties of these body parts with particular relevance for development of hearing aids, mobile devices, and head phones.

HUMAN SKIN MODEL

This model has been developed as a lowest-order, reasonable approximation to experimental data in Ref. 27 (see their Figure 9). It consists of a simple serial RCL circuit applied on a transducer area of A_{t}, and has the impedance

$$Z_{\text{i}} = \frac{R_{\text{s}}}{A_{\text{t}}} + \frac{i\omega L_{\text{s}}}{A_{\text{t}}} + \frac{1}{i\omega C_{\text{s}} A_{\text{t}}}$$

with $A_{\text{t}} = 1.5\ \text{cm}^2$, $R_{\text{s}} = 9.0\ \text{N s m}^{-1}$, $L_{\text{s}} = 0.53\times10^{-3}\ \text{N s}^2\ \text{m}^{-1}$ and $C_{\text{s}} = 5.3\times10^{-6}\ \text{m N}^{-1}$.

MODELS RELATED TO THE HUMAN EAR

Four models related to the human ear have been included. These detailed and experimentally-verified circuit models describe parts of the human ear, as well as the entire human ear, see Ref. 28 to 30 for further details.

Outward human ear radiation

This impedance describes the acoustic radiation losses from the pinna (also known as the auricle, this is the visible part of the ear which is exterior to the head), see Figure 2-8. For cases where you model the ear canal explicitly using pressure acoustics, this boundary condition describes the acoustic losses from the outward acoustic radiation from the ear canal and into the surrounding air.

$$Z_{\text{rad}} = \left[Y_{\text{par}} + \sum_{k=1}^{3} Y_k \right]^{-1}, \qquad (2\text{-}19)$$

$$Y_{\text{par}} = \frac{1}{R_{\text{par}}} + \frac{1}{i\omega L_{\text{par}}} + i\omega C_{par} \qquad Y_k = R_k\left(1 + iQ\left[\frac{\omega}{\omega_k} - \frac{\omega_k}{\omega}\right]\right)$$

where $R_{\text{par}} = 7.0\times10^{6}$ N s m^{-5}, $L_{\text{par}} = 100$ N s^2 m^{-5}, $C_{\text{par}} = 1.7\times10^{-12}$ m^5 N^{-1}, $Q = 6$, $R_1 = R_2 = R_{\text{par}}$, $R_3 = 2\,R_{par}$, $\omega_1 = 6000\cdot2\pi$ Hz, $\omega_1 = 9000\cdot2\pi$ Hz and $\omega_1 = 13000\cdot2\pi$ Hz.

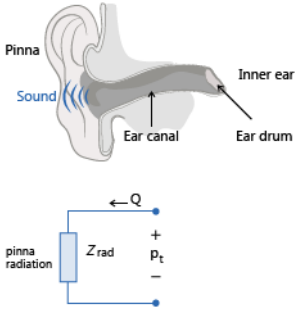

Figure 2-8: Illustration of outward human ear radiation.

Human ear drum impedance

This model describes the impedance of the human ear drum and the entire inner ear, that is, the acoustic impedance experienced in the ear canal when looking into the ear drum, see Figure 2-9. The model equations are given in Equation 2-20 and

Equation 2-21, and the parameter values in Table 2-8.

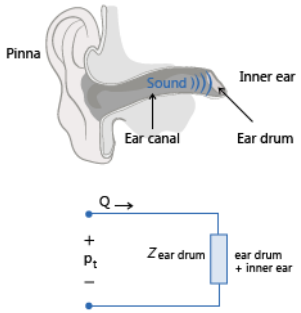

Figure 2-9: Illustration of human eardrum impedance.

$$Z_{\text{eardrum}} = Z_{\text{cav}} + \left[Y_{\text{ac}} + \frac{1}{Z^{\text{a}}_{\text{inner ear}}} \right]^{-1}$$

$$Z^{\text{a}}_{\text{inner ear}} = \frac{1}{Z^{\text{m}}_{\text{inner ear}} A_D(\omega)^2}$$

$$Z^{\text{m}}_{\text{inner ear}} = \left[\frac{1}{Z_{\text{dm}}} + \left[\frac{1}{Y_{\text{mi}}} + \left[\frac{1}{Z_{\text{incus}}} + \frac{1}{Z_{\text{st}}} + \frac{1}{Z^{\text{m}}_{\text{c}}} \right]^{-1} \right]^{-1} \right]^{-1}$$

$$A_D(\omega) = \begin{cases} A(\omega) & \text{if } \omega < \omega_{\text{Aph}} \\ |A(\omega)| e^{i\Phi_A(\omega)} & \text{if } \omega \geq \omega_{\text{Aph}} \end{cases} \qquad \Phi_{\text{A}}(\omega) = s_{\text{Aph}} \log\left(\frac{\omega}{\omega_{\text{Aph}}} \right) + \varphi^{(2\text{-}20)}_{\text{A}}$$

$$A(\omega) = \frac{A_0 + A_\infty}{1 - \left(\frac{\omega}{\omega_{\text{A}}} \right)^2 + \frac{i\omega}{Q_{\text{A}}\omega_{\text{A}}}} - A_\infty$$

$$Z_{\text{cav}} = \left[Y_{\text{tcav}} + \frac{1}{Z_{\text{ada}} + \frac{1}{Z_{\text{ant}}}} \right]^{-1} \qquad Y_{\text{tcav}} = \left[R_{\text{tcav}} + \frac{1}{i\omega R_{\text{tcav}}} \right]$$

$$Z_{\mathrm{ada}} = R_{\mathrm{ada}} + i\omega L_{\mathrm{ada}} \qquad Y_{\mathrm{ant}} = i\omega C_{\mathrm{ant}} + \frac{i\omega C_{\mathrm{mac}}}{1 - \left(\frac{\omega}{\omega_{\mathrm{mac}}}\right)^2 + \frac{i\omega}{Q_{\mathrm{mac}}\omega_{\mathrm{mac}}}}$$

$$Y_{\mathrm{ac}} = \frac{1}{R_{\mathrm{ac}} + i\omega L_{\mathrm{ac}}(\omega) + \frac{1}{i\omega C_{\mathrm{ac}}}} \mathrm{e}^{i\Phi_{\mathrm{Y}}(\omega)} \qquad L_{\mathrm{ac}}(\omega) = L_{\mathrm{ac}_0}\left(1 + \sqrt{\frac{\omega}{\omega_{Y_{L_{\mathrm{ac}}}}}}\right)$$

$$\Phi_{\mathrm{Y}}(\omega) = s_{\mathrm{Yph}} \log\left(1 + \sqrt{\frac{\omega}{\omega_{\mathrm{Yph}}}}\right) \qquad Y_{\mathrm{mi}} = R_{\mathrm{mi}} + \frac{1}{i\omega C_{\mathrm{mi}}}$$

$$Z_{\mathrm{dm}} = \frac{2}{3} Z_{\mathrm{dmi}} \qquad Z_{\mathrm{incus}} = \frac{1}{3} Z_{\mathrm{dmi}} \tag{2-21}$$

$$Z_{\mathrm{dmi}} = \left[\frac{1}{i\omega C_{\mathrm{oss}}} + i\omega L_{\mathrm{oss}} + (Z_{\mathrm{cpl}} + Z_{\mathrm{free}})^{-1}\right]^{-1} \qquad Z_{\mathrm{cpl}} = \left[R_{\mathrm{cpl}} + \frac{1}{i\omega C_{\mathrm{cpl}}}\right]$$

$$Z_{\mathrm{free}} = [R_{\mathrm{free}} + i\omega L_{\mathrm{free}}]^{-1} \qquad Z_{\mathrm{st}} = \frac{1}{R_{\mathrm{st}} + i\omega L_{\mathrm{st}} + \frac{1}{i\omega C_{\mathrm{st}}}}$$

$$Z_{\mathrm{c}}^{\mathrm{m}} = \frac{1}{Z_{\mathrm{c}}^{\mathrm{a}}} \frac{1}{A_F^2} \qquad Z_{\mathrm{c}}^{\mathrm{a}} = R_{\mathrm{c}} + i\omega L_{\mathrm{c}} + \frac{1}{i\omega C_{\mathrm{c}}}$$

The parameter values are given in Table 2-8. Note that the value of φ_{A} is not reported in the papers Ref. 28 to 30, but has instead been read off during model implementation.

TABLE 2-8: LIST OF PARAMETERS FOR THE EAR DRUM IMPEDANCE.

SYMBOL	VALUE	DESCRIPTION
R_{tcav}	2×10^{6} N s m^{-5}	Tympanic cavity resistance
C_{tcav}	$(0{,}5\mathrm{cm}^3)/(\gamma p_{\mathrm{A}})$	Tympanic cavity compliance
R_{ada}	1.7×10^{6} N s m^{-5}	Aditus ad antrum resistance
L_{ada}	880 N s^2 m^{-5}	Aditus ad antrum inertance
C_{ant}	$(0{,}8\mathrm{cm}^3)/(\gamma p_{\mathrm{A}})$	Antrum compliance
Q_{mac}	0.4	Mastoid air cell resonance quality factor
C_{mac}	$(8\mathrm{cm}^3)/(\gamma p_{\mathrm{A}})$	Mastoid air cell compliance
ω_{mac}	2π3500 rad s^{-1}	Mastoid air cell resonance angular frequency
R_{ac}	4×10^{7} N s m^{-5}	Ear drum shunt resistance
C_{ac}	5×10^{-12} N^{-1} m^5	Ear drum shunt compliance
L_{ac_0}	2.4×10^{3} kg m^{-4}	Zero-frequency ear drum shunt inertance
$\omega_{Y_{L_{\mathrm{ac}}}}$	$2\pi\cdot1900$ rad s^{-1}	Ear drum shunt inertance cross-over frequency
ω_{Yph}	$2\pi\cdot8000$ rad s^{-1}	Ear drum shunt admittance cross-over frequency

TABLE 2-8: LIST OF PARAMETERS FOR THE EAR DRUM IMPEDANCE.

SYMBOL	VALUE	DESCRIPTION
s_{Yph}	1.4	Ear drum shunt admittance decay slope
A_0	38 mm^2	Effective ear drum area, low frequency limit
A_∞	2 mm^2	Effective ear drum area, infinite frequency limit
ω_A	$2\pi \cdot 2200$ rad s^{-1}	Effective ear drum area resonance angular frequency
Q_A	1.3	Effective ear drum area resonance quality factor
s_{Aph}	-1.2	Effective ear drum area high-frequency decay slope
ω_{Aph}	$2\pi \cdot 1500$ rad s^{-1}	Effective ear drum area cross-over frequency
φ_A	0.8038	Effective ear drum area lag phase constant
R_{mi}	1 N s m^{-1}	Mechanical resistance of incudomalleal joint
C_{mi}	0.04 mm N^{-1}	Mechanical compliance of incudomalleal joint
C_{oss}	3×10^{-3} m N^{-1}	Mechanical compliance of ossicles
L_{oss}	7×10^{-3} g	Mechanical inertance of ossicles
C_{cpl}	0.5×10^{-3} m N^{-1}	Mechanical compliance of ossicles-drum coupling
R_{cpl}	0.08 N s m^{-1}	Mechanical resistance of ossicles-drum coupling
R_{free}	0.02 N s m^{-1}	Mechanical resistance of peripheral parts of the drum
L_{free}	12×10^{-3} g	Mechanical inertance of peripheral parts of the drum
R_{st}	18×10^{-3} N s m^{-1}	Mechanical resistance of stapes
L_{st}	3×10^{-3} g	Mechanical inertance of stapes
C_{st}	1.2×10^{-3} m N^{-1}	Mechanical compliance of stapes
R_c	70×10^{-3} N s m^{-1}/ A_F^2	Resistance of cochlea
L_c	10×10^{-3} g / A_F^2	Inertance of cochlea
C_c	11×10^{-3} m N^{-1} × A_F^2	Compliance of cochlea
A_F	3 mm^2	Stapes footplate area

Human ear without pinna

This model accounts for the acoustic losses associated with the ear canal and the entire human ear, see Figure 2-10. It does not include the radiation losses associated with the pinna, the visible part of the ear which is external to the head.

$$Z_{\text{ear w/o pinna}} = \frac{c_{11} + \dfrac{c_{12}}{Z_{\text{eardrum}}}}{c_{21} + \dfrac{c_{22}}{Z_{\text{eardrum}}}} \tag{2-22}$$

Here, Z_{eardrum} is the eardrum impedance defined in Equation 2-21 and c_{ij} are the components of the ear canal two port C. The ear canal is treated as N_{tot} small segments each with length Δ_k and radius r_k so its two port C is given by

$$C = \begin{bmatrix} c_{11} & c_{12} \\ c_{21} & c_{22} \end{bmatrix} = \prod_{k=1}^{N_{\text{tot}}} \begin{bmatrix} \cosh(\Gamma_k \Delta_k) & \sinh(\Gamma_k \Delta_k) Z_{\text{tw}_k} \\ \dfrac{\sinh(\Gamma_k \Delta_k)}{Z_{\text{tw}_k}} & \cosh(\Gamma_k \Delta_k) \end{bmatrix}. \tag{2-23}$$

where

$$\begin{gathered} Z_{\text{tw}_k} = \frac{\rho c}{\pi r_k^2} \qquad \Gamma_k = 3\alpha_k + i\frac{\omega}{c} \\ \alpha_k = \frac{\omega}{c}\left[\frac{1}{\sqrt{2}\text{Wo}_k}\zeta_1 + \frac{1}{\text{Wo}_k^2}\zeta_2 + \frac{1}{\text{Wo}_k^3\sqrt{2}}\zeta_3\right] \\ \zeta_1 = 1 + \frac{\gamma - 1}{\sqrt{\text{Pr}}} \qquad \zeta_2 = 1 + \frac{\gamma - 1}{\sqrt{\text{Pr}}} - \frac{\gamma - 1 + (\gamma - 1)^2}{2\text{Pr}} \\ \zeta_2 = \frac{7}{8} + \frac{\gamma - 1}{\sqrt{\text{Pr}}} - \frac{\gamma - 1 + (\gamma - 1)^2}{2\text{Pr}} + \frac{4(\gamma - 1)^2 + 4(\gamma - 1)^3 - (\gamma - 1)}{8\text{Pr}^{\frac{3}{2}}} \end{gathered} \tag{2-24}$$

In these expressions, γ is the ratio of specific heats, Γ_k is the propagation constant ("wave number") of the k^{th} ear canal segment which has the segment-specific attenuation α_k,

$$\text{Wo}_k = \sqrt{\frac{\rho \omega r_k^2}{\mu}}$$

is the segment-specific Womersley number, and

$$\text{Pr} = \frac{C_p \mu}{k}$$

is the Prandtl number expressed in terms of the specific hear C_p, dynamic viscosity μ, and thermal conductivity k. Notice that the papers presenting the model (Ref. 28 to

30) do not exactly specify which expression for the attenuation constants α_k is being used, but only refer to Ref. 31. The expression above for α_k is the most general expression taken from this paper. The values for Δ_k and r_k are listed in Table 2-9.

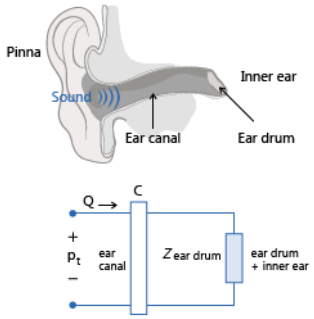

Figure 2-10: Illustration of the impedance of the human ear without pinna.

TABLE 2-9: RADII r_k AND LENGTHS Δ_k OF EAR CANAL SEGMENTS TAKEN FROM Ref. 30

k	r_k [mm]	Δ_k [mm]
1	1.0	1
2	1.9	1
3	2.4	1
4	2.6	1
5	2.75	1
6	2.85	1
7	2.95	1
8	3.05	1
9	3.2	2
10	3.35	2
11	3.5	2
12	3.6	2
13	3.7	2
14	3.8	2
15	3.9	2
16	4.0	2
17	4.1	2
18	4.25	2

The pressure at the eardrum p_{eardrum} is calculated whenever this impedance boundary condition is applied. This pressure is available in the plot group, and is calculated from the expression

$$p_{\text{eardrum}} = p_{\text{t}} \frac{c_{22} - \dfrac{c_{12}}{Z_{\text{ear w/o pinna}}}}{c_{11}c_{22} - c_{12}c_{21}},$$

where p_{t} is the pressure on the boundary, c_{ij} are the coefficients of the ear canal two port C and $Z_{\text{ear w/o pinna}}$ is the ear impedance; C and $Z_{\text{ear w/o pinna}}$ are defined in Equation 2-22 and Equation 2-23 above.

Human ear, full

This model accounts for all acoustic losses associated with the entire human ear, both the internal parts as well as the pinna, the visible, external part of the ear on the head, see Figure 2-11. The model does not include any information about the directivity of the ear (the head related transfer functions, HRTFs) which depends on the ear geometry at higher frequencies. It is valid in the low frequency limit and for normal incidence on the ear. It is given by

$$Z_{\text{ear full}} = Z_{\text{rad}} + Z_{\text{ear w/o pinna}}$$

where Z_{rad} and $Z_{\text{ear w/o pinna}}$ are given by Equation 2-19 and Equation 2-22 above.

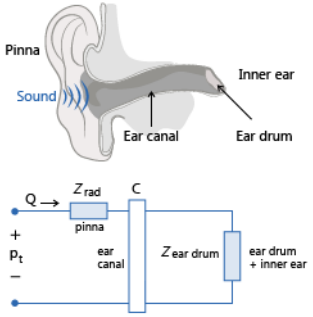

Figure 2-11: Illustration of the impedance of the full human ear including radiation losses due to the pinna.

The pressure at the eardrum p_{eardrum} is calculated whenever this impedance boundary condition model is applied. This pressure is calculated from $Z_{\text{ear full}}$ using the expression

$$p_{\text{eardrum}} = p_t\left(1 - \frac{Z_{\text{rad}}}{Z_{\text{ear full}}}\right)\frac{c_{22} - \dfrac{c_{12}}{Z_{\text{ear full}}}}{c_{11}c_{22} - c_{12}c_{21}}.$$

Waveguide End Impedance Models

Tubes and ducts are acoustic waveguides, and there are acoustic radiation losses when such a waveguide opens into a large domain. Idealized models for these losses have been implemented as boundary impedance models. Thus, instead of explicitly modeling the large domain, an appropriate impedance model Z_{end} can be applied with $Z_i = Z_{\text{end}}$. These models all assume that the domain is infinitely big, that the propagation is in the direction of the waveguide axis, and that the propagating mode is a plane wave. As with all other impedance boundary models, only the boundary-normal velocity component is taken into account.

Flanged pipe, circular

For a pipe of a user-specified radius a, the acoustic losses are given by (see Ref. 6)

$$Z_{\text{end}} = \rho c\left(1 - \frac{2J_1(2ka)}{2ka} + i\frac{2H_1(2ka)}{2ka}\right) \qquad k = \frac{\omega}{c}$$

where $J_1(x)$ is the Bessel function of the first kind of order 1, $H_1(x)$ is the Struve function and k is the wave number of the wave. This expression is also known as the impedance from a baffled piston.

Flanged pipe, rectangular

For a rectangular duct of user-specified inner width w_i and inner height h_i, the acoustic losses are given by (see Ref. 5)

$$Z_{\text{end}} = \frac{\rho c}{2\pi w_i h_i}\left[(w_i h_i)^2 k^2 - ik(w_i h_i)^{\frac{3}{2}} f\left(\frac{w_i}{h_i}\right)\right] \qquad k = \frac{\omega}{c}$$

$$f(x) = 2x^{\frac{1}{2}}\sinh^{-1}\left(\frac{1}{x}\right) + 2x^{-\frac{1}{2}}\sinh^{-1}(x) + \frac{2}{3}\left[x^{\frac{3}{2}} + x^{-\frac{3}{2}} - (x + x^{-1})^{\frac{3}{2}}\right].$$

This relationship applies provided the following requirements are satisfied

$$w_i \geq h_i \qquad kw_i \ll 1 \qquad kh_i \ll 1.$$

Unflanged pipe, circular (low ka limit)

For an unflanged circular pipe of a user-specified radius a in the limit of small radius (low ka), the pipe end impedance is given by the classical expression (see Ref. 6)

$$Z_{\text{end}} = \rho c\left[\frac{(ka)^2}{4} + i0.6133ka\right] \qquad k = \frac{\omega}{c} \qquad ka \ll 1\,.$$

Unflanged pipe, circular

For an unflanged pipe of any user-specified radius a relative to the wave number k, an approximate end impedance is given in Ref. 32. It is

$$Z_{\text{end}} = \rho c\frac{1+R}{1-R} \qquad R = -|R|e^{2ika\delta(ka)} \qquad k = \frac{\omega}{c}$$

$$|R| = \begin{cases} e^{-\frac{(ka)^2}{2}}\left[1 + \frac{(ka)^4}{6}\left(\log\left(\frac{1}{e^{0.5772}ka}\right) + \frac{19}{12}\right)\right] & \text{if } ka < 1 \\ \sqrt{\pi ka}\,e^{-ka}\left[1 + \frac{3}{32}\frac{1}{(ka)^2}\right] & \text{if } 1 < ka < 3.83 \end{cases}$$

where $\delta(ka)$ is a tabulated function reproducing the curve in Fig. 2 in Ref. 32 (where $\delta(ka)$ is referred to by l/a).

Porous Layer Models

A boundary impedance model is implemented to specifically handle cases investigating the acoustical properties of a porous layer of a given thickness d backed by a sound-hard wall. This can be applied instead of explicitly modeling the porous layer, as long as the incident acoustic field is normal to the boundary; as for all the other boundary impedance models, the tangential components of the acoustic field are ignored by this model.

For a porous layer with a user-specified thickness d, the impedance from the porous layer backed by a sound-hard wall is given by (see Ref. 9)

$$Z_{\text{i}} = -i\rho_{\text{c}}c_{\text{c}}\cot(kd) \qquad k = \frac{\omega}{c_{\text{c}}}\,.$$

Here ρ_{c} and c_{c} are the equivalent fluid descriptions of the porous model. This impedance model applies to any type of porous model which can be written as an equivalent fluid model. All porous models implemented in COMSOL are available for

this impedance boundary condition. See details of the poroacoustics equivalent fluid models in the section Theory for the Equivalent Fluid Models.

Characteristic Specific Impedance Models

For well-defined wave types in infinite domains, an impedance condition exists in every point whereby the pressure and normal velocity are related. Thus, these impedances can be imposed on a boundary to model an infinite, open domain in cases where the wave source inside the domain is either a direction (plane wave), a point (spherical wave) or a line (cylindrical wave). However, be aware that only the boundary-normal component of the velocity is used in the impedance boundary condition while the tangential component is ignored; in cases of non-negligible tangential components it is recommended to instead use the options Plane Wave Radiation, Spherical Wave Radiation, and Cylindrical Wave Radiation.

Plane wave

The impedance is given by

$$Z_i = \rho_c c_c ,$$

see Ref. 6. This is given solely by material parameters and has no user input.

Spherical wave

This impedance corresponds to the wave from a point source. It is calculated from the expression (given in Ref. 6)

$$Z_i = \frac{\rho_c c_c}{1 + \frac{1}{ikr_b}} \qquad r_b = |\mathbf{x}_0 - \mathbf{x}| \qquad k = \frac{\omega}{c_c}$$

where $\mathbf{x}_0$ is the user-specified location of the point source generating the spherical waves, $\mathbf{x}$ is the position of the boundary, and r_b is the distance between the two.

Cylindrical wave

This classic infinite-domain wave impedance from an acoustic line source with the user-specified direction $\hat{\mathbf{n}}_0$ and position $\mathbf{x}_0$ is presented in e.g. Ref. 6. The expression for the impedance is

$$Z_{\mathrm{i}} = i\rho_{\mathrm{c}}c_{\mathrm{c}}\frac{H_0^{(2)}(kr_b)}{H_1^{(2)}(kr_b)} \qquad k = \frac{\omega}{c_{\mathrm{c}}}$$

$$r_{\mathrm{b}} = \left|(\mathbf{x}_0 - \mathbf{x}) - \{(\mathbf{x}_0 - \mathbf{x}) \cdot \mathbf{e}_{\mathrm{sa}}\}\mathbf{e}_{\mathrm{sa}}\right| = \frac{\left|(\mathbf{x}_0 - \mathbf{x}) \times \mathbf{e}_{\mathrm{sa}}\right|}{\left|\mathbf{e}_{\mathrm{sa}}\right|}$$

where $H_m^{(2)}(x) = J_m(x) - iY_m(x)$ is the Hankel function of the second kind of order m given in terms of the Bessel functions of order m of the first and second kind, $J_{\mathrm{m}}(x)$ and $Y_{\mathrm{m}}(x)$ respectively. Notice that the source axis vector $\mathbf{e}_{\mathrm{sa}}$ is automatically normalized in this implementation.

Theory for the Interior Impedance Models

In this section:

- Interior Perforated Plate Models

Interior Perforated Plate Models

The transfer impedance of an interior boundary is defined as the ratio of the pressure drop across the boundary to the velocity on the boundary:

$$Z_i = \frac{\Delta p_t}{v_n} = \frac{p_{t,up} - p_{t,down}}{v_n} \tag{2-25}$$

The resulting value of Z_i can be treated as a superposition of several contributions which are derived separately. Note that the theory below is only valid for the perforates with circular-shaped holes. Other types of holes can lead to significantly different results, which will make the models that are considered here inadequate and unreliable.

TRANSFER IMPEDANCE OF A HOLE

Let the z-coordinate axis be directed along the axis of a cylindrical hole of the height t_p (see Figure 2-12). Let the variation of the pressure and the velocity along the z-axis have the following pattern:

$$p(z) = pe^{-ik_c z}, \mathbf{v}(z) = \mathbf{v}e^{-ik_c z} \tag{2-26}$$

The substitution of Equation 2-26 into Equation 2-25 results in

$$Z_i = \frac{p(z+t_p) - p(z)}{v_n(z+t_p/2)} = \frac{p(z)}{v_n(z)}(e^{-ik_c t_p/2} - e^{ik_c t_p/2}) = -2iZ_c \sin\frac{k_c t_p}{2},$$

where Z_c is the characteristic impedance and k_c is the complex wavenumber defined according to the low reduced frequency (LRF) models from the Narrow Region Acoustics for Slits, Circular Ducts, Rectangular Ducts, and Equilateral triangular Ducts. That is,

$$Z_c = \rho c([\gamma-(\gamma-1)\Psi_h]\Psi_v)^{-\frac{1}{2}}, \qquad k_c = \frac{\omega}{c}\left([\gamma-(\gamma-1)\Psi_h]\frac{1}{\Psi_v}\right)^{\frac{1}{2}}.$$

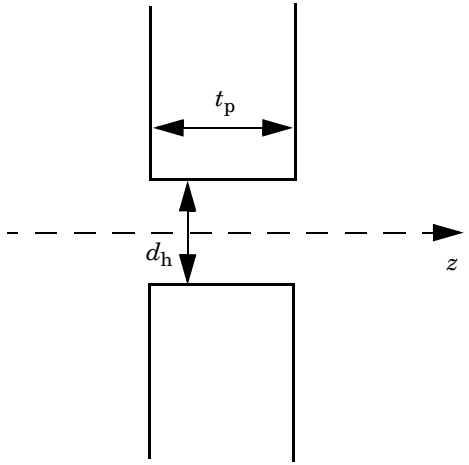

Figure 2-12: To the derivation of the transfer impedance of a one-hole perforate.

The expression for the normalized transfer impedance of an orifice reads

$$z_i = \frac{Z_i}{\rho c} = -\frac{2i\sin\left(\frac{k_c t_p}{2}\right)}{\sqrt{[\gamma-(\gamma-1)\Psi_h]\Psi_v}}. \tag{2-27}$$

For thinner plates ($kt_p \ll 1, k = \omega/c$) the approximation $\sin\left(\frac{k_c t_p}{2}\right) \approx \frac{k_c t_p}{2}$ is valid, which yields

$$z_i = \frac{Z_i}{\rho c} = -\frac{i\omega}{c}\frac{t_p}{\Psi_v}. \tag{2-28}$$

Note that Equation 2-28 coincides with the expression that follows from Crandall's formula for an infinite tube with the circular cross-section (see Ref. 35).
Equation 2-27 accounts for both viscous and thermal effects inside the hole, while the simplified Equation 2-28 contains the viscous part only (thermal effects are negligible for the thin plate limit).

VENA CONTRACTA AND AREA POROSITY

Equation 2-27 and Equation 2-28 are exact if the streamlines of a flow through the hole are parallel to the z-axis throughout the orifice area. In reality, there is a radial component of the flow, which leads to the reduction (contraction) of this area. The minimum area where the streamlines remain parallel to the z-axis is called *vena contracta*. The flow velocity at the *vena contracta* is also different from that of the ideal flow. A coefficient that accounts for these effects is called the discharge coefficient, C_D. The value of the discharge coefficient can be obtained from measurements as function of the plate thickness and the orifice shape and diameter.

Another parameter that used to express the interior impedance of a perforate is the porosity, σ. The holes are usually uniformly distributed over the plate, and the porosity accounts for the distribution as the ratio of the hollow area to the area of the plate. Depending of the pattern the holes are strewn over the plate the porosity is defined as follows

$$\sigma = \frac{\pi d_h^2}{4a^2} \qquad \text{and} \qquad \sigma = \frac{\pi d_h^2}{2\sqrt{3}a^2}$$

for a square and a triangular patterns, respectively. The parameter a determines the hole spacing.

The resulting expression for the interior impedance of a perforate comes out from dividing z_i by the product σC_D:

$$z_{\text{orifice}} = \frac{z_i}{\sigma C_D} . \tag{2-29}$$

Either Equation 2-27 or Equation 2-28 can be used for the substitution for z_i in Equation 2-29. The resulting models will further be referred to as the thick and the thin plate models, respectively.

END CORRECTION AND HOLE-HOLE INTERACTION

The subscript “orifice” in Equation 2-29 means that the expression accounts for the transfer impedance of a perforate caused by the presence of the holes; that is, a piston of fluid of the length t_p. However, the actual mass of fluid affected by an incident wave is larger than that inside the hole. The effective mass of the fluid can be taken into account by the piston which is on each side longer by δ than the initial one (see Figure 2-13). This results in adding two extra terms (for each side of the perforate) of the form Equation 2-28 whit t_p replaced by δ. The choice of Equation 2-28 is due to

the absence of highly conducting walls in the end corrections area.

If two holes a located relatively close to one another, the actual masses of the attached fluid can become overlapped. This makes the total mass less than just the sum of those for the separate hole. In order to take the hole-hole interaction into account, the end

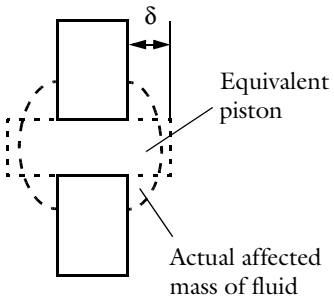

Figure 2-13: To the definition of end corrections.

correction is reduced by a factor f_{int}. The last is a function of the porosity and the most often expressed by the Fok function:

$$\psi(\sigma) = \sum_{n=0}^{N} a_n (\sqrt{\sigma})^n .$$

The end correction is usually considered as function of the hole diameter and can in practice be different for the resistive and the reactive parts of the transfer impedance. For this reason, it is useful to split δ into two parts: δ_{resist} and δ_{react}. The resulting term that accounts for the end correction and the hole-hole interaction reads

$$z_{\text{end}} = -\text{Re}\left(\frac{i\omega}{c\sigma C_{\text{D}}}\frac{2\delta_{\text{resist}}}{\Psi_{\text{v}}}f_{\text{int}}\right) - i\,\text{Im}\left(\frac{i\omega}{c\sigma C_{\text{D}}}\frac{2\delta_{\text{react}}}{\Psi_{\text{v}}}f_{\text{int}}\right) \quad . \qquad (2\text{-}30)$$

Equation 2-30 is only acceptable if the media is the same on both sides of the perforate. If the plate is backed by a porous layer on one side, the values of δ and f_{int} can differ significantly from those on the other side. For example, the interaction between holes is hampered by porous media, which results in neglecting of f_{int} on one side of the plate and leads to the following correction factor: $(\delta + 1)f_{\text{int}}$.

NONLINEAR AND MEAN FLOW EFFECTS

At medium and high sound pressure levels, the displacement of acoustic particles becomes comparable to the diameter of the holes. This causes flow separation and vortex shedding at the entrance and the exit of the hole. This results in the acoustic energy dissipation and increases the acoustic resistance of the perforate Ref. 36. That is, an extra resistance term should be added to the resulting transfer impedance expression.

Different forms of the contribution to the resistance are similar in the following sense:

- the resistance term is directly proportional to the acoustic particle velocity, which makes it nonlinear;
- the term is frequency dependent through the acoustic particle velocity.

The expression incorporated into the Interior Perforated Plate boundary condition reads

$$z_{\text{nl}} = \frac{1-\sigma^2}{\sigma^2 C_{\text{D}}^2} \frac{f_{\text{nl}}}{2c} |v_n| \quad , \tag{2-31}$$

where f_{nl} is a correction factor (equals 1 by default) and v_n is the acoustic particle velocity component normal to the plate. Other expressions for the nonlinear resistance term can be found in Ref. 36.

The presence of a mean flow also results in changing of the transfer impedance. In order to account these (and other possible effects), the Interior Perforated Plate boundary condition feature includes an option to enable user-defined resistance and reactance:

$$z_{\text{user}} = \theta^{(\text{user})} + i\chi^{(\text{user})} . \tag{2-32}$$

Relative to the mean flow, the contribution of a grazing and/or a bias flow to the resistance can be expressed through the flow Mach number as shown in Ref. 37.

Combining the expressions Equation 2-29–Equation 2-32 together yields the full expression for the transfer impedance of a perforate for the thick

$$\frac{Z_{\mathrm{i}}}{\rho c} = -\frac{2i\sin\left(\frac{k_{\mathrm{c}}t_{\mathrm{p}}}{2}\right)}{\sqrt{[\gamma-(\gamma-1)\Psi_{\mathrm{h}}]\Psi_{\mathrm{v}}}} - \mathrm{Re}\left(\frac{i\omega}{c\sigma C_{\mathrm{D}}^{(\mathrm{lin})}}\frac{2\delta_{\mathrm{resist}}}{\Psi_{\mathrm{v}}}f_{\mathrm{int}}\right) - i\,\mathrm{Im}\left(\frac{i\omega}{c\sigma C_{\mathrm{D}}^{(\mathrm{lin})}}\frac{2\delta_{\mathrm{react}}}{\Psi_{\mathrm{v}}}f_{\mathrm{int}}\right)$$

$$+\frac{1-\sigma^2}{(\sigma C_{\mathrm{D}}^{(\mathrm{nl})})^2}\frac{f_{\mathrm{nl}}}{2c}|v_n| + \theta^{(\mathrm{user})} + i\chi^{(\mathrm{user})}$$

and the thin

$$\frac{Z_{\mathrm{i}}}{\rho c} = -\mathrm{Re}\left(\frac{i\omega}{c\sigma C_{\mathrm{D}}^{(\mathrm{lin})}}\frac{t_{\mathrm{p}}+2\delta_{\mathrm{resist}}}{\Psi_{\mathrm{v}}}f_{\mathrm{int}}\right) - i\,\mathrm{Im}\left(\frac{i\omega}{c\sigma C_{\mathrm{D}}^{(\mathrm{lin})}}\frac{t_{\mathrm{p}}+2\delta_{\mathrm{react}}}{\Psi_{\mathrm{v}}}f_{\mathrm{int}}\right)$$

$$+\frac{1-\sigma^2}{(\sigma C_{\mathrm{D}}^{(\mathrm{nl})})^2}\frac{f_{\mathrm{nl}}}{2c}|v_n| + \theta^{(\mathrm{user})} + i\chi^{(\mathrm{user})}$$

plate models. The discharge coefficient C_{D} is taken different for the linear and the nonlinear parts for the sake of flexibility.

Theory for the Equivalent Fluid Models

In this section:

Introduction to the Equivalent Fluid Models

It is possible to define the properties of a fluid in several ways in pressure acoustics. In a Pressure Acoustics domain feature attenuation properties for the bulk fluid may be specified. Acoustic losses in porous materials are modeled by homogenizing the porous matrix and saturating fluid, and are defined by the Poroacoustics domain feature (frequency domain only). The viscous and thermal losses that occur in the acoustic boundary layer can be modeled in a homogenized way using the Narrow Region Acoustics domain feature (frequency domain only). The different ways of defining the properties of a fluid are called *fluid models*. They are also often referred to as *equivalent fluid models*.

Losses and damping occur when acoustic waves propagate in a porous material (where material refers to the homogenization of a fluid and a porous solid) because of bulk viscous and thermal properties, or because of thermal and viscous losses in the acoustic boundary layer at walls in narrow ducts. The purpose of the fluid model is to mimic a special loss behavior by defining a complex-valued density ρ_c and speed of sound c_c. These are often frequency dependent.

In a Pressure Acoustics domain feature, the default Linear elastic fluid model (see Defining a Linear Elastic Fluid Model) enables you to specify a linearly elastic fluid using either the density ρ and speed of sound c, the impedance Z and wave number k, or the equivalent bulk modulus K and the density ρ. When any of these material parameters are complex valued, damping is introduced.

It is always necessary to specify a set of two parameters (for example Z and k) or conditions in order to calculate the complex speed of sound and complex density

needed to specify a fluid model. The choice of parameters typically depend on the application and which equivalent fluid is being modeled. For example:

- It is possible to determine the complex wave number k and impedance Z from directly measuring it in an impedance tube in order to produce curves of the real and imaginary parts (the resistance and reactance, respectively) as functions of frequency. These data can be used directly as input to COMSOL Multiphysics interpolation functions to define k and Z.
- The option to define the equivalent bulk modulus K and density ρ is often used when characterizing the propagation of acoustic waves in a porous material. These parameters may be determined from measurements or by defining an analytical model expression.

The linear elastic fluid model enables the user to enter any desired fluid models with the desired combination of fluid properties. It is the most general fluid model. You may enter any user-defined analytical expressions for your favorite equivalent fluid model or use measurement data to represent the lossy behavior of the fluid.

PROPAGATION IN GENERAL FLUID WITH BULK LOSSES

The options are (see About the Pressure Acoustics Fluid Models and the settings for the Pressure Acoustics node):

- Linear elastic: define density and speed of sound, impedance and wavenumber, or equivalent bulk modulus and density.
- Linear elastic with attenuation: define an attenuation parameter for the fluid.
- Ideal gas is also available but not described here. This fluid model is used to specify the fluid properties by selecting a gas constant type and selecting between entering the heat capacity at constant pressure or the ratio of specific heats. See Defining an Ideal Gas Fluid Model for details.
- Viscous and/or thermally conducting fluids (bulk losses): the losses are due to viscous losses, thermal conduction, or the combined thermal conduction and viscous losses in the bulk of the fluid.

PROPAGATION IN POROUS MATERIALS

A series of fluid models exist for describing the propagation of pressure waves in porous materials. These range from fully empirical models to semi analytical/empirical models with varying degree of complexity. See Poroacoustics and About the Poroacoustics Models for more detail.

PROPAGATION IN NARROW REGIONS, NARROW TUBES, OR WAVEGUIDES

See Narrow Region Acoustics and About the Narrow Region Acoustics Models. The losses are due to absorption/dissipation in the acoustic boundary layer (thermal and viscous losses). The losses are smeared on the domain in a homogenized way.

About the Pressure Acoustics Fluid Models

LINEAR ELASTIC WITH ATTENUATION FLUID MODEL

Use the linear elastic with attenuation fluid model to specify a fluid with attenuation given by an attenuation coefficient α. This results in a complex-valued wave number k. It is used to define the complex speed of sound.

$$c_c = \frac{\omega}{k}$$

There are different attenuation types to select from: attenuation coefficient Np/m, attenuation coefficient dB/m, or attenuation coefficient dB/λ. Select **Attenuation coefficient Np/m** to define an attenuation coefficient in Np/m (nepers per meter), to define:

$$k = \frac{\omega}{c} - i\alpha \qquad \rho_c = \rho\left(\frac{c}{c_c}\right)^2$$

Select **Attenuation coefficient dB/m** to define an attenuation coefficient in dB/m (decibel per meter), to define:

$$k = \frac{\omega}{c} - i\ln(10)\frac{\alpha}{20} \qquad \rho_c = \rho\left(\frac{c}{c_c}\right)^2$$

Select **Attenuation coefficient dB/λ** to define an attenuation coefficient in dB/λ (decibel per wavelength), to define (notice the different definition of the complex density):

$$k = \frac{\omega}{c}\left(1 - i\ln(10)\frac{\alpha}{2\pi \cdot 20}\right) \qquad \rho_c = \rho$$

Defining a Linear Elastic with Attenuation Fluid Model

VISCOUS FLUID MODEL

The viscous model is an equivalent-fluid model that mimics the propagation of sound in a fluid including viscous losses occurring in the bulk of the fluid. The elastic fluid model with viscous losses is defined by:

$$\rho_c = \rho\left(1 + \frac{i\omega b}{\rho c^2}\right)^{-1} \qquad c_c = c\left(1 + \frac{i\omega b}{\rho c^2}\right)^{\frac{1}{2}}$$

$$b = \left(\frac{4}{3}\mu + \mu_B\right)$$

where μ is the dynamic viscosity and μ_B is the bulk viscosity (see Ref. 4 or Ref. 6 chapter 9). This choice is only appropriate for situations where the damping takes place in free space and is not related to interaction between the fluid and a solid skeleton or a wall. These losses, in most fluids, occur over long distances or at very high frequencies.

Defining a Viscous Fluid Model

THERMALLY CONDUCTING FLUID MODEL

The thermally conducting model is an equivalent-fluid model that mimics the propagation of sound in a fluid including losses due to thermal conduction in the bulk. The elastic fluid model with thermal losses is defined by:

$$\rho_c = \rho\left(1 + \frac{i\omega b}{\rho c^2}\right)^{-1} \qquad c_c = c\left(1 + \frac{i\omega b}{\rho c^2}\right)^{\frac{1}{2}}$$

$$b = \left(\frac{(\gamma - 1)k}{C_p}\right)$$

where γ is the ratio of specific heats, C_p is the specific heat at constant pressure, and k is the thermal conductivity (see Ref. 6 chapter 9). This choice is only appropriate for situations where the damping takes place in free space and is not related to interaction between the fluid and a solid skeleton or a wall.

Defining a Thermally Conducting Fluid Model

THERMALLY CONDUCTING AND VISCOUS FLUID MODEL

The thermally conducting and viscous model is an equivalent-fluid model that mimics the propagation of sound in a fluid including losses due to thermal conduction and viscosity in the bulk of the fluid. The elastic fluid model with thermal and viscous losses is defined by:

$$\rho_c = \rho\left(1 + \frac{i\omega b}{\rho c^2}\right)^{-1} \qquad c_c = c\left(1 + \frac{i\omega b}{\rho c^2}\right)^{\frac{1}{2}}$$

$$b = \left(\left(\frac{4}{3}\mu + \mu_B\right) + \frac{(\gamma - 1)k}{C_p}\right)$$

where μ is the dynamic viscosity and μ_B is the bulk viscosity, γ is the ratio of specific heats, C_p is the specific heat at constant pressure, and k is the thermal conductivity (see Ref. 6 chapter 9). This choice is only appropriate for situations where the damping takes place in free space and is not related to interaction between the fluid and a solid skeleton or a wall.

Defining a Thermally Conducting and Viscous Fluid Model

About the Poroacoustics Models

The Poroacoustics node introduces several fluid models for modeling the propagation of acoustic waves in porous materials. After some general remarks about fluid models and the rigid and limp regime approximations, these models are discussed in this section:

- Porous Fluid Models
- Delany-Bazley-Miki
- Zwikker-Kosten
- Attenborough
- Wilson
- Johnson-Champoux-Allard (JCA)
- Johnson-Champoux-Allard-Lafarge (JCAL)
- Johnson-Champoux-Allard-Pride-Lafarge (JCAPL)

- Williams EDFM (equivalent density fluid model)
- Wood (fluid suspension Model)

POROUS FLUID MODELS

The poroacoustics fluid models are equivalent fluid models that mimic the behaviors of a full Poroelastic Material model, which is defined by Biot's theory. A poroacoustics fluid model is based on describing the frequency-dependent effective fluid density $\rho(\omega)$ and the effective fluid bulk modulus $K(\omega)$ of the combined equivalent fluid-solid system (saturating fluid and porous matrix). The description of these models includes the losses associated with the propagation of acoustic waves in porous materials. An *equivalent fluid model* is computationally less demanding than the full poroelastic model. However, it is only physically correct for certain choices of material parameters. Most poroacoustic models are only valid in the rigid or limp porous matrix approximations.

Limp and Rigid Porous Matrix Models

In the *rigid porous matrix* limit, the matrix is assumed to be so stiff that it does not move (sometimes referred to as a *motionless skeleton model*). In this case it is assumed that in Biot's theory $\mathbf{u} = \mathbf{0}$, which yields a wave equation with complex density and bulk modulus. In a rigid porous material the Biot-Willis coefficient is equal to the porosity $\alpha_B = \varepsilon_p$. All the poroacoustic fluid models are based on defining the rigid effective density ρ_{rig} (see Ref. 9).

The *limp porous matrix* limit is the opposite of the rigid assumption. It is used to model materials where the stiffness of solid phases (the porous matrix) is so weak that it cannot support free, structure-borne wave propagation (neither longitudinal nor transverse). That is, the stiffness (in vacuo bulk stiffness) is very small compared to air such that the solid phase motion becomes acoustically significant. If it is light enough, the solid phase still moves because it is "dragged along" by the fluid motion; a limp porous material model is also an equivalent fluid model because it only features a single longitudinal wave type. Typically, the limp assumption can be applied to very light weight fibrous materials (less than 10 kg/m^3) if these are not stiffened by the injection of binder material. In the limp case, it is assumed that the stress tensor vanishes and that in Biot's theory, the Biot-Willis coefficient is $\alpha_B = 1$. The limp density is related to the rigid density by a mixture model (see Ref. 10)

$$\rho_{limp} = \frac{\rho_{rig}\rho_{av} - \rho_f^2}{\rho_{av} + \rho_{rig} - 2\rho_f} \qquad \rho_{av} = \rho_d + \varepsilon_p\rho_f$$

where ρ_f is the fluid density, ρ_d is the drained porous matrix density, ρ_{av} is the average effective density, and ρ_{limp} is the resulting effective limp density.

Fluid Parameters

Defining the equivalent density and bulk modulus results in the usual complex-valued speed of sound and density (the propagation parameters):

$$k_{eq}^2 = \left(\frac{\omega}{c_c}\right)^2 - k_z^2 - \left(\frac{m}{r}\right)^3$$

$$c_c = \sqrt{\frac{K}{\rho_{rig}}} \quad \text{or} \quad c_c = \sqrt{\frac{K}{\rho_{limp}}}$$

$$\rho_c = \rho_{rig} \quad \text{or} \quad \rho_c = \rho_{limp}$$

Ideal Gas and General Fluid Options

We show below that all implemented models of poroacoustics apply for all fluids (gases and liquids) except the purely empirical Delany-Bazley-Miki model which has been obtained from fitting to measurement data.

The applicability of the poroacoustics models to general fluids is not described in the literature, but follows directly from repeating the published derivations without assuming the saturating fluid to be an ideal gas. The only difference is that one needs to use the following general expression for the ratio of specific heats γ (express γ in terms of other material parameters) which is valid for any fluid (for gases γ is typically a known parameter)

$$\gamma - 1 = \frac{\alpha_0^2 T_0}{\rho_0 C_p \beta_0} = \frac{c^2 \alpha_0^2 T_0}{C_p}$$

$$\alpha_0 = -\frac{1}{\rho_0}\frac{d\rho}{dT}\bigg|_p \quad , \tag{2-33}$$

$$\beta_0 = \frac{1}{\rho_0}\frac{d\rho}{dp}\bigg|_T = \frac{1}{K_T}$$

where α_0 is the isobaric thermal expansion coefficient, T_0 is the background quiescent temperature, ρ_0 is the background quiescent density, C_p is the specific heat at constant pressure, β_0 is the isothermal compressibility (inverse isothermal bulk modulus K_T), and c is the speed of sound. This result comes from thermodynamics.

To show the general applicability of the poroacoustics models, we here re-derive the equivalent bulk modulus for the simplest case of the Zwikker-Kosten model for a general fluid. All other implemented models (again except the Delany-Bazley-Miki

model) are extensions of this result (see Ref. 9), and the equivalent bulk moduli can therefore in general be written as

$$K(\omega) = \frac{K_0}{\gamma - (\gamma - 1)\psi(\omega)} \tag{2-34}$$

for any of these models, where the frequency response $\Psi(\omega)$ is specific to each model. Consequently, the following derivation is sufficient to show the general result.

Derivation of the equivalent bulk modulus valid for any fluid in the Zwikker-Kosten theory

In general, the (isentropic) bulk modulus is given by

$$K_0 = \rho \frac{dp}{d\rho}. \tag{2-35}$$

A relationship between the pressure p and the density ρ is needed to define the effective bulk modulus, and the equation of state provides such a relationship. Instead of relying on the ideal gas equation, we here use the following general equation of state

$$\rho = \rho(p, T) \tag{2-36}$$

which merely stipulates that the density ρ is a function of both pressure p and temperature T. The temperature-dependence is non-negligible since we are dealing with small pores, and acoustics in porous materials belong to thermoviscous acoustics (see Thermoviscous Acoustics Interfaces). Assuming the acoustic fields are small harmonic perturbations (denoted by a subscript 1) about a quiescent reference state (denoted by subscript 0)

$$\begin{aligned} \rho &= \rho_0 + \rho_1 e^{i\omega t} \\ p &= p_0 + p_1 e^{i\omega t} \\ T &= T_0 + T_1 e^{i\omega t} \end{aligned} \tag{2-37}$$

and Taylor-expanding the general equation of state Equation 2-36 to first order, we find

$$\rho_1 = \rho_0(\beta_T p_1 - \alpha_0 T_1) \tag{2-38}$$

where β_T is the isothermal compressibility.

In order to calculate the effective bulk modulus from Equation 2-35 using the equation of state Equation 2-38, a relationship between T_1 and p_1 is derived from the

linearized energy equation (see Theory Background for the Thermoviscous Acoustics Branch)

$$0 = \left(\nabla^2 - \frac{i\omega}{D_{th}}\right) T_1 + \frac{\alpha_0 T_0}{k} i\omega p_1 + \frac{Q}{k}$$
$$D_{th} = \frac{k}{\rho_0 C_p} \tag{2-39}$$

where D_{th} is the thermal diffusivity, k is the thermal conductivity, and Q is a volumetric heat source. To proceed with the solution of this equation, it is generally assumed that (*i*) the pore can be modeled as a hollow cylinder, (*ii*) the contributions from radial variations dominate over axial variations so it suffices to solve the radial problem while assuming no axial variations Ref. 9, and (*iii*) the pore radius is much smaller than the wavelength, whereby the pressure does not change noticeably across the radius. By these assumptions the pressure p_1 can be treated as a constant in Equation 2-39 and the equation has only radial variations:

$$\nabla^2 T_1 \rightarrow \frac{1}{r}\frac{\partial}{\partial r} T_1 + \frac{\partial^2}{\partial r^2} T_1$$

The cross-sectional average solution under these assumptions, and with the boundary condition of zero acoustic temperature T_1 on the pore walls at $r = R$, is

$$\bar{T}_1(r) = \left[\frac{\alpha_0 T_0}{\rho_0 C_p} p_1 + \frac{Q}{\rho_0 C_p}\frac{1}{i\omega}\right]\left[1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-i}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-i})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-i})}\right]. \tag{2-40}$$

Here

$$\mathrm{Wo} = \sqrt{\frac{R^2 \omega \rho_0}{\mu}} \qquad \mathrm{Pr} = \frac{\mu}{\rho D_{th}} \tag{2-41}$$

where Wo is the Womersley number (see the note below), and Pr is the Prandtl number, measuring the relative width of the viscous boundary layer thickness to the thermal boundary layer thickness.

With the solution from Equation 2-40, the pressure can now be expressed as a function of the density using the first-order equation of state Equation 2-38:

$$p_1(\rho_1) = \frac{\rho_1 + \frac{Q\alpha_0}{C_p}\left[1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}\right]}{\rho_0\left(\beta_T - \frac{\alpha_0^2 T_0}{\rho_0 C_p}\left[1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}\right]\right)} \tag{2-42}$$

From this we obtain the following expression for the equivalent bulk modulus (see Equation 2-35)

$$K(\omega) = \rho_0\frac{\mathrm{d}p_1}{\mathrm{d}\rho_1} = \frac{K_0}{\gamma - (\gamma - 1)\left[1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}\right]} \tag{2-43}$$

where the ratio of specific heats γ is defined in Equation 2-33 for a general fluid and K_0 is the isentropic bulk modulus of the fluid. This formula Equation 2-43 is mathematically equivalent to the normal Zwikker-Kosten formula for the equivalent bulk modulus (see Ref. 15), but it has been *derived for a general fluid and not just an ideal gas.*

We emphasize that from this result for a general fluid follows the general formula Equation 2-34, in this particular case with

$$\psi(\omega) = 1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})} \tag{2-44}$$

with the Womersley number Wo containing the frequency dependence.

The *Womersley number* Wo measures the influence of viscous effects relative to the oscillation frequency ω. For $\mathrm{Wo} \ll 1$ viscosity dominates and the velocity profile is the well-known Poiseuille parabola, while in the Helmholtz regime for $\mathrm{Wo} \gg 1$ the velocity profile is plug-like with a very small boundary layer close to the walls.

DELANY-BAZLEY-MIKI

The Delany-Bazley-Miki model is an equivalent fluid model that mimics the bulk losses in certain porous/fibrous materials. The model represents a porous medium with the following complex propagation constants:

$$k_c = \frac{\omega}{c}[1 + C_1 X^{-C_2} - iC_3 X^{-C_4}]$$

$$Z_c = \rho_f c[1 + C_5 X^{-C_6} - iC_7 X^{-C_8}]$$

$$X = \frac{f \cdot \rho_f}{R_f}$$

where ρ_f is the fluid density, f is the frequency, and R_f is the flow resistivity. Several predefined sets of the coefficients C_i exist. They are the classic Delany-Bazley model, the Miki model (see Ref. 9, section 2.5 and), the Qunli, several variants of the Mechel model for different configurations, the Komatsu model, and a so-called Modified Champoux and Allard model. These are all empirical models based on fitting the two complex functions to measured data for the complex wave number k_c and complex specific acoustic impedance Z_c. All the models are applicable for materials with a porosity ε_p close to 1. The applicability of the different model parameters is listed in Table 2-10. See also Ref. 23 for further details.

TABLE 2-10: DELANY-BAZLEY-MIKI MODEL OPTIONS AND APPLICABILITY

MODEL NAME	APPLICABILITY
Delany-Bazley	Glass and rock wool with: $0.01 \le X \le 1$ $10^3 \le R_f \le 50 \cdot 10^3$ Pasm^{-2}
Miki	Glass and rock wool with: $0.01 \le X \le 1$ $10^3 \le R_f \le 50 \cdot 10^3$ Pasm^{-2} The validity of the model using the Miki parameters is not well-established for $X < 0.01$, but the model is slightly better behaved mathematically below this limit using the Miki parameters rather than the Delany-Bazley parameters, see Ref. 22.
Qunli	Porous plastic and open foams: $200 \le f \le 2000$ Hz $3 \cdot 10^3 \le R_f \le 24 \cdot 10^3$ Pasm^{-2}

TABLE 2-10: DELANY-BAZLEY-MIKI MODEL OPTIONS AND APPLICABILITY

MODEL NAME	APPLICABILITY
Mechel, glass fiber, low X	Glass fiber: $X \leq 0.025$
Mechel, glass fiber, high X	Glass fiber: $X \geq 0.025$
Mechel, rock fiber, low X	Rock fiber: $X \leq 0.025$
Mechel, rock fiber, high X	Rock fiber: $X \geq 0.025$
Komatsu	Glass and rock wool: $6 \cdot 10^3 \leq R_f \leq 73 \cdot 10^3$ Pasm^{-2}
Modified Champoux and Allard	$45 \leq f \leq 11 \cdot 10^3$ Hz

ZWIKKER-KOSTEN

Zwikker-Kosten is one of the earliest equivalent fluid models for porous materials (Ref. 15). It is a rigid frame model defined by the complex (rigid) density

$$\rho_{\text{rig}} = \frac{\rho_f}{\varepsilon_p} \frac{1}{1 - \frac{2}{\text{Wo}\sqrt{-i}} \frac{J_1(\text{Wo}\sqrt{-i})}{J_0(\text{Wo}\sqrt{-i})}} \qquad \text{Wo} = \sqrt{\frac{\omega \rho_f H_r^2}{\mu}} = \sqrt{2}\frac{H_r}{\delta_v}$$

where H_r is the hydraulic radius of the pores (for straight cylindrical pores $H_r = a$ = radius) and Wo is the Womersley number (see Equation 2-41 and the note below). Wo is related to the ratio between viscous penetration depth δ_v and the hydraulic radius. δ_v gives the scale of the viscous boundary layer thickness (see Theory Background for the Thermoviscous Acoustics Branch for details). The bulk modulus is given by

$$K = \frac{\gamma p_A}{\varepsilon_p} \frac{1}{1 + (\gamma - 1)\frac{2}{\text{Wo}\sqrt{-i\text{Pr}}} \frac{J_1(\text{Wo}\sqrt{-i\text{Pr}})}{J_0(\text{Wo}\sqrt{-i\text{Pr}})}} \qquad \text{Pr} = \frac{C_p \mu}{k}$$

where p_A denotes the ambient pressure, ρ_f the fluid density, γ the ratio of specific heat, Pr the Prandtl number, μ the dynamic viscosity, C_p the heat capacity at constant pressure, and k the coefficient of thermal conduction. J_0 and J_1 are Bessel functions of the first kind. The factor γp_A is the isentropic bulk modulus ($K_0 = \gamma p_A$). The free parameters of the pores are the porosity ε_p and the hydraulic radius H_r.

ATTENBOROUGH

The Attenborough model is also based on the cylindrical-like pore assumption. It is a so-called four parameter semi-empirical model. The model is an extension of the Zwikker-Kosten model and adds two more input parameters. It accounts for the tortuosity (high frequency limit) τ_∞, which is related to the orientation of the pores relative to the propagation direction. The hydraulic diameter of the pores is replaced by an expression that includes the flow resistivity R_f, and a fitting parameter b, (this parameter is related to the anisotropy of the pores). See Ref. 9 and Ref. 16. The equivalent density and bulk modulus are defined as

$$\rho_{rig} = \frac{\rho_f}{\varepsilon_p} \frac{\tau_\infty}{1 - \frac{2}{s'\sqrt{-i}} \frac{J_1(s'\sqrt{-i})}{J_0(s'\sqrt{-i})}}$$

and

$$K = \frac{\gamma p_A}{\varepsilon_p} \frac{1}{1 + (\gamma - 1) \frac{2}{s'\sqrt{\mathrm{Pr}}\sqrt{-i}} \frac{J_1(s'\sqrt{\mathrm{Pr}}\sqrt{-i})}{J_0(s'\sqrt{\mathrm{Pr}}\sqrt{-i})}} \qquad \mathrm{Pr} = \frac{C_p \mu}{k}$$

where p_A denotes the ambient pressure, ρ_f the fluid density, γ the ratio of specific heat, Pr the Prandtl number, μ the dynamic viscosity, C_p the heat capacity at constant pressure, and k the coefficient of thermal conduction. J_0 and J_1 are Bessel functions of the first kind. The variable s' (anisotropy factor) is derived from other material parameters and is related to the Womersley number (see Equation 2-41):

$$s' = b\sqrt{\frac{8\omega\rho_f\tau_\infty}{\varepsilon_p R_f}}$$

Here, ω denotes the angular frequency. The four parameters needed (when the fluid is air at room temperature) are the porosity ε_p, the tortuosity τ_∞, flow resistivity R_f, and the fitting parameter b (dimensionless, close to 1). The fitting parameter b is tabulated for certain well-defined pore cross-sections in Table 2-11.

TABLE 2-11: FITTING FACTOR b FOR DIFFERENT CROSS-SECTIONAL GEOMETRIES (SEE Ref. 9)

CROSS-SECTIONAL SHAPE	b
Circle	1
Square	1.07

TABLE 2-11: FITTING FACTOR b FOR DIFFERENT CROSS-SECTIONAL GEOMETRIES (SEE Ref. 9)

CROSS-SECTIONAL SHAPE	b
Equilateral triangle	1.11
Rectangular slit	0.81

In this way, the hydraulic radius of the Attenborough model is formulated in terms of measurable intrinsic properties of the porous material which alleviates the need to know the pore radii.

The viscous characteristic length of the model L_v can also be defined by

$$L_v = b\sqrt{\frac{8\mu\tau_\infty}{\varepsilon_p R_f}} = s'\sqrt{\frac{\mu}{\omega\rho_f}} = s'\frac{\delta_v}{\sqrt{2}}$$

with δ_v the viscous penetration depth. This length is related to the pores' circular cross section radius R (for a cylinder) and the thickness of the viscous boundary layer. The parameter $b = 1/s$, where s is the viscous characteristic length parameter. See the following models:

- Johnson-Champoux-Allard (JCA)
- Johnson-Champoux-Allard-Lafarge (JCAL)
- Johnson-Champoux-Allard-Pride-Lafarge (JCAPL)).

Note that the tortuosity is related to the angle θ between the cylindrical pores and the direction of propagation of the wave, by

$$\tau_\infty = \frac{1}{\cos^2(\theta)}$$

WILSON

The Wilson model is a generalization of the analytical models for porous materials with constant cross section and parallel pores. This model is intended to match the middle frequency behavior of a porous material (see Ref. 9, Ref. 17, and Ref. 18). It is not a good model for ω tending to 0 or infinity. The equivalent density and bulk modulus are given by

$$\rho_{rig} = \rho_\infty \frac{(1 + i\omega\tau_{vor})^{1/2}}{(1 + i\omega\tau_{vor})^{1/2} - 1}$$

$$K = K_\infty \frac{(1 + i\omega\tau_{ent})^{1/2}}{(1 + i\omega\tau_{ent})^{1/2} + \gamma - 1}$$

where τ_{vor} denotes the vorticity-mode relaxation time, τ_{ent} the entropy-mode relaxation time, ρ_∞ the infinity frequency limit for the density, K_∞ the infinity frequency limit for the bulk modulus, and γ is the ratio of specific heats. These are the four free parameters. With appropriate choices for the relaxation parameters, the Wilson model can be fitted to mimic all the models described here. For example, setting $\tau_{vor} = 2.54/R_f$ and $\tau_{ent} = 3.75/R_f$, the equations mimic the Delany-Bazley model (see Cox and D'Antonio Sec. 5.4.4, Ref. 18).

Approximate expressions based on non-acoustic parameters (properties of the porous matrix) also exist for the relaxation times:

$$\tau_{vor} = \frac{\rho_f l^2}{2\mu} \approx \frac{2\rho_f \tau_\infty}{\varepsilon_p R_f} \qquad \tau_{ent} \approx Pr\,\tau_{vor}$$

$$\rho_\infty \approx \frac{\rho_f \tau_\infty}{\varepsilon_p} \qquad K_\infty \approx \frac{\gamma p_A}{\varepsilon_p}$$

Here τ_∞ denotes the (high frequency limit) tortuosity (it is called q^2 in the Wilson's paper Ref. 17), ε_p the porosity, ρ_f the fluid density, l a characteristic pore dimension, and Pr is the Prandtl number.

JOHNSON-CHAMPOUX-ALLARD (JCA)

The Johnson-Champoux-Allard (JCA) porous matrix model is defined by the following equivalent rigid densities $\rho_{rig}(\omega)$ and equivalent bulk modulus $K(\omega)$:

$$\rho_{rig} = \frac{\tau_\infty \rho_f}{\varepsilon_p}\left[1 + \frac{R_f \varepsilon_p}{i\omega\rho_f\tau_\infty}\sqrt{1 + \frac{4i\omega\tau_\infty^2\mu\rho_f}{R_f^2 L_v^2 \varepsilon_p^{\,2}}}\right]$$

$$K = \frac{\gamma p_A}{\varepsilon_p}\left[\gamma - (\gamma - 1)\left(1 + \frac{8\mu}{i\omega L_{th}^2 Pr\rho_f}\sqrt{1 + \frac{i\omega L_{th}^2 Pr\rho_f}{16\mu}}\right)^{-1}\right]^{-1}$$

Here τ_∞ is the tortuosity factor (high frequency limit), ρ_f is the fluid density, ε_p is the porosity, R_f is the flow resistivity, μ is the dynamic viscosity, p_A is the quiescent

pressure, γ is the ratio of specific heats, L_v is the viscous characteristic length, L_{th} is the thermal characteristic length, and Pr is the Prandtl number. The viscous characteristic length is related to the viscous characteristic length parameter s by

$$L_v = \frac{1}{s}\sqrt{\frac{8\mu\tau_\infty}{\varepsilon_p R_f}}$$

Here s is a pore geometry dependent factor between 0.3 and 3.0 (for example 1 for circular pores, 0.78 for slits)

The expression given for the geometry dependent pore factor s is only valid for values of s close to 1. If this is not the case, enter the viscous characteristic length L_v directly into the model (the default selection).

The viscous L_v and thermal L_{th} characteristic lengths are also sometimes denoted by Λ and Λ', respectively.

Porous Absorber: Application Library path **Acoustics_Module/Building_and_Room_Acoustics/porous_absorber**

JOHNSON-CHAMPOUX-ALLARD-LAFARGE (JCAL)

The Johnson-Champoux-Allard-Lafarge (JCAL) model introduces corrections to the bulk modulus thermal behavior at low frequencies that is not captured by the JCA model (see Ref. 13). The equivalent density is the same as in the JCA model. The correction is to the bulk modulus and is given by

$$K = \frac{\gamma p_A}{\varepsilon_p}\left[\gamma-(\gamma-1)\left(1+\frac{\varepsilon_p\mu}{i\omega k'_0 \mathrm{Pr}\rho_f}\sqrt{1+\frac{4i\omega(k'_0)^2\mathrm{Pr}\rho_f}{\mu L_{th}^2\varepsilon_p^2}}\right)\right]^{-1}$$

with the introduction of the new parameter, k'_0, which is the static thermal permeability (SI unit: m^2). For measurements and details of this parameter see Ref. 11 and Ref. 12 for examples.

JOHNSON-CHAMPOUX-ALLARD-PRIDE-LAFARGE (JCAPL)

The Johnson-Champoux-Allard-Pride-Lafarge (JCAPL) model further extends the JCAL models by introducing a static viscous τ_0 and thermal τ'_0 tortuosity, which both

introduce low frequency corrections to the JCAL and JCA models. See Ref. 14 and Ref. 9.

This model has the complex rigid density given by:

$$\rho_{rig} = \frac{\rho_f \tilde{\tau}(\omega)}{\varepsilon_p}$$

$$\tilde{\tau}(\omega) = \tau_\infty \left[1 + \frac{1}{i\bar{\omega}} \tilde{F}(\omega)\right] \qquad \tilde{F}(\omega) = 1 - P + P\sqrt{1 + \frac{M}{2P^2} i\bar{\omega}}$$

$$i\bar{\omega} = \frac{i\omega\rho_f k_0 \tau_\infty}{\mu\varepsilon_p} \qquad M = \frac{8k_0\tau_\infty}{\varepsilon_p L_v^2}$$

$$P = \frac{M}{4(\tau_0/\tau_\infty - 1)} = \frac{2k_0\tau_\infty^2}{\varepsilon_p L_v^2(\tau_0 - \tau_\infty)}$$

where the new parameter is the static viscous tortuosity τ_0 (dimensionless). The viscous permeability is defined as $k_0 = \mu/R_f$ (SI unit: m^2).

In Ref. 9 (equation 5.32), P is called b, k_0 is called q_0, and Pr is called B^2.

The complex bulk modulus K is given by:

$$K = \frac{\gamma p_A}{\varepsilon_p} \frac{1}{\tilde{\beta}(\omega)}$$

$$\tilde{\beta}(\omega) = \gamma - (\gamma - 1)\left[1 + \frac{1}{i\bar{\omega}'} \tilde{F}'(\omega)\right]^{-1} = \gamma - (\gamma - 1)\tilde{\tau}'(\omega)^{-1}$$

$$\tilde{F}'(\omega) = 1 - P' + P'\sqrt{1 + \frac{M'}{2P'^2} i\bar{\omega}'} \qquad i\bar{\omega}' = \frac{i\omega\rho_f \mathrm{Pr} k'_0}{\mu\varepsilon_p}$$

$$M' = \frac{8k'_0}{\varepsilon_p L_{th}^2} \qquad P' = \frac{M'}{4(\tau'_0 - 1)}$$

where the new parameter is the static thermal tortuosity τ'_0 (dimensionless).

In Ref. 9 (equation 5.35), $\tilde{\tau}'(\omega)$ is called $\alpha'(\omega)$, and $P' = 1$.

The JCA model is recovered by setting $M' = P = P' = 1$ and the JCAL is recovered by setting $P = P' = 1$.

WILLIAMS EDFM (EQUIVALENT DENSITY FLUID MODEL)

The Williams EDFM model is a so-called equivalent density fluid model. The model is derived under the assumption that the bulk and shear moduli of the frame of the porous material are negligible, see Ref. 24 for further details. This is why the model pertains to sediments. The effective bulk modulus K_{eff} and an effective density ρ_{eff} defined my the model are given by

$$K_{\text{eff}} = \left(\frac{1-\varepsilon_p}{K_{\text{gr}}} + \frac{\varepsilon_p}{K_f}\right)^{-1}$$

$$\rho_{\text{eff}}(\omega) = \rho_f \left(\frac{\tau_\infty(1-\varepsilon_p)\rho_{\text{gr}} + \varepsilon_p(\tau_\infty - 1)\rho_f + \dfrac{i\varepsilon_p \rho_{\text{mix}} F(\text{Wo})\mu}{\rho_f \omega \kappa}}{\varepsilon_p(1-\varepsilon_p)\rho_{\text{gr}} + (\tau_\infty - 2\varepsilon_p + \varepsilon_p^2)\rho_f + \dfrac{i\varepsilon_p F(\text{Wo})\mu}{\omega\kappa}} \right)$$

$$\rho_{\text{mix}} = \varepsilon_p \rho_f + (1-\varepsilon_p)\rho_{\text{gr}}$$

where the subscript "gr" pertains to the grains and the subscript "f" to the saturating fluid. The porosity is denoted ε_p, the observed mixture density ρ_{mix}, the tortuosity τ_∞, the dynamic viscosity μ, the angular frequency ω, and the permeability of the sediments κ. The function F is a function of the Womersley number Wo defined as

$$F(\text{Wo}) = \frac{\frac{\text{Wo}}{4}T(\text{Wo})}{1 - \frac{2i}{\text{Wo}}T(\text{Wo})} \qquad T(\text{Wo}) = -\frac{\sqrt{i}J_1(\sqrt{i}\text{Wo})}{J_0(\sqrt{i}\text{Wo})}$$

$$\text{Wo} = \sqrt{\frac{\rho_f \omega H_r^2}{\mu}}$$

where $J_n(x)$ is the Bessel function of the first kind of order n and H_r is the hydraulic radius. Based on these the effective complex speed of sound and density are given as

$$c_c = \sqrt{\frac{K_{\text{eff}}}{\rho_{\text{eff}}(\omega)}} \qquad \rho_c = \rho_{\text{eff}}(\omega)$$

WOOD (FLUID SUSPENSION MODEL)

In a fluid mixture or a fluid suspension (solid inclusions completely surrounded by fluid), the Wood formula can be used to determine the effective speed of sound for the mixture. It is determined by calculating the effective bulk modulus of the suspension and the volume average density. As the Williams EDFM, this model gives effective values for the mixture. This result is exact for low frequencies (when the wavelength is much larger than the size of the inclusions) since the effective bulk modulus in the quasi-static limit. The Wood model defines

$$\frac{1}{K_{\text{eff}}} = \frac{\theta_{\text{f}}}{K_{\text{f}}} + \sum_{i}^{N} \frac{\theta_{\text{i}}}{K_{\text{i}}}$$

$$\rho_{\text{eff}} = \theta_{\text{f}}\rho_{\text{f}} + \sum_{i}^{N} \theta_{\text{i}}\rho_{\text{i}}$$

where θ_{f}, K_{f}, and ρ_{f}, are the fluid's volume fraction, adiabatic bulk modulus, and density, respectively; and θ_{i}, K_{i}, and ρ_{i}, are the inclusion's volume fractions, adiabatic bulk moduli, and densities, respectively. And again the complex speed of sound and density are defined as

$$c_{\text{c}} = \sqrt{\frac{K_{\text{eff}}}{\rho_{\text{eff}}}} \qquad \rho_{\text{c}} = \rho_{\text{eff}}$$

About the Narrow Region Acoustics Models

The Narrow Region Acoustics fluid models are used to mimic the thermal and viscous losses that exist in narrow tubes where the tube cross-section length-scale is comparable to the thermal and viscous boundary layer thickness (boundary-layer absorption). It is essential to include these losses in order to get correct results.

These models are commonly used in situations where solving a full detailed thermoviscous acoustic model is computationally costly; for example when analyzing long narrow ducts/tubes of constant cross section. Here it is possible to add or smear the losses associated with the boundary layer onto the bulk of the fluid: an equivalent fluid model. For many geometries, analytical expressions exist for the losses associated

with the acoustic boundary layers. The models can be applied under different assumptions. The models and assumptions are discussed in this section.

	In more complex geometries where thermal and viscous losses are important, see The Thermoviscous Acoustics, Frequency Domain Interface, which is more fundamental and detailed.

	Narrow Region Acoustics

Several fluid models exist:

- The *wide duct approximation* can be used for any duct cross section in the limit where the duct width is significantly larger than the acoustic boundary layer thickness. See Wide Ducts.
- The *very narrow circular ducts (isothermal)* can only be used when the duct width is so small that isothermal conditions apply. This is when the duct width is much smaller than the acoustic thermal boundary layer thickness. See Very Narrow Circular Ducts (Isothermal).
- The *slit*, *circular duct*, *rectangular duct*, and *equilateral triangular duct* models are based on an analytical solution of the thermoviscous acoustic equations in the limit where the acoustic wavelength is much larger than both the duct cross section (below the cut-off frequency) and the boundary layer thickness. This is the case in most engineering applications. See Slits, Circular Ducts, Rectangular Ducts, and Equilateral triangular Ducts.
- Finally, selecting the *user defined* option enables you to enter expressions for the complex wavenumber and the complex acoustic impedance. These may be analytical expressions or values extracted from a detailed mode analysis study using the full thermoviscous acoustics model.

WIDE DUCTS

For a relatively wide duct, the losses introduced in the acoustic boundary layer may be studied by adding these as an effective wall shear force. This approach is used in Blackstock (Ref. 6) and results in equivalent fluid complex wave number k_c defined by

$$k_c = \frac{\omega}{c}\frac{1}{\sqrt{1 - B\sqrt{\frac{\pi}{i\omega}}}} \cong \frac{\omega}{c}\left(1 + \frac{B}{2}\sqrt{\frac{\pi}{i\omega}}\right)$$
$$B = \frac{4}{H_d}\sqrt{\frac{\mu}{\pi\rho}}\left(1 + \frac{\gamma - 1}{\sqrt{\mathrm{Pr}}}\right) \qquad \mathrm{Pr} = \frac{\mu C_p}{k} \qquad H_d = 4\frac{S}{C} \tag{2-45}$$

where H_d is the hydraulic diameter of the duct, S is the duct cross-section area, C is the duct circumference, μ is the dynamic viscosity, ρ is the fluid density, γ is the ratio of specific heats, C_p is the specific heat at constant pressure, k is the fluid thermal conductivity, and Pr is the Prandtl number. For a cylindrical duct, $H_d = 2a$ where a is the radius.

The approximation in Equation 2-45 is only valid for systems where the effective radius $H_d/2$ is larger than the boundary layer, but not so small that mainstream thermal and viscous losses are important. Thus requiring

$$d_{visc} < \frac{H_d}{2} < \frac{c^2}{\omega^2}\frac{1}{d_{visc}} \qquad d_{visc} = \sqrt{\frac{2\mu}{\omega\rho}}$$

where d_{visc} is the characteristic thickness of the viscous boundary layer (the viscous penetration depth), c is the speed of sound, and ω is the angular frequency. The complex wave number is related to the complex density and speed of sound by the equation (assuming a real valued bulk modulus)

$$c_c = \frac{\omega}{k_c} \qquad \rho_c = \rho\left(\frac{c}{c_c}\right)^2$$

VERY NARROW CIRCULAR DUCTS (ISOTHERMAL)

In the other limit where the duct diameter is sufficiently small or the frequency sufficiently low, the thermal boundary layer thickness becomes much larger than the duct cross section a. This is the case when

$$a \ll d_{therm} \Rightarrow \frac{\omega\rho a^2 C_p}{2k} \ll 1$$

where d_{therm} is the characteristic thickness of the thermal boundary layer (thermal penetration depth), ρ is the density, C_p is the heat capacity at constant pressure, and k is the fluid thermal conductivity. In this case see Pierce (Ref. 5); the system may be seen as isothermal and the acoustic temperature variation is zero everywhere in the duct $T = 0$. The fluid complex wave number k_c is then defined by

$$k_c = \frac{\omega}{c_T}\left(\frac{4\mu}{\rho\omega a^2}\right)^{\frac{1}{2}} - i\left(\frac{4\mu\omega}{\rho c_T^2 a^2}\right)^{\frac{1}{2}} \qquad c_T = \frac{c}{\sqrt{\gamma}} \tag{2-46}$$

where c_T is the isothermal speed of sound, a is the duct radius, μ is the dynamic viscosity, and ω is the angular frequency. The theory is derived for ducts of circular cross section — the model is therefore only applicable for systems with small variations away from a circular cross section. The complex wave number is related to the complex density and speed of sound by the equation (here the bulk modulus is defined in the isothermal limit)

$$c_c = \frac{\omega}{k_c} \qquad \rho_c = \rho\left(\frac{c_T}{c_c}\right)^2$$

SLITS, CIRCULAR DUCTS, RECTANGULAR DUCTS, AND EQUILATERAL TRIANGULAR DUCTS

The slit, circular duct, rectangular duct, and equilateral triangular duct models are based on the so-called *low reduced frequency* (LRF) model that describes the propagation of acoustic waves in small waveguides (ducts and slits) including thermal and viscous losses. Details about these models are in Ref. 19, Ref. 20, and Ref. 21. The models cover the range from fully isothermal conditions (very low frequencies or very narrow tubes) to large ducts where the boundary layer only represents a fraction of the duct size. The models apply as long as the cross section of the duct is much smaller than the acoustic wavelength (the model is below the cut-off frequency).

In a narrow waveguide the complex wave number, k_c, and complex specific acoustic impedance, Z_c, are given by

$$k_c^2 = k_0^2\left(\frac{\gamma - (\gamma-1)\Psi_h}{\Psi_v}\right) \qquad k_0 = \frac{\omega}{c}$$

$$Z_c^2 = \frac{Z_0^2}{\Psi_v(\gamma - (\gamma-1)\Psi_h)} \qquad Z_0 = \rho c$$

where Ψ_v and Ψ_h are geometry and material-dependent functions (specified below) and γ is the ratio of specific heats. The fluid density ρ, the speed of sound is c, and the angular frequency ω define the free space wave number k_0 and the specific acoustic impedance Z_0. The subscripts "v" and "h" stand for viscous and thermal (heat) fields, respectively. Once these are known, the complex speed of sound and complex density are given by

$$c_c = \frac{\omega}{k_c}$$

$$\rho_c = \frac{k_c Z_c}{\omega} = \frac{k_0 Z_0}{\omega}\frac{1}{\Psi_v} = \frac{\rho}{\Psi_v}$$

The values of the Ψ_j functions can be derived by solving the full set of linearized Navier-Stokes equations (the equations solved by the thermoviscous acoustics interfaces, see Theory Background for the Thermoviscous Acoustics Branch) by splitting these into an isentropic (adiabatic), a viscous, and a thermal part. Doing this introduces the viscous and the thermal wave numbers for the system

$$k_v^2 = -i\omega\frac{\rho}{\mu} \qquad k_h^2 = -i\omega\frac{\rho C_p}{k}$$

The equations may be solved analytically under the following assumptions used for the LRF models:

- The viscous and thermal wave numbers must be much larger than the acoustic wave number (the acoustic wavelength must be much larger than the boundary layer thickness):

$$\left|\frac{k_j}{k_0}\right| \gg 1 \quad \Rightarrow \quad \lambda_0 \gg \delta_j$$

- The cross section of the waveguide must be much smaller than the acoustic wavelength (the model is below the cut-off frequency).
- The cross section of the waveguide must be constant or at most slowly varying in the propagation direction.
- The length of the waveguide in the propagation direction should be larger than the boundary layer thickness.

The resulting analytical expressions, for the viscous and thermal Ψ functions, are for the given geometry (these results are reviewed in Ref. 19):

- Slit of height h

$$\Psi_j = 1 - \frac{\tan(k_j h/2)}{k_j h/2}$$

- Circular duct of radius a

$$\Psi_i = -\frac{J_2(k_j a)}{J_0(k_j a)}$$

- Rectangular duct of side lengths W and H

$$\Psi_i = k_j^2 \sum_{m=0}^{\infty} \left[(\alpha_m m')^{-2} \left(1 - \frac{\tan(\alpha_m W/2)}{\alpha_m W/2}\right) + (\beta_m m')^{-2} \left(1 - \frac{\tan(\beta_m H/2)}{\beta_m H/2}\right) \right]$$

$$m' = (m + 1/2)\pi$$

$$\alpha_m = \sqrt{k_j^2 - \left(\frac{2m'}{W}\right)} \qquad \beta_m = \sqrt{k_j^2 - \left(\frac{2m'}{H}\right)}$$

- Equilateral triangular duct of side length d

$$\Psi_i = 1 - 3\frac{\tan\left(\frac{3}{2}(k_j l)\right) - \frac{3}{2}(k_j l)}{\left(\frac{3}{2}(k_j l)\right)^2 \tan\left(\frac{3}{2}(k_j l)\right)} \qquad l = \frac{d}{2\sqrt{3}}$$

User Defined

The user defined option in the Narrow Region Acoustics domain feature can be used to define an LRF model for a waveguide of an arbitrary cross sections. This can be a user defined analytical expression or values derived from a mode analysis study.

To determine the complex propagation constants for a waveguide, set up a mode analysis study on the cross-section geometry of the guide. Use The Thermoviscous Acoustics, Frequency Domain Interface because it solves the problem exactly. The complex wave number is then given by the first mode solved for in the system (`ta.kz`) while the complex acoustic impedance is given by the ratio of the integrated pressure over the integrated out-of-plane velocity (integrated over the cross section of the waveguide).

Theory for the Perfectly Matched Layers in the Time Domain

In this section:

Introduction to Perfectly Matched Layers

The concept of a perfectly matched layer (PML) as an absorbing boundary condition was introduced by Bérenger (Ref. 38) with regard to the system of Maxwell's equations. The PML provided absorption of propagating waves without introducing reflections from the interface between the PML and the physical domain. This made the PML technique attractive for treating open domain problems for acoustic, elastic, and electromagnetic wave propagation.

Bérenger's PML formulation is usually referred to as the split PML, because the unknowns are split into a sum of non-physical components in PML domains. Another formulation that does not require splitting the variables is based on the coordinate transformation in PML domains, where the real coordinate is mapped onto the complex plane:

$$\tilde{x} = f(x) \in C\,. \tag{2-47}$$

For example, the mapping Equation 2-47 will transform the 1D Helmholtz equation as follows:

$$\frac{(i\omega)^2}{c^2}p - \frac{1}{f'(x)}\frac{\partial}{\partial x}\left(\frac{1}{f'(x)}\frac{\partial p}{\partial x}\right) = 0\,. \tag{2-48}$$

Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*.

For the sake of brevity, consider the 1D Equation 2-48. In order to derive the PML formulation in the time domain, the following steps are taken (Ref. 39). First, consider a special form of the mapping Equation 2-47:

$$\tilde{x} = x + \frac{1}{i\omega}\int_{x_0}^{x} \sigma(\xi)d\xi\,, \tag{2-49}$$

which yields $d\tilde{x} = f'(x)dx = (1 + \sigma(x)/i\omega)dx$.

Then, taking Equation 2-49 into account, multiply Equation 2-48 by $1+\sigma(x)/i\omega$. Equation 2-48 transforms to the following form:

$$\left(\frac{(i\omega)^2}{c^2} + \frac{i\omega\sigma}{c^2}\right)p - \frac{\partial}{\partial x}\left(\frac{1}{1 + \frac{\sigma}{i\omega}}\frac{\partial p}{\partial x}\right) = 0\,. \tag{2-50}$$

The transformation to the time domain is performed according to the rule $i\omega p \to \partial p/\partial t$. Its direct application to Equation 2-50 would result in a time integral of p. To avoid this, an auxiliary variable u is introduced:

$$\frac{1}{1 + \frac{\sigma}{i\omega}}\frac{\partial p}{\partial x} = \frac{\partial p}{\partial x} - \frac{\sigma}{\sigma + i\omega}\frac{\partial p}{\partial x} = \frac{\partial p}{\partial x} + u\,. \tag{2-51}$$

Equation 2-50 and Equation 2-51 yield a system of partial differential equations in the time domain equivalent to the frequency domain Equation 2-48:

$$\begin{gathered}\frac{1}{c^2}\left(\frac{\partial^2 p}{\partial t^2} + \sigma\frac{\partial p}{\partial t}\right) - \frac{\partial}{\partial x}\left(u + \frac{\partial p}{\partial x}\right) = 0 \\ \frac{\partial u}{\partial t} + \sigma\left(u + \frac{\partial p}{\partial x}\right) = 0,\end{gathered} \tag{2-52}$$

The derivation of the model in 3D space, where more auxiliary variables are required, is given in Ref. 39.

References for the Pressure Acoustics Branch

1. D. Givoli and B. Neta, "High-order Non-reflecting Boundary Scheme for Time-dependent Waves," *J. Comput. Phys.*, vol. 186, pp. 24–46, 2004.

2. A. Bayliss, M. Gunzburger, and E. Turkel, "Boundary Conditions for the Numerical Solution of Elliptic Equations in Exterior Regions," *SIAM J. Appl. Math.*, vol. 42, no. 2, pp. 430–451, 1982.

3. A.B. Bauer, "Impedance Theory and Measurements on Porous Acoustic Liners," *J. Aircr.*, vol. 14, pp. 720–728, 1977.

4. S. Temkin, *Elements of Acoustics*, Acoustical Society of America, 2001.

5. A.D. Pierce, *Acoustics: An Introduction to its Physical Principles and Applications*, Acoustical Society of America (second print), 1991.

6. D.T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.

7. P.M. Morse and K.U. Ignard, *Theoretical Acoustics*, Princeton University Press, 1986.

8. L.D. Landau and E.M. Lifshitz, *Fluid Mechanics, Course of Theoretical Physics, Volume 6*, Butterworth-Heinemann, 2003.

9. J.F. Allard and N. Atalla, *Propagation of Sound in Porous Media, Modeling Sound Absorbing Materials*, 2nd Edition, John Wiley & Sons, 2009.

10. R. Panneton, "Comment on the Limp Frame Equivalent Fluid Model for Porous Media," *J. Acoust. Soc. Am.* vol. 122, no. 6, pp. EL217–EL222, 2007.

11. M. Sadouki, M. Fellah, Z. E. Fellah, E. Ogam, N. Sebaa, F. G. Mitri, and C. Depollier, "Measuring static thermal permeability and inertial factor of rigid porous materials," *J. Acoust. Soc. Am.*, vol. 130, p. 2627, 2011.

12. X. Olny and R. Panneton, "Acoustical determination of the parameters governing thermal dissipation in porous media," *J. Acoust. Soc. Am.*, vol. 123, 814–824, 2008.

13. D. Lafarge, P. Lemarinier, J.-F. Allard, and V. Tarnow, "Dynamic compressibility of air in porous structures at audible frequencies," *J. Acoust. Soc. Am.*, vol. 102, no. 4, pp. 1994–2006, 1997.

14. S. R. Pride, F. D. Morgan, A. F. and Gangi, "Drag forces of porous-medium acoustics," *Phys. Rev. B*, vol. 47, pp. 4964–4978, 1993.

15. C. Zwikker and C. W. Kosten, *Sound Absorbing Materials*, Elsevier Publishing, New York, 1949.

16. K. Attenborough, "On the acoustic slow wave in air filled granular media," *J. Acoust. Soc. Am.*, vol. 81, no. 1, pp. 93–102, 1987.

17. K. Wilson, "Relaxation-matched modeling of propagation through porous media, including fractal pore structure," *J. Acoust. Soc. Am.*, vol. 94, no. 2, pp. 1136–1145, 1993.

18. T. J. Cox and P. D'Antonio, *Acoustic Absorbers and Diffusers, Taylor and Francis*, 2nd ed., 2009. http://apmr.matelys.com/index.html.

19. R. Kampinga, *Viscothermal Acoustics Using Finite Elements, Analysis Tools for Engineers*, PhD thesis, University of Tweente, The Netherlands, 2010.

20. M. R. Stinson, "The propagation of plane sound waves in narrow and wide circular tubes, and generalization to uniform tubes of arbitrary cross-sectional shapes," *J. Acoust. Soc. Am.*, vol. 89, pp. 550, 1990.

21. H. Tijdeman, "On the propagation of sound waves in cylindrical tubes," *J. Sound Vib.*, vol. 39, pp 1, 1975.

22. Y. Miki, "Acoustical properties of porous materials - modifications of Delany-Bazley models," *J. Acoust. Soc. Jpn (E)*, vol. 11, no. 1, 1990.

23. D. Oliva and V. Hongisto, "Sound absorption of porous materials – Accuracy of prediction methods," *Appl. Acoust.*, vol. 74, issue 12, pp. 1473–1479, 2013.

24. K. L Williams, "An effective density fluid model for acoustic propagation in sediments derived from Biot theory," *J. Acoust. Soc. Am.,* vol. 110, pp. 2276, 2001.

25. A. B. Wood, *A Textbook of Sound,* The Macmillan company, New York, 1941.

26. W. M. Leach, Jr, *Electroacoustics & Audio Amplifier Design*, Kendall Hunt publishing company, 2010.

27. B. Håkansson, P. Carlsson and A. Tjellström, "The mechanical point impedance of the human head, with and without skin penetration", *J. Acoust. Soc. Am.*, vol. 80, issue 4, pp.1065-1075 (1986).

28. H. Hudde and A. Engel, “Measuring and modeling basic properties of the human middle ear and ear canal. Part I: Model structure and measure techniques”, *ACOUSTICA acta acoustica*, vol 84, pp. 720-738 (1998).

29. H. Hudde and A. Engel, “Measuring and modeling basic properties of the human middle ear and ear canal. Part II: Ear canal, middle ear cavities, eardrum, and ossicles”, *ACOUSTICA acta acoustica*, vol 84, pp. 894-913 (1998).

30. H. Hudde and A. Engel, “Measuring and modeling basic properties of the human middle ear and ear canal. Part III: Eardrum impedances, transfer functions and model calculations”, *ACOUSTICA acta acoustica*, vol 84, pp. 1091-1109 (1998).

31. D. H. Keefe, “Acoustical wave propagation in cylindrical ducts: Transmission line parameter approximations for isothermal and nonisothermal boundary conditions”, *J. Acoust. Soc. Am.*, vol. 75, issue 1, pp. 58-62 (1984).

32. H. Levine and J. Schwinger, “On the radiation of sound from an unflanged circular pipe”, *Phys. Rev.*, vol. 73, number 4, pp. 383-406 (1948).

33. R. Courant, K.O. Friedrichs, and H. Lewy, “On the Partial Difference Equations of Mathematical Physics,” IBM Journal, vol. 11, pp. 215–234, 1956.

34. Y. Saad, “A Flexible Inner-outer Preconditioned GMRES Algorithm,” *SIAM J. Sci. Statist. Comput.*, vol. 14, pp. 461–469, 1993.

35. M. R. Stinson and E. A. G. Shaw, “Acoustic impedance of small, circular orifices in thin plates,” *J. Acoust. Soc. Am.*, vol. 77, issue 6, pp. 2039-2042, 1985.

36. M. A. Temiz, J. Tournadre, and I. L. Arteaga, “Non-linear acoustic transfer impedance of micro-perforated plates with circular orifices,” *J. Sound and Vibration*, vol. 366, pp. 418-428, 2016.

37. T. Elnady, “Modelling and characterization of perforates in lined ducts and mufflers,” *PhD Thesis*, The Royal Institute of Technology (KTH), 2004.

38. J.-P. Bérenger, “A perfectly matched layer for the absorption of electromagnetic waves,” *J. Comp. Phys.*, vol. 114, issue 2, pp. 185–200, 1994.

39. B. Kaltenbacher, M. Kaltenbacher, and I. Sim, “A modified and stable version of a perfectly matched layer technique for the 3-d second order wave equation in time domain with an application to aeroacoustics,” *J. Comp. Phys.*, vol. 235, pp. 407–422, 2013.

3

Acoustic-Structure Interaction Interfaces

This chapter describes the physics interfaces found under the **Acoustic-Structure Interaction** branch ( ).

The Acoustic-Solid Interaction, Frequency Domain Interface

The **Acoustic-Solid Interaction, Frequency Domain** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) when adding a physics interface, combines the Pressure Acoustics, Frequency Domain and Solid Mechanics interfaces to connect the acoustics pressure variations in the fluid domain with the structural deformation in the solid domain. It can, for example, be used to determine the transmission of sound through an elastic structure or solve for the coupled vibroacoustics phenomena present in a loudspeaker.

Acoustic-structure interaction refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid domain, and the structural acceleration acts on the fluid domain as a normal acceleration across the fluid-solid boundary.

A dedicated multiphysics coupling condition is readily defined for the fluid-solid boundary and sets up the fluid loads on the solid domain and the effect of the structural accelerations on the fluid. The interface exists for frequency domain and eigenfrequency studies.

The physics interface is available for 3D, 2D, and 2D axisymmetric geometries and has the capability to model pressure acoustics and solid mechanics in the frequency domain, including a special acoustic-solid boundary condition for the fluid-solid interaction.

When a predefined **Acoustic-Solid Interaction, Frequency Domain** interface is added from the **Acoustics>Acoustic-Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain** and **Solid Mechanics** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Acoustic-Structure Boundary**.

See The Multiphysics Node and Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

On the Constituent Physics Interfaces

The Pressure Acoustics, Frequency Domain interface is used to compute the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. It solves the Helmholtz equation and is suited for all linear frequency-domain acoustic simulations with harmonic variations of the pressure field. The physics interface includes domain conditions to model losses in a homogenized way, so-called fluid models for porous materials as well as losses in narrow regions. The domain features also include background incident acoustic fields.

The Solid Mechanics interface is intended for general structural analysis of 3D, 2D or axisymmetric bodies. In 2D, the plane strain assumption should be used. The physics interface is based on solving Navier's equations, and results such as displacements, stresses, and strains are computed.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, for example **Acoustic-Structure Boundary**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Frequency Domain** on one side and **Solid Mechanics** on the other.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Acoustic-Structure Boundary coupling feature is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface
- The available physics features for The Solid Mechanics Interface are listed in the *Structural Mechanics Module User's Guide*.

- Theory section in the *Structural Mechanics Module User's Guide*
- Theory Background for the Pressure Acoustics Branch

Acoustic-Structure Interaction: Application Library path **Acoustics_Module/Tutorials/acoustic_structure**

Suggestions for setting up the solvers for solving large acoustic-structure interaction problems are given in Solving Large Acoustic-Structure Interaction Models in the Modeling with the Acoustic-Structure Interaction Branch section.

The Acoustic-Solid Interaction, Transient Interface

The **Acoustic-Solid Interaction, Transient** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) when adding a physics interface, combines the Pressure Acoustics, Transient and Solid Mechanics interfaces to connect the acoustics pressure variations in the fluid domain with the structural deformation in the solid domain. It can, for example, be used to determine the transmission of sound through an elastic structure or solve for the coupled vibroacoustics phenomena present in a loudspeaker.

Acoustic-structure interaction refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid domain, and the structural acceleration acts on the fluid domain as a normal acceleration across the fluid-solid boundary.

Special physics interface conditions are readily defined at the fluid-solid boundary and set up the fluid loads on the solid domain and the effect of the structural accelerations on the fluid. The physics interface allows for transient studies but also solves in the frequency domain with the available boundary conditions.

When a predefined **Acoustic-Solid Interaction, Transient** interface is added from the **Acoustics>Acoustic-Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Transient** and **Solid Mechanics** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Acoustic-Structure Boundary**.

See The Multiphysics Node and Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

On the Constituent Physics Interfaces

The Pressure Acoustics, Transient interface is used to compute the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. It solves the scalar wave equation and is suited for time-dependent simulations with arbitrary time-dependent fields and sources. Domain conditions also include background incident acoustic fields. User-defined sources can be added to, for

example, include certain nonlinear effects such as a square pressure dependency of the density variations.

The Solid Mechanics interface is intended for general structural analysis of 3D, 2D, or axisymmetric bodies. In 2D, the plane strain assumption should be used. The physics interface is based on solving Navier's equations, and results such as displacements, stresses, and strains are computed.

For modeling of acoustic-structure interaction in the frequency domain, The Acoustic-Solid Interaction, Frequency Domain Interface contains additional functionality that is not applicable for modeling in the time domain.

- Theory section in the *Structural Mechanics Module User's Guide*
- Theory Background for the Pressure Acoustics Branch

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined coupling features the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Transient** on one side and **Solid Mechanics** on the other.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Acoustic-Structure Boundary coupling feature is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Pressure Acoustics, Transient Interface are listed in the section Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface
- The available physics features for The Solid Mechanics Interface are listed in the *Structural Mechanics Module User's Guide*

- Theory section in the *Structural Mechanics User's Guide*
- Theory Background for the Pressure Acoustics Branch

The Acoustic-Piezoelectric Interaction, Frequency Domain Interface

The **Acoustic-Piezoelectric Interaction, Frequency Domain** interface ( ), found when adding a physics interface under the **Acoustics>Acoustic-Structure Interaction** branch ( ), combines the Pressure Acoustics, Frequency Domain, Solid Mechanics, and Electrostatics interfaces to connect and solve for the acoustic pressure variations in fluids with the structural deformation in both solids and piezoelectric solid domains. The physics interface also includes features from Electrostatics to solve for the electric field in the piezoelectric material. It may for example be used for modeling piezoelectric transducers for sonar or medical applications and, for example, enhancing the impedance matching layers as well as the far-field radiation patterns of the transducer.

The Helmholtz equation is solved in the fluid domain and the structural equations in the solid together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled, and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms.

When a predefined **Acoustic-Piezoelectric Interaction, Frequency Domain** interface is added from the **Acoustics>Acoustic-Structure Interaction** branch ( ) of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain**, **Solid Mechanics**, and **Electrostatics** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling features **Acoustic-Structure Boundary** and **Piezoelectric Effect**.

See The Multiphysics Node and Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual.*

On the Constituent Physics Interfaces

The Pressure Acoustics, Frequency Domain interface computes the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. The physics interface solves the Helmholtz equation and is suited for all linear frequency-domain acoustic simulations with harmonic variations of the pressure

field. It includes domain conditions to model losses in a homogenized way, so-called fluid models for porous materials as well as losses in narrow regions. The domain features also include background incident acoustic fields.

The Piezoelectric Devices interface combines Solid Mechanics and Electrostatics together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms.

The equations solved in the solid and fluid domains can be found in the *Structural Mechanics Module User's Guide* and in Theory Background for the Pressure Acoustics Branch, respectively.

Piezoacoustic Transducer: Application Library path **Acoustics_Module/Piezoelectric_Devices/piezoacoustic_transducer**

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined coupling features, for example **Acoustic-Structure Boundary**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Frequency Domain** on one side and **Solid Mechanics** on the other.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Acoustic-Structure Boundary coupling feature is described in the Multiphysics Couplings chapter. The Piezoelectric Effect is described for The Piezoelectric Devices Interface.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface
- The available physics features for The Solid Mechanics Interface that are described in the *Structural Mechanics Module User's Guide*.
- The available physics features for The Electrostatics Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Electrostatics Interface in the *COMSOL Multiphysics Reference Manual*
- Some additional piezoelectric devices features for the Solid Mechanics and Electrostatics interfaces are described for The Piezoelectric Devices Interface.

- Theory section in the *Structural Mechanics Module User's Guide*
- Theory of Electrostatics in the *COMSOL Multiphysics Reference Manual*
- Theory Background for the Pressure Acoustics Branch

The Acoustic-Piezoelectric Interaction, Transient Interface

The **Acoustic-Piezoelectric Interaction, Transient** interface ( ), found when adding a physics interface under the **Acoustics>Acoustic-Structure Interaction** branch ( ), combines the Pressure Acoustics, Transient and Piezoelectric Devices interfaces to connect and solve for the acoustic pressure variations in fluids with the structural deformation in both solids and piezoelectric solid domains. The physics interface also includes features from Electrostatics to solve for the electric field in the piezoelectric material. Examples include modeling piezoelectric transducers for sonar or medical applications and, for example, enhancing the impedance matching layers.

The pressure wave equation is solved in the fluid domain and the structural dynamic equations in the solid together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled, and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms.

When a predefined **Acoustic-Piezoelectric Interaction, Transient** interface is added from the **Acoustics>Acoustic-Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain**, **Solid Mechanics**, and **Electrostatics** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling features **Acoustic-Structure Boundary** and **Piezoelectric Effect**.

See The Multiphysics Node and Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

On the Constituent Physics Interfaces

The Pressure Acoustics, Transient interface computes the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. The physics interface solves the scalar wave equation and is suited for time-dependent analyses with arbitrary time-dependent fields and sources. Domain conditions also include background incident acoustic fields. User-defined sources can be added to, for example, include certain nonlinear effects such as a square pressure dependency of the density variations.

The Piezoelectric Devices interface combines Solid Mechanics and Electrostatics together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined coupling features, for example **Acoustic-Structure Boundary**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Frequency Domain** on one side and **Solid Mechanics** on the other.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Acoustic-Structure Boundary coupling feature is described in the Multiphysics Couplings chapter. The Piezoelectric Effect is shortly described in The Piezoelectric Devices Interface.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Pressure Acoustics, Transient Interface are listed in the section Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface
- The available physics features for The Solid Mechanics Interface are listed in the *Structural Mechanics Module User's Guide*.
- The available physics features for The Electrostatics Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Electrostatics Interface in the *COMSOL Multiphysics Reference Manual*
- Some additional piezoelectric devices features for the Solid Mechanics and Electrostatics interfaces are described for The Piezoelectric Devices Interface.

- Theory section in the *Structural Mechanics Module User's Guide*
- Theory of Electrostatics in the *COMSOL Multiphysics Reference Manual*
- Theory Background for the Pressure Acoustics Branch

The Solid Mechanics (Elastic Waves) Interface

The **Solid Mechanics (Elastic Waves)** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) when adding a physics interface, is a shortcut to add the Solid Mechanics interface which is used to compute the displacement field in solids with propagating elastic waves. The Solid Mechanics interface supports studies in many forms, specifically the dynamic Navier's equation is solved in the solid in the frequency domain. Dedicated Multiphysics Couplings exist to couple fluid, solid, and porous domains.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linear Elastic Material**, **Free**, and **Initial Values**. For 2D axisymmetric components an **Axial Symmetry** node is also added.

The Poroelastic Waves Interface

The **Poroelastic Waves (pelw)** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) when adding a physics interface, is used to compute the displacement field and acoustic pressure fluctuation in porous materials with propagating poroelastic waves. Dedicated Multiphysics Couplings exist that define the couplings between fluid, solid, and porous domains.

Examples of applications include the propagation of elastic waves in rocks and soils, modeling the acoustic attenuation properties of particulate filters, characterizing sound absorbers and liners, or modeling the porous foams in headphones. The physics interface is valid for modeling the propagation of the coupled linear elastic and linear acoustic waves in the frequency domain. Harmonic variations of the displacement field and the sources are assumed. In the porous domains, Biot's equations are solved accounting for the coupled propagation of elastic waves in the elastic porous matrix and pressure waves in the saturating pore fluid. This includes the damping effect of the pore fluid due to viscous losses only (the Biot model), typically with a saturating liquid like water or oil, or the combined effect of viscous and thermal losses (the Biot-Allard model), typically when the saturating fluid is air.

See the Theory for the Poroelastic Waves Interfaces for details about the governing equations. The specifics of the Biot and the Biot-Allard models are also discussed here.

When the **Poroelastic Waves** interface is added, these default nodes are also added to the **Model Builder** — **Poroelastic Material**, **Porous, Free**, and **Initial Values**. For 2D axisymmetric components an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Porelastic Waves** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to

different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `pelw`.

The rest of the physics interface settings are the same as for The Pressure Acoustics, Frequency Domain Interface and The Solid Mechanics Interface.

Acoustics of a Particulate-Filter-Like System: Application Library path **Acoustics_Module/Automotive/acoustics_particulate_filter**

Domain, Boundary, and Pair Nodes for the Poroelastic Waves Interfaces

The Poroelastic Waves Interface has these domain, boundary, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users)

- Added Mass[1]
- Boundary Load[1]
- Fixed Constraint
- Free[1]
- Initial Stress and Strain[1]
- Initial Values
- Periodic Condition
- Poroelastic Material
- Porous, Free
- Porous, Pressure
- Prescribed Acceleration
- Prescribed Displacement
- Prescribed Velocity
- Rigid Connector[1]
- Roller
- Septum Boundary Load
- Spring Foundation[1]
- Symmetry
- Thin Elastic Layer[1]

[1] These features are described for the Solid Mechanics interface in the *Structural Mechanics Module User's Guide*

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Poroelastic Material

Use the **Poroelastic Material** node to define the poroelastic material and fluid properties, that is the properties of the porous matrix and the saturating fluid.The

subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

POROELASTIC MODEL

Select the **Model** used to describe the losses to include in the porous material:

- **Biot (viscous losses)**, this model is primarily used in earth sciences when the saturating fluid is a liquid like water or oil. The model is based on Biot's original work and only includes the effects of viscous losses in the pores.
- **Biot-Allard (thermal and viscous losses)**, this model is intended for simulating porous materials where the saturating fluid is air. This is for modeling sound absorbers, liners, foams used in headphones and loudspeakers, cloth and much more.

When selecting a specific model the required material input will change in order to align with the data normally available for the intended applications.

POROUS MATRIX PROPERTIES

The default **Porous elastic material** uses the **Domain material** (the material defined for the domain). Select another material as needed.

Select a **Porous model**: **Drained matrix, isotropic**, **Drained matrix, orthotropic**, or **Drained matrix, anisotropic**. Then enter or select the settings as described.

- Porous Model Drained Matrix, Isotropic
- Porous Model for Drained Matrix, Orthotropic
- Porous Model for Drained Matrix, Anisotropic
- Porous Matrix Parameters for Biot (viscous losses) Model
- Porous Matrix Parameters for Biot-Allard (thermal and viscous losses) Model

Porous Model Drained Matrix, Isotropic

If **Drained matrix, isotropic** is selected from the **Porous model** list, select a pair of elastic properties to describe an isotropic drained porous material. The drained parameters are also known as the in vacuo elastic parameters, the are in principle measured without the presence of the saturating fluid. From the **Specify** list, select:

- **Young's modulus and Poisson's ratio** to specify drained Young's modulus (elastic modulus) E_d (SI unit: Pa) and Poisson's ratio ν_d (dimensionless). For an isotropic material Young's modulus is the spring stiffness in Hooke's law, which in 1D form is $\sigma = E_d\varepsilon$ where σ is the stress and ε is the strain. Poisson's ratio defines the normal strain in the perpendicular direction, generated from a normal strain in the other direction and follows the equation $\varepsilon_\perp = -\upsilon\varepsilon_\parallel$

- **Shear modulus and Poisson's ratio** (the default for the Biot-Allard model) to specify drained shear modulus G_d (SI unit: Pa) and Poisson's ratio ν_d (dimensionless).
- **Young's modulus and Shear modulus** to specify drained Young's modulus (elastic modulus) E_d (SI unit: Pa) and drained shear modulus G_d (SI unit: Pa).
- **Bulk modulus and shear modulus** (the default for the Biot model) to specify the drained bulk modulus K_d (SI unit: Pa) and the drained shear modulus G_d (SI unit: Pa). The bulk drained modulus is a measure of the solid porous matrix's resistance to volume changes. The shear modulus is a measure of the solid porous matrix's resistance to shear deformations.
- **Lamé parameters** to specify the drained Lamé parameters λ_d (SI unit: Pa) and μ_d (SI unit: Pa).
- **Pressure-wave and shear-wave speeds** to specify the drained pressure-wave speed c_p (SI unit: m/s) and the shear-wave speed c_s (SI unit: m/s).

For each pair of properties, select from the applicable list to use the value **From material** or enter a **User defined** value or expression. Each of these pairs define the drained elastic properties and it is possible to convert from one set of properties to another.

Porous Model for Drained Matrix, Orthotropic

When **Drained matrix, orthotropic** is selected from the **Porous model** list, the material properties of the solid porous matrix vary in orthogonal directions only.

The default properties take values **From material**. For **User defined** enter values or expressions for the drained **Young's modulus** $\mathbf{E}$ (SI unit: Pa), the drained **Poisson's ratio** ν (dimensionless), and the drained **Shear modulus** $\mathbf{G}$ (SI unit: Pa).

Porous Model for Drained Matrix, Anisotropic

When **Drained matrix, anisotropic** is selected from the **Porous model** list, the material properties of the solid porous matrix vary in all directions, and the stiffness comes from the symmetric **Elasticity matrix**, $\mathbf{D}$ (SI unit: Pa). The default uses values **From material**. For **User defined** enter values in the 6-by-6 symmetric matrix that displays.

Porous Matrix Parameters for Biot (viscous losses) Model

Enter the following (remaining) parameters necessary to defined the properties of a **Biot (viscous losses)** porous material model. The defaults use values **From material**. For **User defined** enter other values or expressions as needed.

- **Bulk modulus and shear modulus** (the default for the Biot model as described above) to specify the drained bulk modulus K_d (SI unit: Pa) and the drained shear modulus G_d (SI unit: Pa).

- **Drained density of porous material** to specify the drained density of the porous material in vacuum ρ_d (SI unit: kg/m^3). The drained density ρ_d is equal to $(1 - \varepsilon_p)\,\rho_s$ where ρ_s is the density of the solid material from which the matrix is made and ε_p is the porosity.
- **Permeability** to specify the permeability of the porous material κ_p (SI unit: m^2). The permeability is a measure of the ability of the porous material to let fluid pass through it. It hence gives some measure of the pore size and thus correlates to the viscous damping experienced by pressure waves propagating in the saturating fluid.
- **Porosity** to specify the porosity of the material ε_p (dimensionless). It defines the amount of void volume inside the porous matrix and takes values between 0 (no porous material only fluid) and 1 (fully solid material no fluid).
- **Biot-Willis coefficient** to specify the Biot-Willis coefficient α_B (dimensionless). This coefficient relates the bulk modulus (compressibility) of the drained porous matrix to a block of solid material. It is defined as

$$\alpha_B = 1 - \frac{K_d}{K_s}$$

  where K_d is the drained bulk modulus and K_s is the bulk module of a block of solid material (made of the matrix material). The drained bulk modulus is related to the stiffness of the porous matrix, while the solid bulk modulus is related to the compressibility of the material or grains from which the porous matrix is made. The Biot-Willis coefficient is bound by $\varepsilon_p \le \alpha_B \le 1$. A rigid porous matrix (Voigt upper bound) has $\alpha_B = \varepsilon_p$ and a soft or limp porous matrix (Reuss lower bound) has $\alpha_B = 1$.
- **Tortuosity factor** (high frequency limit) or the structural form factor τ_∞ (dimensionless). This is a purely geometrical factor that depends on the microscopic geometry and distribution of the pores inside the porous material. It is independent of the fluid and solid properties and is normally >1. The default is 2. The more complex the propagation path through the material, the higher is the absorption. The tortuosity partly represents this complexity.

Porous Matrix Parameters for Biot-Allard (thermal and viscous losses) Model

Enter the following (remaining) parameters necessary to defined the properties of a **Biot-Allard (thermal and viscous losses)** porous material model. The defaults use values **From material**. For **User defined** enter other values or expressions as needed.

- **Shear modulus and Poisson's ratio** (the default for the Biot-Allard model as described above) to specify drained shear modulus G_d (SI unit: Pa) and Poisson's ratio ν_d (dimensionless).
- **Drained density of porous material** to specify the drained density of the porous material in vacuum ρ_d (SI unit: kg/m^3). The drained density ρ_d is equal to $(1 - \varepsilon_p)\,\rho_s$ where ρ_s is the density of the solid material from which the matrix is made and ε_p is the porosity.
- **Porosity** to specify the porosity of the material ε_p (dimensionless). It defines the amount of void volume inside the porous matrix and takes values between 0 (no porous material only fluid) and 1 (fully solid material no fluid).
- **Flow resistivity** to specify the (static) flow resistivity of the porous material R_f (SI unit: $Pa{\cdot}s/m^2$). The flow resistivity is a measure of the ability of the porous material to let fluid pass through it. It hence gives some measure of the pore size and thus correlates to the viscous damping experienced by pressure waves propagating in the saturating fluid. The flow resistivity is also sometimes denoted σ (using the unit $N{\cdot}s/m^4$) and it is related to the permeability through $\kappa_p = \mu/R_f$.
- **Isotropic structural loss factor** to specify the loss factor of the porous matrix η_s (dimensionless). This value introduces the damping due to losses in the porous structure by transform the elastic moduli into complex valued quantities. This quantity can be frequency dependent if necessary.
- **Tortuosity factor** (high frequency limit) or the structural form factor τ_∞ (dimensionless). This is a purely geometrical factor that depends on the microscopic geometry and distribution of the pores inside the porous material. It is independent of the fluid and solid properties and is normally >1. The default is 2. The more complex the propagation path through the material, the higher is the absorption. The tortuosity partly represents this complexity.
- **Viscous characteristic length** to specify the viscous length scale L_v (SI unit: m). This value is sometimes denoted Λ and replaces the hydraulic radius used in simpler models to account for the viscous losses that appear in the acoustic boundary layer at pore walls.
- **Thermal characteristic length** to specify the thermal length scale L_{th} (SI unit: m). This value is sometimes denoted Λ' and replaces the hydraulic radius used in simpler

models to account for the thermal losses that appear in the acoustic boundary layer at pore walls.

- **Biot-Willis coefficient** how to calculate the Biot-Willis coefficient α_B by selecting **From material**, **Rigid assumption** (the default), **General model**, or **User defined**. This coefficient relates the bulk modulus (compressibility) of the drained porous matrix to a block of solid material.
  - **From material** to pick up the value from the domain material.
  - **Rigid assumption** (the default) and the model defines a rigid porous matrix (Voigt upper bound) where $\alpha_B = \varepsilon_p$.
  - **General model** to define the Biot-Willis coefficient α_B according to its general definition

$$\alpha_B = 1 - \frac{K_d}{K_s}$$

    where K_d is the drained bulk modulus and K_s is the bulk module of the skeleton material (bulk modulus of a block of solid material made of the matrix material). When this option is selected also enter the **Bulk modulus of skeleton material** K_s (taken from material as default). The drained bulk modulus K_d is related to the stiffness of the porous matrix, while the skeleton bulk modulus K_s is related to the compressibility of the material or grains from which the porous matrix is made. The Biot-Willis coefficient is bound by $\varepsilon_p \le \alpha_B \le 1$. A rigid porous matrix (Voigt upper bound) has $\alpha_B = \varepsilon_p$ and a soft or limp porous matrix (Reuss lower bound) has $\alpha_B = 1$.
  - **User defined** enter a value for the Biot-Willis coefficient α_B (dimensionless).

FLUID PROPERTIES

Define the properties of the saturating fluid in terms of its density, viscosity and compressibility but also the viscosity model. The defaults use values for the material parameters are **From material**. For **User defined** enter other values or expressions as needed.

- Fluid Parameters for Biot (viscous losses) Model
- Fluid Parameters for Biot-Allard (thermal and viscous losses) Model

Fluid Parameters for Biot (viscous losses) Model

- **Density** defines the density of the saturating fluid ρ_f (SI unit: kg/m^3).

- **Dynamic viscosity** to define the dynamic viscosity of the saturating fluid μ_f (SI unit: Pa·s). The parameter is important for the amount of viscous damping experienced by the acoustic waves.
- **Compressibility** of the saturating fluid χ_f (SI unit: 1/Pa). Remember that the fluid compressibility χ_f is related to the fluid bulk modulus K_f (SI unit: Pa) and the speed of sound c, through the relation

$$\chi_f = \frac{1}{K_f} = \frac{1}{c^2 \rho_f}$$

The compressibility of the fluid also enters the expression for Biot's module M, give by

$$M = \frac{K_s}{1 - \varepsilon_p - \frac{K_d}{K_s} + \varepsilon_p K_s \chi_f} \qquad K_s = \frac{K_d}{1 - \alpha_B}$$

It should be noted that Biot-Willis coefficient α_B only depends on the properties of the porous matrix while Biot's module M depends on both fluid and porous matrix properties.

Select a **Viscosity Model**, either **Biot's low frequency range** or **Biot's high frequency range**.

- **Biot's low frequency range** models damping at low frequencies where the acoustic boundary layer (the viscous penetration depth) is assumed to span the full width of the pores. This is also the so-called Poiseuille limit.
- For **Biot's high frequency range** also select **Specify** as **Reference frequency** or **Characteristic pore size**. Either enter a **Reference frequency** f_r (SI unit: Hz) or a **Characteristic pore size** a (SI unit: m). This model implements a correction factor to the viscosity that accounts for the relative scale difference between a typical pore diameter and the acoustic boundary layer thickness. The modified viscosity is of the form

$$\tilde{\mu}_f = \mu_f F\left(\sqrt{\frac{f}{f_r}}\right) \qquad f_r = \frac{\mu_f}{2\pi a^2 \rho_f}$$

where f_r is the reference frequency and a is a characteristic size of the pores. The expression for f_r is one typically used in literature but it is often measured or

empirically determined. The expression for f_r corresponds to finding the frequency at which the viscous boundary layer thickness is of the scale a.

See High Frequency Correction (Biot Model) for more details.

Fluid Parameters for Biot-Allard (thermal and viscous losses) Model

- **Density** to define the density of the saturating fluid ρ_f (SI unit: kg/m^3).
- **Dynamic viscosity** to define the dynamic viscosity of the saturating fluid μ_f (SI unit: Pa·s). The parameter is important for the amount of viscous damping experienced by the acoustic waves.
- **Ratio of specific heats** to define the ratio of specific heats (adiabatic index) of the saturating fluid γ (dimensionless).
- **Heat capacity at constant pressure** to define the (specific) heat capacity at constant pressure of the saturating fluid C_p (SI unit: J/(kg·K)).
- **Thermal conductivity** to define the thermal conductivity of the saturating fluid $\mathbf{k}$ (SI unit: W/(m·K)). The parameter is important for the amount of thermal damping experienced by the acoustic waves.

Porous, Free

The **Porous, Free** node is the default boundary condition for The Poroelastic Waves Interface. It means that there are no constraints and no loads acting on the porous matrix, and a sound-soft boundary for the fluid pressure.

Porous, Free (Sound-Soft Boundary) Theory

Initial Values

The **Initial Values** node adds initial values for the pressure and displacement field. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the **Pressure** p (SI unit: Pa) and **Displacement field** $\mathbf{u}$ (SI unit: m) initial values. The default is 0 Pa for the pressure and 0 m for the displacement field.

Fixed Constraint

The **Fixed Constraint** node adds a condition that makes the porous matrix fixed (fully constrained); that is, the displacements are zero in all directions. This boundary condition also sets an impervious (sound-hard) boundary for the fluid pressure.

Fixed (Sound-Hard Boundary) Equations

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and physics interfaces being modeled. The condition is found under the **Connections** submenu.

PERIODICITY SETTINGS

Select a **Type of periodicity**: **Continuity** (the default), **Floquet periodicity** (Bloch periodicity), or **Cyclic symmetry**, or **Antiperiodicity**.

For details see the Periodic Condition description in Pressure Acoustics.

Porous, Pressure

The **Porous, Pressure** node creates a boundary condition that acts as a pressure source at the boundary, which means a constant acoustic pressure $p = p_0$ is specified. In the frequency domain, p_0 is the amplitude of a harmonic pressure source.

PRESSURE

Enter the value of the **Pressure** p_0 (SI unit: Pa) at the boundary.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

	Pressure Equations

Prescribed Displacement

The **Prescribed Displacement** node adds a condition where the displacements are prescribed in one or more directions to the porous matrix boundary.

If a displacement is prescribed in one direction, this leaves the porous matrix free to deform in the other directions. Also define more general displacements as a linear combination of the displacements in each direction.

- If a prescribed displacement is not activated in any direction, this is the same as a **Free** constraint.
- If a zero displacement is applied in all directions, this is the same as a **Fixed Constraint** for the porous matrix and a **Sound Hard Wall** for the fluid.

PRESCRIBED DISPLACEMENT

Define the prescribed displacements using a **Standard notation** or a **General notation**.

Standard Notation

To define the displacements individually, click the **Standard notation** button (the default).

	To define a prescribed displacement for each space direction (x, y, and z for 3D), select one or more of the **Prescribed in x direction**, **Prescribed in y direction**, and **Prescribed in z direction** check boxes. Then enter a value or expression for the prescribed displacements u_0, v_0, or w_0 (SI unit: m).

	For 2D axisymmetric components and to define a prescribed displacement for each space direction (r and z), select one or both of the **Prescribed in r direction** and **Prescribed in z direction** check boxes. Then enter a value or expression for the prescribed displacements u_0 or w_0 (SI unit: m).

General Notation

To specify the displacements using a **General notation** that includes any linear combination of displacement components, click the **General notation** button.

Enter values in the **H matrix** and **R vector** fields. For the H matrix, also select an **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** matrix and enter values as needed.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

	Prescribed Displacement Equations

Prescribed Velocity

The **Prescribed Velocity** node adds a boundary condition where the velocity of the porous matrix is prescribed in one or more directions. With this boundary condition it is possible to prescribe a velocity in one direction, leaving the porous matrix free in the other directions. The condition is found under the **More Constraints** submenu.

COORDINATE SYSTEM SELECTION

	Coordinate systems with directions which change with time should not be used.

PRESCRIBED VELOCITY

	To define a porous, prescribed velocity for each space direction (x, y, and z for 3D), select one or all of the **Prescribed in x direction, Prescribed in y direction,** and **Prescribed in z direction** check boxes. Then enter a value or expression for the components v_x, v_y, and v_z (SI unit: m/s).

	For 2D axisymmetric components and to define a prescribed velocity for each space direction (r and z), select one or both of the **Prescribed in r direction** and **Prescribed in z direction** check boxes. Then enter a value or expression for v_r and v_z (SI unit: m/s).

Prescribed Velocity Equations

Prescribed Acceleration

The **Prescribed Acceleration** node adds a boundary condition, where the acceleration of the porous matrix is prescribed in one or more directions. With this boundary condition, it is possible to prescribe a acceleration in one direction, leaving the porous matrix free in the other directions. The condition is found under the **More Constraints** submenu.

COORDINATE SYSTEM SELECTION

Coordinate systems with directions which change with time should not be used.

PRESCRIBED ACCELERATION

To define a porous, prescribed acceleration for each space direction (x, y, and z for 3D), select one or all of the **Prescribed in x direction, Prescribed in y direction,** and **Prescribed in z direction** check boxes. Enter a value or expression for the prescribed acceleration a_x, a_y, and a_z (SI unit: $\mathrm{m/s^2}$).

For 2D axisymmetric components and to define a porous, prescribed acceleration for each space direction (r and z), select one or both of the **Prescribed in r direction** and **Prescribed in z direction** check boxes. Then enter a value or expression for the prescribed acceleration a_r and a_z (SI unit: $\mathrm{m/s^2}$).

Prescribed Acceleration Equations

Roller

The **Roller** node adds a roller (sliding wall) constraint as the boundary condition; that is, the porous matrix displacement is zero in the direction perpendicular (normal) to the boundary, but the porous matrix is free to move in the tangential direction. This boundary condition also sets an impervious (sound-hard) boundary for the fluid pressure. The condition is found under the **More Constraints** submenu

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Roller Equations

Septum Boundary Load

Add a **Septum Boundary Load** to boundaries for a pressure acting on the porous matrix through a septum layer.

SEPTUM BOUNDARY LOAD

Enter a **Surface density** ρ_{sep} (SI unit: kg/m^3). Enter coordinates for the **Load** $\mathbf{F}_A$ (SI unit: N/m^2).

Septum Boundary Load Equations

Symmetry

The **Symmetry** node adds a boundary condition where there is symmetry in the pressure and displacement field. Use this condition to reduce the size of a model by cutting it in half where there are symmetries. The condition is found under the **More Constraints** submenu.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

The Acoustic-Poroelastic Waves Interaction Interface

The **Acoustic-Poroelastic Waves Interaction** interface ( ) combines Pressure Acoustics, Frequency Domain and Poroelastic Waves together with the Acoustic-Porous Boundary multiphysics coupling feature.

On the Constituent Physics Interfaces

The Pressure Acoustics, Frequency Domain interface computes the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. The physics interface solves the Helmholtz equation and is suited for all linear frequency-domain acoustic simulations with harmonic variations of the pressure field. It includes domain conditions to model losses in a homogenized way, so-called fluid models for porous materials, as well as losses in narrow regions. The domain features also include background incident acoustic fields.

The Poroelastic Waves interface is used to compute the displacement field and acoustic pressure fluctuation in porous materials with propagating poroelastic waves. Examples of applications include the propagation of elastic waves in rocks and soils, modeling the acoustic attenuation properties of particulate filters, characterizing sound absorbers and liners, or modeling the porous foams in headphones. The physics interface is valid for modeling the propagation of the coupled linear elastic and linear acoustic waves in the frequency domain. Harmonic variations of the displacement field and the sources are assumed. In the porous domains, Biot's equations are solved accounting for the coupled propagation of elastic waves in the elastic porous matrix and pressure waves in the saturating pore fluid. This includes the damping effect of the pore fluid due to viscous losses only (the Biot model) or the combined effect of viscous and thermal losses (the Biot-Allard model).

When a predefined **Acoustic-Poroelastic Waves Interaction** interface is added from the **Acoustics>Acoustic-Structure Interaction** branch ( ) of the **Model Wizard** or the **Add Physics** windows, the **Poroelastic Waves** and **Pressure Acoustics, Frequency Domain** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling features **Acoustic-Porous Boundary**.

See The Multiphysics Node and Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined coupling features, for example **Acoustic-Poroelastic Waves Interaction**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Frequency Domain** on one side and **Poroelastic Waves** on the other.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Poroelastic Waves** and **Pressure Acoustics, Frequency Domain** interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Acoustic-Porous Boundary coupling feature nodes is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Poroelastic Waves Interface are listed in the section Domain, Boundary, and Pair Nodes for the Poroelastic Waves Interfaces
- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface

The Acoustic-Solid-Poroelastic Waves Interaction Interface

The **Acoustic-Solid-Poroelastic Waves Interaction** interface ( ) combines Pressure Acoustics, Frequency Domain, Solid Mechanics, and Poroelastic Waves together with the Acoustic-Structure Boundary, Porous-Structure Boundary, and Acoustic-Porous Boundary multiphysics coupling feature.

On the Constituent Physics Interfaces

The Pressure Acoustics, Frequency Domain interface computes the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. The physics interface solves the Helmholtz equation and is suited for all linear frequency-domain acoustic simulations with harmonic variations of the pressure field. It includes domain conditions to model losses in a homogenized way, so-called fluid models for porous materials, as well as losses in narrow regions. The domain features also include background incident acoustic fields.

The Solid Mechanics interface is intended for general structural analysis of 3D, 2D or axisymmetric bodies. In 2D, the plane strain assumption should be used. The physics interface is based on solving Navier's equations, and results such as displacements, stresses, and strains are computed.

The Poroelastic Waves interface is used to compute the displacement field and acoustic pressure fluctuation in porous materials with propagating poroelastic waves. Examples of applications include the propagation of elastic waves in rocks and soils, modeling the acoustic attenuation properties of particulate filters, characterizing sound absorbers and liners, or modeling the porous foams in headphones. The physics interface is valid for modeling the propagation of the coupled linear elastic and linear acoustic waves in the frequency domain. Harmonic variations of the displacement field and the sources are assumed. In the porous domains, Biot's equations are solved accounting for the coupled propagation of elastic waves in the elastic porous matrix and pressure waves in the saturating pore fluid. This includes the damping effect of the pore fluid due to viscous losses only (the Biot model) or the combined effect of viscous and thermal losses (the Biot-Allard model).

When a predefined **Acoustic-Solid-Poroelastic Waves Interaction** interface is added from the **Acoustics>Acoustic-Structure Interaction** branch ( ) of the **Model Wizard** or the

Add Physics windows, the **Pressure Acoustics, Frequency Domain**, **Solid Mechanics**, and **Poroelastic Waves** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling features **Acoustic-Porous Boundary**, **Porous-Structure Boundary**, and **Acoustic-Structure Boundary**.

See The Multiphysics Node and Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined coupling features, for example **Acoustic-Solid-Poroelastic Waves Interaction**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the relevant multiphysics couplings are automatically active on all relevant boundaries connecting two physics.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Poroelastic Waves** and **Pressure Acoustics, Frequency Domain** interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Acoustic-Porous Boundary, Porous-Structure Boundary, and Acoustic-Structure Boundary coupling feature nodes are described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Poroelastic Waves Interface are listed in the section Domain, Boundary, and Pair Nodes for the Poroelastic Waves Interfaces
- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface

Theory for the Poroelastic Waves Interfaces

The Poroelastic Waves Interface theory is described in this section:

- Elastic Waves Introduction
- Poroelastic Waves Theory
- Boundary Conditions for Poroelastic Waves
- Postprocessing Variables
- References for the Poroelastic Waves Interfaces

Elastic Waves Introduction

The most general linear relation (more details are found in the Structural Mechanics Theory section of the *Structural Mechanics Module User's Guide*) between the stress and strain tensors in solid materials can be written as

$$\sigma_{ij} = \mathbf{c}_{ijkl}\varepsilon_{kl}$$

here, σ is the Cauchy's stress tensor, ε is the strain tensor, and $\mathbf{c}_{ijkl}$ is a fourth-order elasticity tensor. For small deformations, the strain tensor is defined as

$$\varepsilon(\mathbf{u}) = \frac{1}{2}(\nabla\mathbf{u} + \nabla\mathbf{u}^T)$$

where $\mathbf{u}$ represents the displacement vector.

The elastic wave equation is then obtained from Newton's second law

$$\rho\frac{\partial^2}{\partial t^2}\mathbf{u} - \nabla\cdot(\sigma(\mathbf{u}) - \mathbf{s}_0) = \mathbf{F}$$

here, ρ is the medium density, and $\mathbf{s}_0$ and $\mathbf{F}$ represent source terms.

An important case is the time-harmonic wave, for which the displacement varies with time as

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

with f (SI unit: Hz) denoting the frequency and $\omega = 2\pi f$ (SI unit: rad/s) the angular frequency. Assuming the same time-harmonic dependency for the source terms $\mathbf{s}_0$ and $\mathbf{F}$, the wave equation for linear elastic waves reduces to an inhomogeneous Helmholtz equation:

$$-\rho\omega^2\mathbf{u} - \nabla \cdot (\sigma(\mathbf{u}) - \mathbf{s}_0) = \mathbf{F} \tag{3-1}$$

Alternatively, treat this equation as an eigenvalue PDE to solve for eigenmodes and eigenfrequencies as described in the *Structural Mechanics Module User's Guide* in the Structural Mechanics Modeling chapter under Eigenfrequency Analysis. Also add damping as described in Mechanical Damping and Losses.

Poroelastic Waves Theory

In his seminal work, Biot extended the classical theory of linear elasticity to porous media saturated with fluids (Ref. 1, Ref. 2, and Ref. 3).

In Biot's theory, the bulk moduli and compressibilities are independent of the wave frequency, and can be treated as constant parameters. The porous matrix is described by linear elasticity and damping is introduced by considering the viscosity of the fluid in the pores, which can be frequency dependent. This description is adequate for the propagation of poroelastic waves in soils and rocks where the saturating fluid is a liquid, like oil or water. This formulation is referred to as the *Biot model* (this is in some sense the classical formulation).

- High Frequency Correction (Biot Model)

When the considered porous material is saturated by a gas, like air, thermal losses need to be included in order to properly model its behavior. This is the case when modeling sound absorbers, car cabin liners, or foams used in headsets or loudspeakers. The formulation of the equations where both the thermal and viscous losses are included is sometimes referred to as *Biot-Allard model*. In this case both the viscosity and the fluid compressibility are considered to be frequency dependent and complex valued (Ref. 7, Ref. 8, Ref. 9, Ref. 10, and Ref. 11).

- Biot-Allard Model (Viscous and Thermal Losses)

Consider Biot's expressions for poroelastic waves (Ref. 3, Ref. 4, and Ref. 6)

$$\rho_{av}\frac{\partial^2}{\partial t^2}\mathbf{u} + \rho_f\frac{\partial^2}{\partial t^2}\mathbf{w} - \nabla\cdot\sigma = 0$$
$$\rho_f\frac{\partial^2}{\partial t^2}\mathbf{u} + \frac{\mu_f}{\kappa}\frac{\partial}{\partial t}\mathbf{w} + \frac{\tau}{\varepsilon_p}\rho_f\frac{\partial^2}{\partial t^2}\mathbf{w} + \nabla p_f = 0 \quad (3\text{-}2)$$

here, $\mathbf{u}$ is the displacement of the porous material, σ is the total stress tensor (fluid and porous material), $\mathbf{w}$ is the fluid displacement with respect to the porous matrix, ρ_f and μ_f are the fluid's density and viscosity, τ is the tortuosity, ε_p is the porosity, p_f is the fluid pore pressure, κ is the permeability and ρ_{av} the average density. The average density is the total density (porous material plus pore fluid) $\rho_{av} = \rho_d + \varepsilon_p\rho_f$.

Assuming a time-harmonic dependency for the variables, $\mathbf{u}(\mathbf{x},t) = \mathbf{u}(\mathbf{x})e^{i\omega t}$, $\mathbf{w}(\mathbf{x},t) = \mathbf{w}(\mathbf{x})e^{i\omega t}$, the time derivatives can be removed, so the system in Equation 3-2 becomes

$$-\rho_{av}\omega^2\mathbf{u} + \rho_f\omega^2\mathbf{w} - \nabla\cdot\sigma = 0$$
$$-\rho_f\omega^2\mathbf{u} - \omega^2\rho_c(\omega)\mathbf{w} + \nabla p_f = 0 \quad (3\text{-}3)$$

here, the complex density $\rho_c(\omega)$ (Ref. 5) accounts for the tortuosity, porosity and fluid density, and the viscous drag on the porous matrix

$$\rho_c(\omega) = \frac{\tau}{\varepsilon_p}\rho_f + \frac{\mu_f}{i\omega\kappa} \quad (3\text{-}4)$$

HIGH FREQUENCY CORRECTION (BIOT MODEL)

At low frequencies or for small pore sized the flow profile inside the pores can be assumed to be Poiseuille like. In this case the viscosity in Equation 3-4 effectively has a constant value. For increasing frequency the profile changes and a frequency dependent correction factor needs to be taken into account. This is done by selecting the **Biot's high frequency range** option from the **Viscosity model** list. In this case Equation 3-4 is implemented with a frequency-dependent viscosity $\mu_c(f)$ (Ref. 2, Ref. 3, Ref. 5)

$$\mu_c(f) = \mu_f \cdot F_c\left(\sqrt{\frac{f}{f_r}}\right)$$

here, f_r is a reference frequency (SI unit: Hz) which determines the low-frequency range $f << f_r$ and the high-frequency range $f >> f_r$.

The reference frequency f_r can be interpreted as the limit when viscous forces equal inertial forces in the fluid motion. In pore with characteristic size a this happens when the viscous penetration depth is equal to the pore radius.

$$\delta_v = \sqrt{\frac{\mu_f}{2\pi f_r \rho_f}} = a \quad \Rightarrow \quad f_r = \frac{\mu_f}{2\pi a^2 \rho_f}$$

In the low-frequency limit, viscous effects dominate, while in the high-frequency limit, inertial effects dominate fluid motion in the pores (losses occur in the viscous boundary layer). In Biot's low frequency range, $\omega \to 0$ and $F_c = 1$.

In order to account for a frequency dependence on the viscous drag, Biot defined the operator $F_c(\Theta)$ as

$$F_c(\Theta) = \frac{1}{4}\left(\frac{\Theta T(\Theta)}{1 + 2iT(\Theta)/\Theta}\right)$$

here, $T(\Theta)$ is related to the Kelvin functions $\mathrm{Ber}(\Theta)$ and $\mathrm{Bei}(\Theta)$

$$T(\Theta) = \frac{\mathrm{Ber}'(\Theta) + i\mathrm{Bei}'(\Theta)}{\mathrm{Ber}(\Theta) + i\mathrm{Bei}(\Theta)} = \frac{-\sqrt{-i}J_1(\sqrt{-i}\Theta)}{J_0(\sqrt{-i}\Theta)}$$

and J_0 and J_1 are Bessel functions of the first kind. This expression can be recognized as the loss terms in Zwikker-Kosten like equivalent fluid models (Derivation of the equivalent bulk modulus valid for any fluid in the Zwikker-Kosten theory) or the loss models for cylindrical waveguides in the narrow region acoustics or LRF models (About the Narrow Region Acoustics Models) models.

U-P FORMULATION

The formulation in terms of the displacements $\mathbf{u}$ and $\mathbf{w}$ is not optimal from the numerical viewpoint, since it requires to solve for two displacement fields (Ref. 7, Ref. 8, Ref. 9). The Poroelastic Waves interface solves for the fluid pore pressure variable p_f instead of the fluid displacement field $\mathbf{w}$.

The second row in Equation 3-3 is simplified to

$$\mathbf{w} = \frac{1}{\omega^2 \rho_c(\omega)}(\nabla p_f - \rho_f \omega^2 \mathbf{u})$$

so the first row in Equation 3-3 becomes

$$-\rho_{av}\omega^2\mathbf{u}-\frac{\rho_f}{\rho_c(\omega)}(\nabla p_f-\rho_f\omega^2\mathbf{u})-\nabla\cdot\sigma = 0 \tag{3-5}$$

The total stress tensor σ is then divided into the contributions from the elastic porous (drained) matrix and from the pore fluid

$$\sigma(\mathbf{u}, p_f) = \sigma_d(\mathbf{u})-\alpha_B p_f\mathbf{I}$$

here, the identity tensor $\mathbf{I}$ means that the pore pressure p_f only contributes to the diagonal of the total stress tensor σ. The parameter α_B is the so-called Biot-Willis coefficient. The drained, elastic stress tensor is written as $\sigma_d = \mathbf{c}:\varepsilon$ when ε is the strain tensor of the porous matrix, and the elasticity tensor $\mathbf{c}$ contains the drained porous matrix's elastic properties (see the Linear Elastic Material feature in the *Structural Mechanics Module User's Guide*).

Finally, arrange Equation 3-5 in terms of the variables $\mathbf{u}$ and p:

$$-\left(\rho_{av}-\frac{\rho_f^2}{\rho_c(\omega)}\right)\omega^2\mathbf{u}-\nabla\cdot(\sigma_d(\mathbf{u})-\alpha_B p_f\mathbf{I}) = \frac{\rho_f}{\rho_c(\omega)}\nabla p_f \tag{3-6}$$

The next Biot's equation comes from taking the divergence of the second row in Equation 3-3, previously divided by $-\rho_c(\omega)$

$$\omega^2\nabla\cdot\left(\frac{\rho_f}{\rho_c(\omega)}\mathbf{u}\right)+\omega^2\nabla\cdot\mathbf{w}+\nabla\cdot\left(-\frac{1}{\rho_c(\omega)}\right)\nabla p_f = 0 \tag{3-7}$$

Using the expressions for the volumetric strain $\varepsilon_{vol} = \nabla\cdot\mathbf{u}$ and fluid displacement (Ref. 3, Ref. 4),

$$-\nabla\cdot\mathbf{w} = \frac{p_f}{M}+\alpha_B\varepsilon_{vol}$$

Biot's modulus M is calculated from the porosity ε_p, fluid compressibility χ_f, Biot-Willis coefficient α_B and the drained bulk modulus of the porous matrix K_d

$$\frac{1}{M} = \varepsilon_p\chi_f+\frac{\alpha_B-\varepsilon_p}{K_d}(1-\alpha_B) \tag{3-8}$$

so Equation 3-7 simplifies to

$$\omega^2\nabla\cdot\left(\frac{\rho_f}{\rho_c(\omega)}\mathbf{u}\right)-\omega^2\left(\frac{1}{M}p_f+\alpha_B\varepsilon_{vol}\right)+\nabla\cdot\left(-\frac{1}{\rho_c(\omega)}\right)\nabla p_f = 0 \tag{3-9}$$

and Biot's wave equations (Equation 3-6 and Equation 3-9) can be written in terms of the variable $\mathbf{u}$ and p_f as

$$\begin{aligned} -\omega^2\left(\rho_{av}-\frac{\rho_f^2}{\rho_c(\omega)}\right)\mathbf{u}-\nabla\cdot(\sigma_d(\mathbf{u})-\alpha_B p_f\mathbf{I}) &= \frac{\rho_f}{\rho_c(\omega)}\nabla p_f \\ -\frac{\omega^2}{M}p_f+\nabla\cdot-\frac{1}{\rho_c(\omega)}(\nabla p_f-\omega^2\rho_f\mathbf{u}) &= \omega^2\alpha_B\varepsilon_{vol} \end{aligned} \tag{3-10}$$

The saturated (also called Gassmann) modulus can be obtained from the drained bulk modulus K_d, Biot modulus M, and Biot-Willis coefficient α_B as $K_{sat} = K_d + \alpha_B{}^2 M$ (Ref. 5).

Further arranging the first row in Equation 3-10 to fit the formulation in the Elastic Waves interface (Equation 3-1) gives

$$-\omega^2\left(\rho_{av}-\frac{\rho_f^2}{\rho_c(\omega)}\right)\mathbf{u}-\nabla\cdot(\sigma_d(\mathbf{u})-\mathbf{s}_0) = \mathbf{F} \tag{3-11}$$

The body load $\mathbf{F}$ depends on the angular frequency and the gradient of fluid pressure and the fluid pressure acts as a spherical contribution to the diagonal of Cauchy stress tensor

$$\mathbf{F} = \frac{\rho_f}{\rho_c(\omega)}\nabla p_f$$
$$\mathbf{s}_0 = \alpha_B p_f\mathbf{I}$$

Arranging the second row in Equation 3-10 to fit the implementation of the Pressure Acoustics, Frequency Domain interface gives (see Theory Background for the Pressure Acoustics Branch)

$$-\frac{\omega^2}{M}p_f+\nabla\cdot-\left(\frac{1}{\rho_c(\omega)}(\nabla p_f-\mathbf{q}_d)\right) = Q_m \tag{3-12}$$

The monopole domain source Q_m (SI unit: $1/s^2$) and the dipole domain source $\mathbf{q}_d$ (SI unit: N/m^3) depend on the angular frequency ω, the displacement of the porous matrix $\mathbf{u}$, the fluid density and Biot-Willis coefficient α_B

$$Q_m = \omega^2 \alpha_B \varepsilon_{vol}$$
$$\mathbf{q}_d = \omega^2 \rho_f \mathbf{u}$$

BIOT-ALLARD MODEL (VISCOUS AND THERMAL LOSSES)

When both thermal and viscous losses are included the viscosity in Equation 3-4 and the fluid compressibility in Equation 3-8 are replaced by frequency dependent expressions. The losses due to viscosity are considered by the viscosity expression and the losses due to thermal conduction by the fluid compressibility expression, see Ref. 9.

The frequency dependent complex viscosity is given by

$$\mu(\omega) = \mu\left(1 + \frac{4i\omega\tau_\infty^2 \mu \rho_f}{R_f^2 L_v^2 \varepsilon_p{}^2}\right)^{\frac{1}{2}}$$

where the viscous characteristic length L_v has been introduced (it is sometimes referred to as Λ). The frequency dependent complex fluid compressibility is given by

$$\chi_f(\omega) = \frac{\varepsilon_p}{\gamma p_A}\left[\gamma - (\gamma - 1)\left(1 + \frac{8\mu}{i\omega L_{th}^2 \mathrm{Pr} \rho_f}\sqrt{1 + \frac{i\omega L_{th}^2 \mathrm{Pr} \rho_f}{16\mu}}\right)^{-1}\right]$$

where the thermal characteristic length L_{th} has been introduced (it is sometimes referred to as Λ'). The two expressions can be recognized in the JCA equivalent fluid model (Johnson-Champoux-Allard (JCA)) available in Pressure Acoustics.

Different loss models or formulations for the frequency dependent viscosity and fluid compressibility can be entered manually. In order to do so, select the **Biot (viscous losses)** model and then set the fluid **Compressibility** and the fluid **Dynamic viscosity** to **User defined**. In these two fields enter the desired model expression. It can, for example, depend on the frequency, using the variable `freq`.

Boundary Conditions for Poroelastic Waves

Although boundary conditions can be set up for the porous matrix and fluid independently of each other, there exist a few common boundary conditions which

deserve special attention. The following sections refer to the boundary conditions for the system written in Equation 3-11 and Equation 3-12. See derivation in Ref. 7, Ref. 8, and Ref. 9.

POROUS, FREE (SOUND-SOFT BOUNDARY) THEORY

The Porous, Free boundary condition is the default for the porous matrix. It means that the displacement of the porous matrix in Equation 3-11 is unconstrained, so it can move freely without experiencing any loads.

The *sound soft* boundary condition for acoustics creates a boundary condition for Equation 3-12 where the acoustic pressure vanishes, so it sets $p_f = 0$.

FIXED (SOUND-HARD BOUNDARY) EQUATIONS

For simulating a poroelastic medium bounded by a rigid impervious wall, impose a Fixed Constraint node for the porous matrix displacement in Equation 3-11, $\mathbf{u} = 0$, and a *sound-hard* boundary condition for the pore pressure in Equation 3-12:

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

PRESSURE EQUATIONS

For a given fluid pressure p_0 on the boundary, set the pressure in Equation 3-12 to $p_f = p_0$. Since the fluid pressure is set to p_0, the normal stress on the porous matrix in Equation 3-11 reduces to

$$\mathbf{n} \cdot \sigma_d(\mathbf{u}) = \mathbf{n}(\alpha_B - 1)p_0$$

For a rigid porous matrix $\alpha_B = \varepsilon_p$, the load is equivalent to

$$\mathbf{n} \cdot \sigma_d(\mathbf{u}) = \mathbf{n}(\varepsilon_p - 1)p_0$$

and for a soft porous matrix $\alpha_B = 1$, there is no load since

$$\mathbf{n} \cdot \sigma_d(\mathbf{u}) = \mathbf{0}$$

PRESCRIBED DISPLACEMENT EQUATIONS

For a prescribed displacement $\mathbf{u}_0$ at the boundary, set the displacement of the porous matrix in Equation 3-11 as $\mathbf{u} = \mathbf{u}_0$ and assume a sound-hard (impervious) boundary for the fluid pressure in Equation 3-12:

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

PRESCRIBED VELOCITY EQUATIONS

For a prescribed velocity $\mathbf{v}_0$ at the boundary, set the displacement of the porous matrix in Equation 3-11 as

$$\mathbf{u} = \frac{1}{i\omega} \mathbf{v}_0$$

and assume a sound-hard (impervious) boundary condition for the fluid pressure in Equation 3-12

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

PRESCRIBED ACCELERATION EQUATIONS

For a prescribed acceleration $\mathbf{a}_0$ at the boundary, set the displacement of the porous matrix in Equation 3-11 as

$$\mathbf{u} = \frac{1}{-\omega^2} \mathbf{a}_0$$

and assume a sound-hard (impervious) boundary condition for the fluid pressure in Equation 3-12

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

ROLLER EQUATIONS

The roller, or sliding wall boundary, means that the boundary is impervious (sound-hard) to fluid displacements, but it allows tangential displacements of the porous matrix.

The normal displacement of the porous matrix in Equation 3-11 is constrained, but the porous matrix is free to move in the tangential direction

$$\mathbf{n} \cdot \mathbf{u} = 0$$

The impervious (sound hard) boundary condition for the fluid pressure in Equation 3-12 is obtained from

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

SEPTUM BOUNDARY LOAD EQUATIONS

For a prescribed load $\mathbf{F}_A$ at the boundary, suppose that one side of the septum is fixed to the porous matrix and the other side bears the load.

A septum is a very limp and thin impervious layer with surface density ρ_{sep}. Since the septum can be seen as a boundary mass density, this boundary condition is achieved by setting an effective load $\mathbf{F}_S = \mathbf{F}_A + \rho_{sep}\omega^2\mathbf{u}$ on the porous matrix, so the normal stress in Equation 3-11 reduces to

$$\mathbf{n} \cdot (\sigma_d(\mathbf{u}) - \alpha_B p_f \mathbf{I}) = \mathbf{F}_S$$

and a sound-hard (impervious) boundary condition is applied for the fluid pressure in Equation 3-12

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

Postprocessing Variables

Other well known formulations of the poroelastic waves equations exist which use other parameters and variables in their definition (see Ref. 10). To help users that are familiar with these formulations some of the most common parameters have been added as postprocessing variables under the *Biot-Allard coefficients* menu (when clicking the **Replace Expressions** button in plots). The variables included are:

TABLE 3-1: BIOT-ALLARD COEFFICIENTS

COMSOL NAME	DEFINITIONS	NAME
pelw.rho11	$\rho_{11} = \rho_d - \rho_{12}$	Mass coefficient (11)
pelw.rho22	$\rho_{22} = \varepsilon_p \rho_f - \rho_{12}$	Mass coefficient (22)
pelw.rho12	$\rho_{12} = -\rho_a = -\varepsilon_p \rho_f (\tau_\infty - 1)$	Inertial interaction mass coefficient (12)
pelw.rho11_c	$\tilde{\rho}_{11} = \rho_{11} + \tilde{b}(\omega)/(i\omega)$	Complex mass coefficient (11)
pelw.rho22_c	$\tilde{\rho}_{22} = \rho_{22} + \tilde{b}(\omega)/(i\omega)$	Complex mass coefficient (11)
pelw.rho12_c	$\tilde{\rho}_{12} = \rho_{12} - \tilde{b}(\omega)/(i\omega)$	Complex inertial interaction mass coefficient (12)
pelw.A_c	See Ref. 10	Complex elastic coefficient

TABLE 3-1: BIOT-ALLARD COEFFICIENTS

COMSOL NAME	DEFINITIONS	NAME
pelw.Q_c	See Ref. 10	Complex elastic coupling coefficient
pelw.R_c	See Ref. 10	Complex bulk modulus coefficient
pelw.gamma_c	See Ref. 10	Diagonal stress coupling coefficient

References for the Poroelastic Waves Interfaces

1. M. A. Biot, "Theory of Propagation of Elastic Waves in a Fluid-saturated Porous Solid. I. Low-frequency Range," *J. Acoust. Soc. Am.*, vol. 28, pp 168–178, 1956.

2. M. A. Biot, "Theory of Propagation of Elastic Waves in a Fluid-saturated Porous Solid. II. Higher Frequency Range," *J. Acoust. Soc. Am.*, vol. 28, pp 179–191, 1956.

3. M. A. Biot, "Generalized Theory of Acoustic Propagation in Porous Dissipative Media," *J. Acoust. Soc. Am.*, vol. 34, pp 1254–1264, 1962.

4. M. A. Biot, "Mechanics of Deformation and Acoustic Propagation in Porous Media," *J. Appl. Phys.*, vol. 33, pp 1482–1498, 1962.

5. G. Mavko et al., *The Rock Physics Handbook*, 2nd ed., Cambridge University Press, 2009.

6. J. M. Carcione, *Wave Fields in Real Media: Wave Propagation in Anisotropic, Anelastic, Porous and Electromagnetic Media*, 2nd ed. Elsevier (*Handbook of Geophysical Exploration*, vol. 38, Seismic Exploration), 2007.

7. P. Debergue, R. Panneton, and N. Atalla, "Boundary Conditions for the Weak Formulation of the Mixed (u,p) Poroelasticity Problem," *J. Acoust. Soc. Am.*, vol. 106, pp 2383–2390, 1999.

8. N. Atalla, M. A. Hamdi, and R. Panneton, "Enhanced Weak Integral Formulation for the Mixed (u,p) Poroelastic Equations," *J. Acoust. Soc. Am.*, vol. 109, pp 3065–3068, 2001.

9. J.F. Allard and N. Atalla, *Propagation of Sound in Porous Media*, 2nd ed., John Wiley & Sons, 2009.

10. N. Atalla, R. Panneton, and P. Debergue, "A mixed displacemnt-pressure formulation for poroelastic materials," *J. Acoust. Soc. Am.*, vol. 104, pp 1444, 1998.

11. N. Atalla, F. Sgard, and C. K. Amedin, "On the modeling of sound radiation from poroelastic materials," *J. Acoust. Soc. Am.*, vol. 120, pp 1990, 2006.

The Acoustic-Shell Interaction, Frequency Domain Interface

This physics interface requires a Structural Mechanics Module license. For theory and physics interface feature descriptions relating to the Shell interface, see the *Structural Mechanics Module User's Guide*.

The **Acoustic-Shell Interaction, Frequency Domain** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) when adding a physics interface, combines features from the Pressure Acoustics, Frequency Domain and Shell interfaces to connect the acoustics pressure variations in the fluid domain with the structural deformation of a shell boundary. It may for example be used for determining the transmission of sound through a thin elastic structure such as a car hood and analyzing the vibroacoustics of a loudspeaker cone.

Acoustic-structure interaction refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid surface, and the structural acceleration acts on the fluid domain as a normal acceleration across the fluid-structure boundary.

Special physics interface conditions are readily defined at the fluid-shell boundary and set up the fluid loads on the shell boundary and the effect of the structural accelerations on the fluid. The physics interface is only available for 3D geometries, and it is capable of modeling the coupled pressure acoustics and shell vibrations in the frequency domain.

When a predefined **Acoustic-Shell Interaction, Frequency Domain** interface is added from the **Acoustics>Acoustic-Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain** and **Shell** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Acoustic-Structure Boundary**.

On the Constituent Physics Interfaces

The Pressure Acoustics, Frequency Domain interface computes the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. The physics interface solves the Helmholtz equation and is suited for all linear frequency-domain acoustic simulations with harmonic variations of the pressure

field. It includes domain conditions to model losses in a homogenized way, so-called fluid models for porous materials as well as losses in narrow regions. The domain features also include background incident acoustic fields.

The Shell interface is used to model structural shells on 3D faces. Shells are thin flat or curved structures, having significant bending stiffness. The physics interface uses shell elements of the MITC type, which can be used for analyzing both thin (Kirchhoff theory) and thick (Mindlin theory) shells. Geometric nonlinearity can be taken into account. The material is assumed to be linearly elastic.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined coupling features, for example **Acoustic-Shell Interaction, Frequency Domain**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all **Shell** boundaries with **Pressure Acoustics, Frequency Domain** on one side or both sides.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Shell** and **Pressure Acoustics, Frequency Domain** interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Acoustic-Structure Boundary coupling feature is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface.
- The available physics features for Results Evaluation are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*

- Theory Background for the Pressure Acoustics Branch
- Theory for Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*

The Acoustic-Shell Interaction, Transient Interface

This physics interface requires a Structural Mechanics license. For theory and physics interface feature descriptions relating to the Shell interface, see the *Structural Mechanics Module User's Guide*.

The **Acoustic-Shell Interaction, Transient** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) when adding a physics interface, combines features from the Pressure Acoustics, Transient and Shell interfaces to connect the acoustics pressure variations in the fluid domain with the structural deformation of a shell boundary. It may for example be used for determining the transmission of sound through a thin elastic structure such as a car hood and analyzing the vibroacoustics of loudspeaker cone.

Acoustic-structure interaction refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid surface, and the structural acceleration affects the fluid domain as a normal acceleration across the fluid-structure boundary.

Special physics interface conditions are readily defined at the fluid-shell boundary and set up the fluid loads on the shell boundary and the effect of the structural accelerations on the fluid. The physics interface is only available for 3D geometries, and it is capable of modeling the coupled pressure acoustics and shell vibrations in the time domain.

When a predefined **Acoustic-Shell Interaction, Transient** interface is added from the **Acoustics>Acoustic-Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Transient** and **Shell** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Acoustic-Structure Boundary**.

On the Constituent Physics Interfaces

The Pressure Acoustics, Transient interface computes the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. The physics interface solves the scalar wave equation and is suited for time-dependent simulations with arbitrary time-dependent fields and sources. Domain

conditions also include background incident acoustic fields. User-defined sources can be added to, for example, include certain nonlinear effects such as a square pressure dependency of the density variations.

The Shell interface is used to model structural shells on 3D faces. Shells are thin flat or curved structures, having significant bending stiffness. The physics interface uses shell elements of the MITC type, which can be used for analyzing both thin (Kirchhoff theory) and thick (Mindlin theory) shells. Geometric nonlinearity can be taken into account. The material is assumed to be linearly elastic.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined coupling features, for example **Acoustic-Shell Interaction, Transient**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all **Shell** boundaries with **Pressure Acoustics, Transient** on one side or both sides.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Shell** and **Pressure Acoustics, Transient** interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The Acoustic-Structure Boundary coupling feature is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Pressure Acoustics, Transient Interface are listed in the section Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface.
- The available physics features for Results Evaluation are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*.

- Theory Background for the Pressure Acoustics Branch
- Theory for Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*

Modeling with the Acoustic-Structure Interaction Branch

In this section:

Prestressed Acoustic-Structure Interaction

When modeling transducers like microphones or ultrasound horns there are often parts of the structure, like the diaphragm or a bolt, that are under tension or prestressed. In these cases it is essential to include this effect when modeling the traducer. A prestressed part will shift the resonance frequency of the mechanical system and thus the overall vibroacoustic behavior will change.

- *Piezoelectric Tonpilz Transducer with a Prestressed Bolt*: Application Library path **Acoustics_Module/Piezoelectric_Devices/tonpilz_transducer_prestressed**.
- *The Brüel & Kjær 4134 Condenser Microphone*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/bk_4134_microphone**.

Such prestressed acoustic-structure iteration models can be set up and solved fully coupled including all effects. The procedure is as follows:

1. Set up the model including all relevant physics.
2. Couple the physics using the Multiphysics couplings.
3. All acoustic sources, structural loads, and any forcing that are non-static, for example, any acoustic source should be defined using the `linper()` operator. This will ensure that they are used only in the frequency domain part of the study (that one is set to linear perturbation).
4. Add a **Prestressed Analysis, Frequency Domain** study.

5 Note that in the **Stationary** step the acoustics physics interfaces have an orange warning triangle under the **Physics and Variables Selection**. This simply means they will not be used in the stationary study since acoustics is not supporting the study type. Therefore the **Solve for this field** will automatically not be marked under the **Dependent Variables** in the **Solver Configurations**.

6 Note also that the **Include geometric nonlinearity** box is checked in the **Frequency-Domain, Perturbation** step. If it is not checked the prestress effect is lost.

7 Solve the model for the desired frequencies.

A prestressed type of analysis can also be done on pure structural problems in the frequency domain and when searching for eigenfrequencies of structures.

See also Prestressed Structures, Pre-tensioned Bolts, and Mechanical Damping and Losses in the *Structural Mechanics Module User's Guide*

Solving Large Acoustic-Structure Interaction Models

In models that involve acoustic-structure interaction, the strategy for solving large problems involves solving the system in a segregated way. That is, the system is not solved fully coupled in one step but iterations are used solving one physics interface at a time.

To set up such a solver right-click on the Stationary Solver step and select **Segregated**. In the first **Segregated Step** solve for the structural dependent variables (displacement). Set up a second segregated step where you select the pressure dependent variable. Under each of the steps select the solver of choice for solving the single physics interface problem. For example, in the case where only a small structural domain is included, use an iterative multigrid approach for the acoustics (see Solving Large Acoustics Problems Using Iterative Solvers) and a direct solver for the structure.

This strategy is only readily applicable when the coupling between the solid and the acoustic domain is done via Neumann conditions; this is the case for all models where the acoustic domain uses Pressure Acoustics. In, for example, models with Thermoacoustic-Structure interaction the coupling is based on a Dirichlet condition (a pointwise constraint) and required reformulating the continuity condition using weak constraints. This approach does not work either for models coupling

piezoelectric domains, structures, and acoustics, here a fully coupled approach is necessary.

Studies and Solvers and Multigrid in the *COMSOL Multiphysics Reference Manual*

Configuration of Perfectly Matched Layers (PMLs) for Acoustic-Structure Interaction Models

Care should be taken when setting up perfectly matched layers (PMLs) in models that include fluid domains and solid domains. In general models with several different materials, that being different solids or different fluids.

Two configurations exist for the PMLs in these systems:

- The PML regions of two physics (or two different material models) are adjacent and in contact. In this case, a single PML feature should be used for the two PML domains. Set the **Typical wavelength from** to **User defined** and enter a value based on the longest wavelength. For example, for an air-steel system enter: `5770[m/s]/freq`

  The shorter wavelengths should be captured by increasing the mesh resolution and/or increase the **PML scaling curvature parameter**. This will ensure optimal damping of all wavelengths in the system. In any case control the convergence of the results by increasing the number of mesh layers in the PML.

- Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*.
- See Perfectly Matched Layers (PMLs) in Modeling with the Pressure Acoustics Branch in the *Acoustics Module User's Guide*.

- The PML regions of two physics (or two different material models) do not touch. In this case, use a PML feature for each PML domain. Keep the default setting of **Typical wavelength from** to **Physics interface** and select the relevant physics interface under **Physics**.

The reason that two PML features cannot be used when the PML domains a adjacent to each other is that the scaling inside the PMLs will differ (because of different speed

of sounds). This will lead to an unphysical situation at the interface between the two PMLs which results in errors in the solution. Spurious waves may be generated.

See the *Acoustic-Structure Interaction with a Perfectly Matched Layer (PML)* model in the Application Gallery on www.comsol.com. It is located here: https://www.comsol.com/model/acoustic-structure-interaction-with-a-perfectly-matched-layer-pml-23521

The Pipe Acoustics Interfaces

In this section:

- The Pipe Acoustics, Frequency Domain Interface
- The Pipe Acoustics, Transient Interface
- For links to all the physics features, go to Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces

These interfaces require both the Pipe Flow Module and the Acoustics Module.

The Pipe Acoustics, Frequency Domain Interface

The **Pipe Acoustics, Frequency Domain (pafd)** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) when adding a physics interface, is used to compute the acoustic pressure and velocity variations when modeling the propagation of sound waves in flexible pipe systems. The governing equations are formulated in a general way to include the possibility of a stationary background flow. The physics interface can for example be used to compute the propagation of sound waves in HVAC systems, other large piping systems, or simply in an organ pipe.

In the frequency domain all sources and variations are assumed to be harmonic. The solved equations assume that the propagating waves are plane. The propagation of higher-order modes that exist above their cut-off frequency, dictated by the pipe cross section, is not modeled.

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduce the equations to a 1D component with scalar dependent variables. The physics interface is available in 3D on edges and points, and in 2D on boundaries and points.

When this physics interface is added, these default nodes are also added to the **Model Builder**—**Fluid Properties**, **Pipe Properties**, **Closed**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and point conditions. You can also right-click **Pipe Acoustics, Frequency Domain** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `pafd`.

SOUND PRESSURE LEVEL SETTINGS

The zero level on the dB scale varies with the type of fluid. That value is a reference pressure that corresponds to 0 dB. This variable occurs in calculations of the sound pressure level L_p based on the root mean square (rms) pressure p_rms, such that

$$L_\mathrm{p} = 20\log\left(\frac{p_\mathrm{rms}}{p_\mathrm{ref}}\right) \qquad \text{with} \qquad p_\mathrm{rms} = \sqrt{\frac{1}{2}pp^*}$$

where p_ref is the reference pressure and the star (*) represents the complex conjugate. This is an expression valid for the case of harmonically time-varying acoustic pressure p.

Select a **Reference pressure for the sound pressure level** based on the fluid type:

- **Use reference pressure for air** to use a reference pressure of 20 µPa ($20{\cdot}10^{-6}$ Pa).
- **Use reference pressure for water** to use a reference pressure of 1 µPa ($1{\cdot}10^{-6}$ Pa).
- **User-defined reference pressure** to enter a reference pressure $p_\mathrm{ref,\,SPL}$ (SI unit: Pa). The default value is the same as for air, 20 µPa.

For postprocessing, plot the sound pressure level `pafd.Lp`, which depends on the selected reference pressure. You can also plot the rms intensity magnitude `pafd.I_rms` or the instantaneous intensity magnitude `pafd.I_inst`.

DEPENDENT VARIABLES

This section is used to define the dependent variables (fields) for **Pressure** p (SI unit: Pa) and **Tangential velocity** u (SI unit: m/s). If required, edit the name, but dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. It controls the element types used in the finite element formulation.

- Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces
- Theory for the Pipe Acoustics Interfaces

The Pipe Acoustics, Transient Interface

The **Pipe Acoustics, Transient (patd)** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) when adding a physics interface, is used to compute the acoustic pressure and velocity variations when modeling the propagation of sound waves in flexible pipe systems. The governing equations are formulated in a general way to include the possibility of a stationary background flow. The physics interface can for example be used to compute the propagation of sound waves in HVAC systems, other large piping systems, or simply in an organ pipe.

The solved equations assume that the propagating waves are plane. The propagation of higher-order modes that exist above their cut-off frequency, dictated by the pipe cross section, is not modeled.

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduce the equations to a 1D component with scalar dependent variables. The physics interface is available in 3D on edges and points, and in 2D on boundaries and points.

When this physics interface is added, these default nodes are also added to the **Model Builder**—**Fluid Properties**, **Pipe Properties**, **Closed**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and point conditions. You can also right-click **Pipe Acoustics, Transient** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to

different physics interfaces, the `name` string must be unique. Only letters, numbers and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `patd`.

TRANSIENT SOLVER SETTINGS

Select the **Time stepping** (method) as **Manual** (default and recommended) or **Automatic/free** and then enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

The rest of the settings are the same as for The Pipe Acoustics, Frequency Domain Interface.

- Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces
- Theory for the Pipe Acoustics Interfaces

Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces

The Pipe Acoustics, Frequency Domain Interface and The Pipe Acoustics, Transient Interface have these edge, boundary, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Closed
- End Impedance
- Fluid Properties
- Initial Values
- Pipe Properties
- Pressure
- Velocity
- Volume Force is described for the Pipe Flow interface in the *Pipe Flow User's Guide*.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Initial Values

The **Initial Values** node adds initial values for the pressure and tangential velocity that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter values or expressions for the initial value of the **Pressure** p (SI unit: Pa) and the **Tangential Velocity** u (SI unit: m/s).

Fluid Properties

The **Fluid Properties** node adds the momentum and continuity equations solved by the physics interface, except for volume forces which are added by the Volume Force node. The node also provides an interface for defining the material properties of the fluid.

Volume Force is described for the Pipe Flow interface in the *Pipe Flow User's Guide*.

MODEL INPUTS

Enter a value for the and **Background mean flow pressure** p_0 (SI unit: Pa). This pressure will also serve as input the material parameters that depend on the pressure.

BACKGROUND VELOCITY

Enter a value or expression for the **Background mean flow velocity** u_0 (SI unit: m/s)

Physically sound background variables for the pressure p_0 and velocity u_0 can be obtained by solving a Pipe Flow model on the same geometry.

PHYSICAL PROPERTIES

Select a **Fluid model**—**Linear elastic** (the default).

The default **Density** ρ (SI unit: kg/m^3) and **Speed of sound** c_s (SI unit: m/s) use the values **From material**. For **User defined** enter different values or expressions.

Pipe Properties

The **Pipe Properties** node is used to define the pipe shape, pipe model, wall drag force, and flow profile correction factor.

PIPE SHAPE

Select a pipe shape from the list—**Not set** (the default), **Circular**, **Square**, **Rectangular**, or **User defined**.

- For **Circular** enter a value or expression for the **Inner diameter** d_i (SI unit: m). The default is 10 cm (0.01 m).
- For **Square** enter a value or expression for the **Inner width** w_i (SI unit: m). The default is 5 cm (0.005 m).
- For **Rectangular** enter a value or expression for the **Inner width** w_i (SI unit: m; the default is 5 cm) and **Inner height** h_i (SI unit: m; the default is 10 cm).
- For **User defined** enter a value or expression for the **Cross sectional area** A (SI unit: m^2; the default is 0.01 m^2) and **Wetted perimeter** Z (SI unit: m; the default is 0.4 m).

PIPE MODEL

Select a **Pipe model**—**Incompressible cross section** (the default), **Zero axial stress**, **Anchored at one end**, or **Anchored at both ends**.

When **Zero axial stress**, **Anchored at one end**, or **Anchored at both ends** is chosen, select an option from the **Young's modulus** E (SI unit: Pa) and **Wall thickness** Δw lists—**Not set** (the default) or **User defined**. For **User defined** in either case, enter different values or expressions.

For **Anchored at one end** or **Anchored at both ends** also select an option from the **Poisson's ratio** υ (dimensionless) list—**Not set** (the default) or **User defined**. For **User defined** enter a value or expression.

WALL DRAG FORCE

Enter a value or expression for τ_w (SI unit: N/m^2). The default is 0 N/m^2.

FLOW PROFILE CORRECTION FACTOR

Enter a value or expression for β (dimensionless). The default is 1. For most practical applications this correction factor is 1 as the propagating waves are assumed plane and uniform. This value should typically be changed if a wall drag force is introduces or if a non-plug-flow background flow field is used. The flow profile correction factor is defined as

$$\beta = \left(\int(\tilde{\mathbf{u}} + \tilde{\mathbf{u}}_0)^2 dA\right) / \left(\int(\tilde{\mathbf{u}} + \tilde{\mathbf{u}}_0) dA\right)^2$$

where the velocity field is the actual one that exists in the pipe cross section. The factor thus measures the deviation from a flat background flow profile (plug flow) and a plane propagating acoustic wave.

Flow Profile Correction Factor β

Closed

Use the **Closed** node to impose zero velocity. This is the default condition added on all end points.

Theory for the Pipe Acoustics Boundary Conditions

Pressure

Use the **Pressure** node to define the boundary pressure at the pipe ends.

PRESSURE

Enter a value or expression for the **Pressure** p (SI unit: Pa). The default is 0 Pa.

In the frequency domain p represents the amplitude and phase (if it is complex valued) of a harmonic pressure source.

In the time domain enter an expression for the pressure p, for example, a forward moving sinusoidal wave of amplitude 1 Pa can be written as `1[Pa]*sin(omega*t-k*x)`, where `omega` and `k` are parameters defining the angular frequency and wave number.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Theory for the Pipe Acoustics Boundary Conditions

Velocity

Use the **Velocity** node to prescribe a velocity at the pipe ends.

VELOCITY

Enter a value or expression for the velocity u_{in} (SI unit: m/s) at the inlet and/or outlet of a pipe. The default is 0 m/s. The velocity u_{in} is defined relative to background flow u_0 and thus in the tangential coordinate system. Enable the **Show physics symbols** from the **Graphics and Plot Windows** menu on the Preference dialog box in order to visualize the boundary or edge tangent direction. Click the **Fluid Properties** node to see the tangents as a red arrows.

- Theory for the Pipe Acoustics Boundary Conditions
- The Preferences Dialog Box in the *COMSOL Multiphysics Reference Guide*

End Impedance

Use the **End Impedance** node to model conditions at the end of a pipe. The condition can either model an infinite pipe and thus represent the characteristic impedance of the pipe system at that point. This results in a zero reflection condition. Alternatively the condition can represent the radiation impedance of an open pipe in either a flanged (in an infinite baffle) or unflanged (a pipe ending in free open space). The end impedance

can also be user-defined and could represent modeled or experimental values for a specific pipe configuration.

The wave speed c in the pipe can be different from the speed of sound c_s in an open space. It then depends on the elastic properties of the pipe structure. It is defined in Equation 3-17 in the Governing Equations section.

The wave speed can be evaluated as `sqrt(1/patd.invc2)` or `sqrt(1/pafd.invc2)` during the analysis and results stage.

END IMPEDANCE

Select an **Impedance model**.

For The Pipe Acoustics, Transient Interface select **Infinite pipe (low Mach number limit)** (the default) or **User defined**. The **Infinite pipe (low Mach number limit)** models and infinite pipe by specifying the characteristic impedance at that point. This condition creates a non-reflecting boundary. The expression is valid for small values of the Mach number
$Ma = u_0/c$. For **User defined** enter an **End impedance** Z_{end} (SI unit: Pa·s/m). The default is `patd.rho*(sqrt(1/patd.invc2))` which is $\rho \cdot c$.

For The Pipe Acoustics, Frequency Domain Interface select from the following—**Infinite pipe (low Mach number limit)** (the default), **Infinite pipe**, **Flanged pipe, circular**, **Flanged pipe, rectangular**, **Unflanged pipe, circular (low ka limit)**, **Unflanged pipe, circular**, or **User defined**.

- For **Infinite pipe** enter a **Wave number** k (SI unit: rad/m). The default expression is `pafd.omega*(sqrt(pafd.invc2))`. This end impedance models the infinite pipe using the full (nonlinear) dispersion relation. It is valid for all Mach numbers but require the additional input of the wave number k.
- For **Flanged pipe, circular** enter an **Inner radius** a (SI unit: m). The default expression is `pafd.dh/2`. This end impedance models the radiation impedance of a circular pipe terminated in an infinite baffle. It is an exact analytical result valid for all frequencies and pipe radii. In the low frequency limit it reduces to the classical results:

$$Z_{\text{end}} = \rho c\left(\frac{1}{2}(ka)^2 + i(0.8216 \cdot ka)\right)$$

- For **Flanged pipe, rectangular** enter an **Inner width** w_i (SI unit: m). The default is 5 cm (0.005 m). Also enter an **Inner height** h_i (SI unit: m). The default is 10 cm (0.01 m). This end impedance models the radiation impedance of a pipe of rectangular cross section terminated in an infinite baffle. The model is only valid in the low frequency range where $kw_i \ll 1$ and $kh_i \ll 1$.
- For **Unflanged pipe, circular (low ka limit)** or **Unflanged pipe, circular** enter an **Inner radius** a (SI unit: m). The default expression is `pafd.dh/2`. These two end impedance models prescribe the radiation impedance of an unflanged circular pipe (a pipe ending in free open space). The first model is the classical low frequency approximation valid for $ka \ll 1$. While the second model extends the frequency range to $ka < 3.83$.
- For **User defined** enter an **End impedance** Z_{end} (SI unit: Pa·s/m). The default expression is `pafd.rho*(sqrt(1/pafd.invc2))` which is $\rho \cdot c$.

For a detailed review of the end impedance models see: Theory for the Pipe Acoustics Boundary Conditions.

Theory for the Pipe Acoustics Interfaces

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduces the equations to a 1D component with scalar dependent variables. The present theory assumes no thermal conduction and thus no losses due to thermal conduction (isentropic sound propagation). The Pipe Acoustics, Transient and the Pipe Acoustics, Frequency Domain interfaces require both the Pipe Flow Module and the Acoustics Module.

In this section:

Governing Equations

The continuity equation derived for a control volume is given by

$$\frac{\partial(A\rho)}{\partial t} + \nabla \cdot (A\rho\mathbf{u}) = 0 \tag{3-13}$$

and the corresponding momentum balance equation is

$$\frac{\partial(\rho A\mathbf{u})}{\partial t} + \nabla(\rho A\beta\mathbf{u}^2) = -A\nabla p + \tau_w Z + A\mathbf{F} \tag{3-14}$$

where Z is the inner circumference of the pipe and $A = A(x,p,...)$ is the inner wetted cross-sectional area, $\mathbf{u}$ is the area-averaged mean velocity, which is also defined in the tangential direction $\mathbf{u} = u\mathbf{e}_t$, p is the mean pressure along the pipe, τ_w is the wall drag force, and $\mathbf{F}$ is a volume force. The gradient is taken in the tangential direction $\mathbf{e}_t$. The

term β is a flow profile correction factor relating the mean of the squared total velocity to the square of the mean velocity. Such that

$$u = \frac{1}{A}\int \tilde{\mathbf{u}} \cdot \mathbf{dA} \qquad p = \frac{1}{A}\int \tilde{p}\, dA \qquad \beta = \left(\frac{1}{A}\left(\int \tilde{u}^2 dA\right)\right)/u^2 \tag{3-15}$$

where

$$\tilde{p} = \tilde{p}(\mathbf{x}) \text{ and } \tilde{u} = \tilde{u}(\mathbf{x})$$

are the local non-averaged parameters. Again p and u are the area-averaged dependent variables.

LINEARIZATION

The governing equations are now linearized, that is, all variables are expanded to first order assuming stationary zero (0th) order values (steady-state background properties). The acoustic variations of the dependent variables are assumed small and on top of the background values. This is done according to the following scheme:

$$\begin{aligned}
\mathbf{u}(x,t) &= \mathbf{u}_0(x) + \mathbf{u}_1(x,t) \\
p(x,t) &= p_0(x) + p_1(x,t) \\
\rho(x,t) &= \rho_0(x) + \rho_1(x,t) \\
A(x,t) &= A_0(x) + A_1(x,t)
\end{aligned}$$

where A_0 is often only function of $\mathbf{x}$; however, A_0 can be changed by external factors such as heating or structural deformation, thus the time dependency. The 1st order terms represent small perturbations on top of the background values (0th order). They are valid for

$$\rho_1 \ll \rho_0 \qquad p_1 \ll \rho_0 c_0^2 \qquad |\mathbf{u}_1| \ll c_0 \qquad A_1 \ll A_0$$

Moreover, the perturbations for the fluid density and cross-sectional area are expanded to first order in p_0 in a Taylor series such that

$$\rho_1 = \rho - \rho_0 = (p - p_0)\left[\frac{\partial \rho}{\partial p}\right]_s\Bigg|_0$$

$$A_1 = A - A_0 = (p - p_0)\left[\frac{\partial A}{\partial p}\right]_s\Bigg|_0$$

where the subscript s refers to constant entropy; that is, the processes are isentropic. The relations for the fluid compressibility and the cross-sectional area compressibility are

$$\beta_0 = \frac{1}{K_0} = \frac{1}{K_s} = \frac{1}{\rho_0}\left[\frac{\partial \rho}{\partial p}\right]_s\Bigg|_0 = \frac{1}{\rho_0}\frac{1}{c_s^2}$$

$$\beta_A = \frac{1}{A_0}\left[\frac{\partial A}{\partial p}\right]_s\Bigg|_0 = \frac{1}{K_A}$$

Here, β_0 is the fluid compressibility at the given reference pressure p_0, the isentropic bulk speed of sound is denoted c_s, and ρ_0 is the fluid density at the given reference temperature and reference pressure. β_A is the effective compressibility of the pipe's cross-sectional A_0 due to changes in the inner fluid pressure. The bulk modulus K is equal to one over the compressibility.

Inserting the above expansions into the governing equations (Equation 3-13 and Equation 3-14) and retaining only 1st-order terms yield the pipe acoustics equations including background flow. These are:

$$\begin{gathered}
A_0\frac{1}{c^2}\frac{\partial p_1}{\partial t} + \nabla\cdot\left(A_0\rho_0\left(\mathbf{u}_1 + \frac{\mathbf{u}_0}{\rho_0 c^2}p_1\right)\right) = 0 \\
\rho_0 A_0\left(\frac{\partial \mathbf{u}_1}{\partial t} + \frac{\mathbf{u}_0}{\rho_0 c^2}\frac{\partial p_1}{\partial t}\right) + \nabla\left(A_0\beta\frac{\mathbf{u}_0^2}{c^2}p_1 + 2\rho_0 A_0\beta\mathbf{u}_0\mathbf{u}_1\right) \\
+ A_0(\nabla p_1 + p_1\beta_A\nabla p_0) + \tau_w Z + A\mathbf{F} = 0 \\
\frac{1}{c^2} = \rho_0(\beta_0 + \beta_A) = \rho_0\left(\frac{1}{K_0} + \frac{1}{K_A}\right) = \frac{1}{c_s^2} + \frac{\rho_0}{K_A}
\end{gathered} \tag{3-16}$$

where c is the effective speed of sound in the pipe (it includes the effect due to the elastic properties of the pipe defined through K_A). The bulk modulus for the cross-sectional area K_A is given by the pipe material properties according to the so-called Korteweg formula (see Ref. 2). For a system with rigid pipe walls $c_s = c$ as K_A tends to infinity.

Using the fact that the velocity is taken along the tangential direction $\mathbf{e}_t$ the governing equations are rewritten in terms of the scalar values u and p and projected onto the tangent. The 0 subscript is dropped on the density and area and the 1 subscript is also dropped on the dependent variables.

$$
\begin{aligned}
&A\frac{1}{c^2}\frac{\partial p}{\partial t}+\nabla_t\left(A\rho\left(u+\frac{u_0}{\rho c^2}p\right)\right)\cdot\mathbf{e}_t = 0\\
&\rho A\left(\frac{\partial u}{\partial t}+\frac{u_0}{\rho c^2}\frac{\partial p}{\partial t}\right)+\nabla_t\left(A\beta\frac{u_0^2}{c^2}p+2\rho A\beta u_0 u\right)\cdot\mathbf{e}_t\\
&\qquad +A(\nabla_t p+p\beta_A\nabla_t p_0)\cdot\mathbf{e}_t+\tau_w Z+A(\mathbf{F}\cdot\mathbf{e}_t) = 0\\
&\frac{1}{c^2} = \rho(\beta_0+\beta_A) = \rho\left(\frac{1}{K_0}+\frac{1}{K_A}\right) = \frac{1}{c_s^2}+\frac{\rho}{K_A}
\end{aligned}
\tag{3-17}
$$

where ∇_t is the tangential derivative, τ_w is the tangential wall drag force (SI unit: N/m^2) and $\mathbf{F}$ is a volume force (SI unit: N/m^3).

GOVERNING EQUATIONS

Pipe Acoustics, Transient Interface

Finally, the expression for the time derivative of the pressure in the momentum equation is replaced by spatial derivatives using the continuity equation. This yields the equations solved in the Pipe Acoustics, Transient interface:

$$
\begin{aligned}
&A\frac{1}{c^2}\frac{\partial p}{\partial t}+\nabla_t\left(A\rho\left(u+\frac{u_0}{\rho c^2}p\right)\right)\cdot\mathbf{e}_t = 0\\
&\rho A\frac{\partial u}{\partial t}+\nabla_t\left(A\beta\frac{u_0^2}{c^2}p+2\rho A\beta u_0 u\right)\cdot\mathbf{e}_t+u_0\nabla_t\left(A\rho\left(u+\frac{u_0}{\rho c^2}p\right)\right)\cdot\mathbf{e}_t\\
&\qquad +A(\nabla_t p+p\beta_A\nabla_t p_0)\cdot\mathbf{e}_t+\tau_w Z+A(\mathbf{F}\cdot\mathbf{e}_t) = 0\\
&\frac{1}{c^2} = \rho(\beta_0+\beta_A) = \rho\left(\frac{1}{K_0}+\frac{1}{K_A}\right) = \frac{1}{c_s^2}+\frac{\rho}{K_A}
\end{aligned}
\tag{3-18}
$$

Pipe Acoustics, Frequency Domain Interface

In the frequency domain all variables are assumed to be time harmonic such that

$$
\begin{aligned}
p &= \tilde{p}(\mathbf{x})e^{i\omega t}\\
u &= \tilde{u}(\mathbf{x})e^{i\omega t}
\end{aligned}
\tag{3-19}
$$

inserting this into the governing Equation 3-18 (and dropping the tilde) yields the equations solved in the Pipe Acoustics, Frequency Domain interface:

$$i\omega\frac{A}{c^2}p + \nabla_t\left(A\rho\left(u + \frac{u_0}{\rho c^2}p\right)\right)\cdot \mathbf{e}_t = 0$$

$$i\omega\rho A u + \nabla_t\left(A\beta\frac{u_0^2}{c^2}p + 2\rho A\beta u_0 u\right)\cdot \mathbf{e}_t + u_0\nabla_t\left(A\rho\left(u + \frac{u_0}{\rho c^2}p\right)\right)\cdot \mathbf{e}_t \qquad (3\text{-}20)$$
$$+ A(\nabla_t p + p\beta_A\nabla_t p_0)\cdot \mathbf{e}_t + \tau_w Z + A(\mathbf{F}\cdot \mathbf{e}_t) = 0$$

$$\frac{1}{c^2} = \rho(\beta_0 + \beta_A) = \rho\left(\frac{1}{K_0} + \frac{1}{K_A}\right) = \frac{1}{c_s^2} + \frac{\rho}{K_A}$$

where $\omega = 2\pi f$ is the angular frequency and f is the frequency.

Theory for the Pipe Acoustics Boundary Conditions

PRESSURE, OPEN, AND CLOSED CONDITIONS

The simplest boundary conditions to specify are to prescribe the pressure or the velocity at the pipe ends. These result in the Pressure condition

$$p = p_{in}$$

and the Velocity condition

$$u = u_{in}$$

and can be set independently of each other leaving the other dependent variable free. A special subclass of the velocity condition is the Closed condition where

$$u = 0$$

this corresponds to the sound-hard wall condition in pressure acoustics. It is also assumed here that $u_0 = 0$ at a closed boundary.

END IMPEDANCE CONDITION

At the end of pipes the relation between the pressure and the velocity can be defined in terms of an end impedance Z_{end}. The End Impedance condition is in the Pipe Acoustics interface given by

$$A\left(\rho u + \frac{u_0}{c^2}p\right) = A\left(\rho\frac{1}{Z_{end}} + \frac{u_0}{c^2}\right)p \qquad (3\text{-}21)$$

where $Z_{\text{end}} = p/u$ (SI unit: (Pa·s)/m). Different models for the end impedance exist in the Pipe Acoustics interfaces. The variety depend on if the transient or the frequency domain equations are solved.

Transient End-Impedance Models

In the transient version of the physics interface the end impedance can be user-defined or set to mimic an infinite long pipe for low Mach number background flow conditions. In this case it is assumed that the pipe continues with constant cross section A and that there is no external body force $\mathbf{F}$ and drag τ_w. Because the acoustic waves are, by design, always normal to the pipe ends. In order to define the relation between the pressure and the velocity (the impedance) the dispersion relation for a plane wave needs to be determined.

In order to do so insert the assumed plane wave form

$$p = \text{Re}(\tilde{p}e^{i(\omega t - kx)})$$
$$u = \text{Re}(\tilde{u}e^{i(\omega t - kx)})$$

into the governing Equation 3-18 and solve for the desired relations. After some manipulation this results in

$$\frac{1}{Z_{\text{end}}} = \frac{u}{p} = \frac{1}{c^2}\frac{1}{\rho}\left(\frac{\omega}{k} - u_0\right)$$

with the dispersion relation

$$\frac{\omega}{k} = \beta u_0 \pm c\sqrt{\beta(\beta-1)\left(\frac{u_0}{c}\right)^2 + \left(1 - \frac{1}{k}\beta_A \nabla p_0\right)} \tag{3-22}$$

This dispersion relation is nonlinear in k. In the limit where β_A tends to zero and for small Mach numbers M ($= u_0/c$) the expression is expanded to

$$\frac{\omega}{k} \cong \beta u_0 \pm c\left(1 + \frac{1}{2}\beta(\beta-1)\left(\frac{u_0}{c}\right)^2\right)$$

Hence, the infinite pipe (low Mach number limit) end impedance relation reads

$$\frac{1}{Z_{\text{end}}} = \frac{1}{c^2}\frac{1}{\rho}\left[(\beta-1)u_0 \pm c\left(1 + \frac{1}{2}\beta(\beta-1)\left(\frac{u_0}{c}\right)^2\right)\right] \tag{3-23}$$

where the sign in front of c depends on the direction of propagation of the wave.

Frequency Domain End-Impedance Models

In the frequency domain many engineering relations exist for the end impedance or radiation impedance of a pipe or waveguide. Most of the relations apply only to a specific geometry or frequency range. The relations available in the Pipe Acoustics, Frequency Domain interface are:

- *Infinite pipe (low Mach number limit)*: This is the same relation as for the transient study and the end impedance is given by Equation 3-23. This can be thought of as the characteristic impedance of the tube.
- *Infinite pipe*: This relation uses the full dispersion relation given in Equation 3-22 and yields the expression

$$\frac{1}{Z_{\text{end}}} = \frac{1}{c^2}\frac{1}{\rho}\left[(\beta-1)u_0 \pm c\sqrt{\beta(\beta-1)\left(\frac{u_0}{c}\right)^2+\left(1-\frac{1}{k}\beta_A\nabla p_0\right)}\right] \tag{3-24}$$

where the wave number k at the right hand side is a user input. In the frequency domain a good estimate for this quantity is simply ω/c.

- *Flanged pipe, circular*: In the case of a circular pipe terminated in an infinite baffle (a flanged pipe) an analytical expression exists for the radiation impedance (see Ref. 1),

$$Z_{\text{end}} = \rho c\left(1-\frac{2J_1(2ka)}{2ka}+i\frac{2H_1(2ka)}{2ka}\right) \tag{3-25}$$

where J_1 is the Bessel function of order 1, H_1 is the Struve function of order 1, a is the pipe radius, and k is the wave number. The Struve function is approximated according to Ref. 3 by

$$H_1(x) \approx \frac{2}{\pi} - J_0(x) + \left(\frac{16}{\pi}-5\right)\frac{\sin x}{x} + \left(12-\frac{36}{\pi}\right)\frac{1-\cos x}{x^2} \tag{3-26}$$

In the low frequency limit (small ka) Equation 3-25 reduces to the classical expression for the radiation impedance

$$Z_{\text{end}} = \rho c\left(\frac{1}{2}(ka)^2 + i(0{,}8216\cdot ka)\right) \tag{3-27}$$

- *Flanged pipe, rectangular*: In the case of a pipe of rectangular cross-section (with sides w_i and h_i) terminated in an infinite baffle (a flanged pipe) the radiation impedance can be approximated by

$$Z_{\text{end}} = \frac{\rho c}{2\pi}\left(k^2(w_i h_i)^2 + ik(w_i h_i)^{3/2} f\left(\frac{w_i}{h_i}\right)\right) \qquad kw_i \ll 1, \quad kh_i \ll 1$$
$$f(x) = 2x^{1/2}\sinh^{-1}x^{-1} + 2x^{-1/2}\sinh^{-1}x + \frac{2}{3}x^{3/2} + \frac{2}{3}x^{-3/2} - \frac{2}{3}(x + x^{-1})^{3/2} \tag{3-28}$$

see Ref. 4 and Ref. 5.

- *Unflanged pipe, circular (low ka limit)*: In the case of a circular pipe of radius a ending in free air the classical low ka limit for the radiation impedance is given by

$$Z_{\text{end}} = \rho c\left(\frac{1}{4}(ka)^2 + i(0{,}6133 \cdot ka)\right) \qquad ka \ll 1 \tag{3-29}$$

see Ref. 1 and Ref. 5.

- *Unflanged pipe, circular*: A solution for the unflanged pipe exists for the case when $ka \ll 3{,}83 = 1{,}22\pi$, it is presented in Ref. 6 and is based on solving the Wiener-Hopf integral, it reads

$$Z_{\text{end}} = \rho c\frac{1+R}{1-R} \qquad R = |R|e^{2ika\delta}$$
$$|R| = e^{-(ka)^2/2}\left(1 + \frac{1}{6}(ka)^4\left[\ln\left(\left(\frac{1}{\gamma ka}\right) + \frac{19}{12}\right)\right]\right) \quad \gamma = e^{0{,}5772} \quad ka < 1 \tag{3-30}$$
$$|R| = \sqrt{\pi ka}e^{-ka}\left(1 + \frac{3}{32}\frac{1}{(ka)^2}\right) \quad 1 < ka < 3{,}83$$

where δ is an interpolation function found by numerical integration for $ka = 0$, $\delta = 0.6133$.

Common for the last four radiation impedance relations is that they do only apply when there is no background flow present $u_0 = 0$ (or at least when it is very small).

Solving Transient Problems

When solving transient acoustic problems where the wave shape is not necessarily harmonic it might be necessary to resolve its spatial variations with a fine mesh, say with a minimal scale dx. Now, in order for the numerical solution of the temporal development of the acoustic field to be good it is necessary to restrict the maximal time steps dt taken by the solver. The condition is known as the CFL condition (Courant–Friedrichs–Lewy condition). For transient acoustic problems it is defined as

$$C = c \cdot \frac{dt}{dx}$$

where C is the Courant number, and c is the velocity.

For applications where all the shape functions are quadratic the Courant number should be around 0.2. This condition restricts any acoustic disturbances to propagate more than 20% of the mesh size dx during one time step dt. In the Pipe Acoustics interface where a mixed formulation exists, with linear elements for the pressure and quadratic elements for the velocity, the condition might have to be tightened such that $C < 0.2$.

For an example where the CFL condition is used see *Water Hammer*: Application Library path **Pipe_Flow_Module/Verification_Examples/water_hammer_verification**.

Cut-off Frequency

The Pipe Acoustics interface assumes plane wave propagation. This means that it cannot model the propagation of the higher order modes that can propagate above their cut-off frequency f^c. In a rectangular pipe of cross section width w_i and height h_i the cut-off frequency is

$$f^c_{mn} = \frac{1}{2}c\sqrt{\left(\frac{m}{w_i}\right)^2 + \left(\frac{n}{h_i}\right)^2}$$

In a pipe of circular cross section (with radius a) the cut-off frequency is

$$f^c_{mn} = \frac{\alpha'_{mn}c}{2\pi a}$$

where α'_{mn} is the n'th zero of the differential of the Bessel function $J'_m(x)$ or order m. The first few values are $\alpha'_{01} = 0$, $\alpha'_{02} = 3.83$, $\alpha'_{11} = 1.84$, and $\alpha'_{21} = 3.05$ (see Ref. 1 and Ref. 5 for further details).

Flow Profile Correction Factor β

The flow profile correction factor β accounts for the ratio of the integrated local square velocity field to the square of the integrated local velocity field (see Equation 3-15). It is defined in terms of the total velocity field (background plus acoustic variations).

In the case of no-background flow ($u_0 = 0$) β is 1 in the absence of a wall drag coefficient, as only plane wave modes propagate. If a wall drag force is introduced, to

model some loss mechanism, β starts to differ slightly from 1. This can for example be losses introduced to model viscous and thermal effects in narrow pipes.

In the presence of a background u_0 the factor β can be set different 1 in order to model a non-flat velocity profile inside the tube. The value of b (and the actual shape of the background field) influences the convective momentum transfer balances. The places where β enter the governing equations are multiplied with either the Mach number or the Mach number squared, indicating that the effects become important for an increasing background flow.

References for the Pipe Acoustics Interfaces

1. D.T. Blackstock, ***Fundamentals of Physical Acoustics***, John Wiley & Sons, 2000.

2. M.S. Ghidaoui, M. Zhao, D.A. McInnis, and D.H. Axworthy, "A Review of Water Hammer Theory and Practice," *Applied Mechanics Reviews*, ASME, 2005.

3. R.M. Aarts and A.J.E.M. Janssen, "Approximation of the Struve Function H1 Occurring in Impedance Calculations," *J. Acoust. Soc. Am.*, vol. 113, pp. 2635–2637, 2003.

4. O.A. Lindemann, "Radiation Impedance of a Rectangular Piston at Very Low Frequencies," *J. Acoust. Soc. Am.*, vol 44, pp. 1738–1739, 1968.

5. A.D. Pierce, *Acoustics: An Introduction to its Physical Principles and Applications*, Acoustics Society of America, 1994.

6. H. Levine and J. Schwinger, "On the Radiation of Sound from an Unflanged Circular Pipe," *Phys. Rev.,* vol. 73, pp. 383–406, 1948.

4

Aeroacoustics Interfaces

This chapter describes the physics interfaces found under the **Aeroacoustics** branch .

The aeroacoustic branch has physics interfaces to solve the fully linearized acoustic equation for several physical conditions, they are: the *linearized potential flow* (LPF) equations, the *linearized Euler* (LE) equations, and the *linearized Navier-Stokes* (LNS) equations.

In this chapter:

- Theory Background for the Aeroacoustics Branch
- References for the Aeroacoustics Branch Interfaces

The Linearized Potential Flow, Frequency Domain Interface

The **Linearized Potential Flow, Frequency Domain (ae)** interface ( ), found under the **Acoustics>Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in the velocity potential in the presence of an inviscid and irrotational background mean-flow, that is, a potential flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized compressible potential flow equations.

The equations are formulated in the frequency domain and assume harmonic variation of all sources and fields. The physics interface is limited to flows with a Mach number Ma < 1, partly due to limitations in potential flow and partly due to the acoustic boundary settings needed for supersonic flow. The coupling between the acoustic field and the background flow does not include any predefined flow-induced noise.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Aeroacoustics Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**. For axisymmetric components an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Potential Flow, Frequency Domain** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `ae`.

LINEARIZED POTENTIAL FLOW EQUATION SETTINGS

For 1D axisymmetric components, the **Circumferential wave number** m (dimensionless) is 0 by default. The **Out-of-plane wave number** k_z (SI unit: rad/m) is 0 rad/m by default.

For 2D components, the **Out-of-plane wave number** k_z (SI unit: rad/m) is 0 rad/m by default.

For 2D axisymmetric components the **Circumferential wave number** m (dimensionless) is 0 by default.

SOUND PRESSURE LEVEL SETTINGS

The settings are the same as Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface.

TYPICAL WAVE SPEED

The settings are the same as Typical Wave Speed for the Pressure Acoustics, Frequency Domain interface.

DEPENDENT VARIABLES

This physics interface defines one dependent variable (field), the **Velocity potential** `phi`. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. From the list select the element order and type (Lagrange or serendipity) for the velocity potential, the default is **Quadratic Lagrange**.

Choosing between Lagrange and Serendipity Shape Functions has influence on the number of DOFs solved for and on stability for distorted mesh.

	• Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Frequency Domain Interface • Theory Background for the Aeroacoustics Branch

	• *Flow Duct*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct** • *Doppler Shift*: Application Library path **Acoustics_Module/Tutorials/doppler_shift**

Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Frequency Domain Interface

The Linearized Potential Flow, Frequency Domain Interface has these domain, boundary, edge, point and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Continuity
- Initial Values
- Impedance, Interior Impedance, and Pair Impedance
- Interior Sound Hard Boundary (Wall)
- Linearized Potential Flow Model
- Mass Flow Circular Source
- Mass Flow Edge Source
- Mass Flow Point Source
- Normal Mass Flow
- Normal Velocity
- Periodic Condition
- Plane Wave Radiation
- Sound Hard Boundary (Wall)
- Sound Soft Boundary
- Velocity Potential
- Vortex Sheet

For axisymmetric components, COMSOL Multiphysics takes the axial symmetry (at $r = 0$) into account and automatically adds an **Axial Symmetry** node to the component that is valid on the axial symmetry edges/points only.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Linearized Potential Flow Model

The **Linearized Potential Flow Model** node adds the equations for frequency domain aeroacoustics modeling. You here need to enter the material properties as well as the background mean flow information.

LINEARIZED POTENTIAL FLOW MODEL

The default values for the **Density** ρ_0 (SI unit: kg/m^3) and the **Mean flow speed of sound** c_0 (SI unit: m/s) are taken **From material**. For **User defined** enter other values or expressions. For example, this could be to select the values taken from a simulation run using The Compressible Potential Flow Interface.

Also enter values or expressions for the **Mean flow velocity** $\mathbf{V}$ (SI unit: m/s).

It is important to know that the velocity field needs to be a solution to a compressible potential flow simulation. It has to be an irrotational and inviscid flow, for example, a constant flow field $\mathbf{V}$ is of this type. Any other type of flow yields non-physical solutions for this formulation of the governing equations.

Coupling between the flow form The Compressible Potential Flow Interface interface to the linearized potential flow model can be set up using the Background Potential Flow coupling found under the Multiphysics node. The feature appears when both physics interfaces are present in a model.

Initial Values

The **Initial Values** node adds initial values for the velocity potential. For The Linearized Potential Flow, Transient Interface it also adds initial values for the velocity potential, first time derivative. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial value of the **Velocity potential** *phi* (SI unit: m^2/s).

For The Linearized Potential Flow, Transient Interface also enter a **Velocity potential, first time derivative**, $\partial\phi/\partial t$ (SI unit: m^2/s^2).

Sound Hard Boundary (Wall)

Use the **Sound Hard Boundary (Wall)** condition to model rigid boundary surfaces or walls. It prescribes a vanishing normal component of the particle velocity at the boundary. Multiplied by the density, it can equivalently be expressed as a *no-flow* condition:

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{V}\cdot\nabla\phi\right)\right)=0$$

The sound-hard boundary condition is available for all analysis types. The equation above applies to the time domain calculations in The Linearized Potential Flow, Transient Interface; to obtain the corresponding condition for frequency domain, simply replace $\partial/\partial t$ by $i\omega$.

The Linearized Potential Flow, Boundary Mode Interface the no-flow or wall condition, known as *sound hard*, sets the normal acceleration — and thus also the normal velocity — to zero at the edge.

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi+(\nabla\phi\cdot\mathbf{V}))\right)=0$$

Velocity Potential

Use the **Velocity Potential** node when coupling two Linearized Potential Flow, Frequency Domain interfaces together because it can sometimes be necessary to set the velocity potential: $\phi=\phi_0$.

VELOCITY POTENTIAL

Enter a **Velocity potential** ϕ_0 (SI unit: m^2/s).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Normal Mass Flow

Use the **Normal Mass Flow** node to set the inward mass flow boundary condition.

For The Linearized Potential Flow, Frequency Domain Interface, the natural boundary condition for the total wave has the meaning of a mass flow through the boundary surface:

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{V}\cdot\nabla\phi\right)\right)=m_n$$

For The Linearized Potential Flow, Boundary Mode Interface, the natural edge condition for the total wave has the meaning of normal mass flow.

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi+(\nabla\phi\cdot\mathbf{V})+\lambda\phi V_n)\right)=m_n$$

NORMAL MASS FLOW

Enter an **Inward mass flow** m_n (SI unit: $kg/(m^2\cdot s)$).

Plane Wave Radiation

The **Plane Wave Radiation** is a class of non-reflecting boundary conditions, which assume that there is an outgoing plane wave, and optionally also an incoming exciting wave.

For transient analysis the boundary condition is

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}+(\nabla\phi\cdot\mathbf{V})\right)\right)-\rho_0 k_n\frac{\partial\phi}{\partial t}-\mathbf{n}\cdot\mathbf{V}\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}-k_n\frac{\partial\phi}{\partial t}\mathbf{n}\cdot\mathbf{V}\right)=$$

$$\rho_0\frac{\partial}{\partial t}\phi_0 k_k(\mathbf{n}\cdot\mathbf{n}_\mathbf{k})-\mathbf{n}\cdot\mathbf{V}\frac{\rho_0}{c_0^2}k_k\frac{\partial\phi_0}{\partial t}\mathbf{n}_\mathbf{k}\cdot\mathbf{V}-\rho_0 k_n\frac{\partial\phi_0}{\partial t}+\mathbf{n}\cdot\mathbf{V}\frac{\rho_0}{c_0^2}\left(k_n\frac{\partial\phi_0}{\partial t}\mathbf{n}\cdot\mathbf{V}\right)$$

$$k_k=\frac{1}{c_0+\mathbf{V}\cdot\mathbf{n}_\mathbf{k}}\qquad k_n=\frac{1}{c_0+\mathbf{V}\cdot\mathbf{n}}$$

while the corresponding time-harmonic equation reads

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi+\nabla\phi\cdot\mathbf{V})\right)-\rho_0 ik_n\phi-\mathbf{n}\cdot\mathbf{V}\frac{\rho_0}{c_0^2}(i\omega-ik_n\mathbf{n}\cdot\mathbf{V})\phi=$$

$$\rho_0\left(ik_k\mathbf{n}\cdot\mathbf{n}_\mathbf{k}-\mathbf{n}\cdot\frac{\mathbf{V}}{c_0^2}(ik_k\mathbf{n}_\mathbf{k}\cdot\mathbf{V})-ik_n+\mathbf{n}\cdot\frac{\mathbf{V}}{c_0^2}(ik_n\mathbf{n}\cdot\mathbf{V})\right)\phi_0 e^{-i\mathbf{k}\cdot\mathbf{r}}$$

$$k_k=\frac{\omega}{c_0+\mathbf{V}\cdot\mathbf{n}_\mathbf{k}}\qquad k_n=\frac{\omega}{c_0+\mathbf{V}\cdot\mathbf{n}}\qquad \mathbf{k}=k\,\mathbf{n}_\mathrm{k}\qquad \mathbf{n}_\mathrm{k}=\frac{\mathbf{e}_\mathrm{k}}{|\mathbf{e}_\mathrm{k}|}$$

Specify an Incident Velocity Potential (incoming plane wave)

$$\phi_0 e^{-i\mathbf{k}\cdot\mathbf{r}}$$

by supplying its amplitude, ϕ_0, and propagation wave direction vector, $\mathbf{e}_\mathrm{k}$. The vector $\mathbf{n}_\mathrm{k}$ is the normalized wave direction vector of unit length.

This boundary condition is most relevant for ports, because many waveguide structures are only interesting in the plane-wave region.

Incident Velocity Potential

The **Incident Velocity Potential** subnode is available from the context menu (right-click the Plane Wave Radiation parent node) or from the **Physics** toolbar, **Attributes** menu. Use this to add a velocity potential and wave direction.

INCIDENT VELOCITY POTENTIAL

Enter a **Velocity potential** ϕ_0 (SI unit: m^2/s) and **Wave direction** $\mathbf{e}_k$ (SI unit: m). The default for the wave direction is the inward normal direction of the boundary.

Sound Soft Boundary

The **Sound Soft Boundary** creates a boundary condition for a sound soft boundary, where the acoustic pressure vanishes and $p = 0$.

This boundary condition is an appropriate approximation for a liquid-gas interface and in some cases for external waveguide ports.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and physics interfaces being modeled. This feature works well for cases like opposing parallel boundaries. In other cases, use a **Destination Selection** subnode to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

PERIODICITY SETTINGS

Select a **Type of periodicity**: **Continuity** (the default) or **Antiperiodicity**.

ORIENTATION OF SOURCE

For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Normal Velocity

Use the **Normal Velocity** node in time-harmonic analysis to specify the velocity component normal to the boundary:

$$\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi+\mathbf{V}\cdot\nabla\phi)\right)=\rho_0\left(v_n+\frac{1}{i\omega}\mathbf{V}\cdot\nabla v_n\right)$$

Here v_n denotes the outward normal velocity at the boundary surface, which is specified in the v_n text field.

NORMAL VELOCITY

Enter a **Normal velocity** v_n (SI unit: m/s).

Impedance, Interior Impedance, and Pair Impedance

Use the **Impedance**, **Interior Impedance,** or **Pair Impedance** node in time-harmonic analysis to define the input impedance of an external domain, at an interior boundary, or at the boundary between parts in an assembly as the ratio of pressure (or pressure drop) to normal velocity, $Z_i = p/(\mathbf{n}\cdot\mathbf{v})$ at the boundary. The associated impedance boundary condition is

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\mathbf{V}\right) = \rho_0\left(\frac{p}{Z_i} + \frac{1}{i\omega}(\mathbf{V}\cdot\nabla)\frac{p}{Z_i}\right) \qquad i = \text{up, down}$$

On pair (interior) impedance conditions the up/down, that refers to the up and down side of an interior boundary, is replaced by a 1 and 2 index in the equation display. These refer to the two faces in the pair.

IMPEDANCE/INTERIOR IMPEDANCE/PAIR IMPEDANCE

Enter an input **Impedance** Z_i (SI unit: Pa·s/m).

Vortex Sheet

Use the **Vortex Sheet** boundary condition to model a shear layer that separates a stream from the free velocity field. Because the velocity potential is discontinuous over this boundary, use a slit boundary condition or a pair in an assembly. Vortex sheets are only applicable on interior boundaries.

The equations defining the vortex sheet boundary condition are

$$\left[\mathbf{n}\cdot\left(\rho_0\nabla\phi - \mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\right)\right]_i = [\rho_0(i\omega + \mathbf{V}\cdot\nabla)w]_i \qquad i = \text{up, down}$$

$$p_{\text{up}} = p_{\text{down}} \qquad w_{\text{up}} = -w_{\text{down}}$$

where w denotes the outward normal displacement (SI unit: m) of the boundary surface, which this boundary condition adds as `ae.vs1.w`, using the default name for the physics interface. On pair (interior) vortex sheet conditions the up/down, that refers to the up and down side of an interior boundary, is replaced by a 1 and 2 index in the equation display. These refer to the two faces in the pair.

Interior Sound Hard Boundary (Wall)

For The Linearized Potential Flow, Frequency Domain Interface and The Linearized Potential Flow, Transient Interface, use the **Interior Sound Hard Boundary (Wall)** condition to model interior rigid boundary surfaces, or walls. It prescribes a vanishing normal component of the particle velocity at the boundary. Multiplied by the density, it can equivalently be expressed as a *no-flow* condition:

$$\left[-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{V}\cdot\nabla\phi\right)\right)\right]_i = 0 \qquad i = \text{up, down}$$

The up/down refers to the up and down side of an interior boundary. The sound-hard boundary condition is available for all analysis types. The equation above applies to the time domain calculations in the Linearized Potential Flow, Transient interface; to obtain the corresponding condition for the frequency domain, simply replace $\partial/\partial t$ with $i\omega$.

Continuity

Continuity is available as an option at interfaces between parts in a pair. This condition gives continuity in the velocity potential as well as continuity in the mass flow. It corresponds to a situation where the boundary has no direct effect on the acoustic velocity potential field (subscripts 1 and 2 in the equation refers to the two sides of the pair):

$$\mathbf{n}\cdot\left[\left(\rho_0\left(\nabla\phi-\frac{\mathbf{V}}{c_0^2}(i\omega+\mathbf{V}\cdot\nabla)\phi\right)\right)_1-\left(\rho_0\left(\nabla\phi-\frac{\mathbf{V}}{c_0^2}(i\omega+\mathbf{V}\cdot\nabla)\phi\right)\right)_2\right] = 0$$

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Mass Flow Edge Source

For 3D components, use a **Mass Flow Edge Source** to specify the mass flow rate on an edge:

$$-\frac{\rho_0}{c_0^2}i\omega(i\omega\phi+\mathbf{V}\cdot\nabla\phi)+\nabla\cdot\left(\rho_0\nabla\phi-\mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi+\mathbf{V}\cdot\nabla\phi)\right) = m'e^{i\theta}\delta(\mathbf{x}-\mathbf{x}_0)\mathrm{d}l$$

In the time domain, factors $i\omega$ are replaced by partial time derivatives, $\frac{\partial}{\partial t}$.

MASS FLOW EDGE SOURCE

Enter a **Mass flow rate** m' (SI unit: kg/(m^2·s)) and enter a **Phase** θ (dimensionless).

Mass Flow Point Source

Add a **Mass Flow Point Source** node to specify the mass flow rate on a point:

$$-\frac{\rho_0}{c_0^2}i\omega(i\omega\phi + \mathbf{V}\cdot\nabla\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\right) = m'e^{i\theta}\delta(\mathbf{x}-\mathbf{x}_0)$$

In the time domain, factors $i\omega$ are replaced by partial time derivatives, $\frac{\partial}{\partial t}$.

MASS FLOW POINT SOURCE

Enter a **Mass flow rate** m' (SI unit: kg/s for 3D and 2D axisymmetric components; kg/(m·s) for 2D components).

Enter a **Phase** θ (dimensionless).

Doppler Shift: Application Library path
Acoustics_Module/Tutorials/doppler_shift

Mass Flow Circular Source

For 2D axisymmetric components, use a **Mass Flow Circular Source** node to add a circular source located at $\mathbf{x} = \mathbf{x}_0$:

$$-\frac{\rho_0}{c_0^2}i\omega(i\omega\phi + \mathbf{V}\cdot\nabla\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\right) = m'e^{i\theta}\delta(\mathbf{x}-\mathbf{x}_0)r\,\mathrm{d}\varphi$$

In the time domain, factors $i\omega$ are replaced by partial time derivatives, $\frac{\partial}{\partial t}$.

MASS FLOW CIRCULAR SOURCE

Enter a **Mass flow rate** m' (SI unit: kg/(m·s) and enter a **Phase** θ (dimensionless).

Mass Flow Line Source on Axis

For 2D axisymmetric components, use a **Mass Flow Line Source on Axis** node to add a line source along the symmetry axis:

$$-\frac{\rho_0}{c_0^2} i\omega(i\omega\phi + \mathbf{V}\cdot\nabla\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\right) = m' e^{i\theta}\delta(\mathbf{x}-\mathbf{x}_0)\mathrm{d}z$$

In the time domain, factors $i\omega$ are replaced by partial time derivatives, $\frac{\partial}{\partial t}$.

MASS FLOW LINE SOURCE ON AXIS

Enter a **Mass flow rate** m' (SI unit: kg/(m·s)) and enter a **Phase** θ (dimensionless).

Axial Symmetry

The **Axial symmetry** feature is a default node added for all axisymmetric components. The boundary condition is active on all boundaries on the symmetry axis.

The Linearized Potential Flow, Transient Interface

The **Linearized Potential Flow, Transient (aetd)** interface ( ), found under the **Acoustics>Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in the velocity potential in the presence of an inviscid and irrotational background mean-flow, that is, a potential flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized compressible potential flow equations.

The equations are formulated in the time domain. The physics interface is limited to flows with a Mach number Ma < 1, partly due to limitations in potential flow and partly due to the acoustic boundary settings needed for supersonic flow. The coupling between the acoustic field and the background flow does not include any predefined flow-induced noise.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Potential Flow Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**. For axisymmetric components an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Potential Flow, Transient** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `aetd`.

TRANSIENT SOLVER SETTINGS

Select the **Time stepping** (method) — **Manual** (default and recommended) or **Automatic/free** and then enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency

content of the sources used in the model. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

Details about Time Stepping in Transient Models are found in the Modeling with the Aeroacoustics Branch section.

The remainder of the settings are shared with The Linearized Potential Flow, Frequency Domain Interface.

- Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Transient Interface
- Theory Background for the Aeroacoustics Branch

Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Transient Interface

The Linearized Potential Flow, Transient Interface has these domain, boundary, edge, point, and pair nodes, listed in alphabetical order and described for The Linearized Potential Flow, Frequency Domain Interface. The nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Continuity
- Initial Values
- Interior Sound Hard Boundary (Wall)
- Linearized Potential Flow Model
- Mass Flow Point Source
- Mass Flow Circular Source
- Mass Flow Edge Source
- Normal Mass Flow
- Periodic Condition
- Plane Wave Radiation
- Sound Hard Boundary (Wall)
- Velocity Potential

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

The Linearized Potential Flow, Boundary Mode Interface

The **Linearized Potential Flow, Boundary Mode (aebm)** interface ( ), found under the **Acoustics>Aeroacoustics** branch ( ) when adding a physics interface, is used to compute eigenmodes and out-of-plane wave numbers for the linearized compressible potential flow equations. This study is used, for example, when specifying sources at inlets or analyzing transverse acoustic modes in ducts.

The physics interface solves an eigenvalue equation on boundaries, searching for the out-of-plane wave numbers at a given frequency.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Potential Flow Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**. For 2D axisymmetric models an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Potential Flow, Boundary Mode** to select physics features from the context menu.

This physics interface is limited to flows with a Mach number $\mathrm{Ma} < 1$, partly due to limitations in the potential flow formulation and partly due to the acoustic boundary settings needed for supersonic flow.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `aebm`.

LINEARIZED POTENTIAL FLOW EQUATION SETTINGS

For 2D axisymmetric components, the **Circumferential mode number** m (dimensionless) is 0 by default. It is an integer entering the axisymmetric expression for the velocity potential:

$$\phi(r, z, \varphi) = \phi(r)\, e^{-i(k_z z + m\varphi)}$$

SOUND PRESSURE LEVEL SETTINGS

The settings are the same as Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface.

DEPENDENT VARIABLES

This physics interface defines one dependent variable (field), the **Velocity potential** *phi*. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. From the list select the element order and type (Lagrange or serendipity) the default is **Quadratic Lagrange**.

Choosing between Lagrange and Serendipity Shape Functions has influence on the number of DOFs solved for and on stability for distorted mesh.

- Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Boundary Mode Interface
- Theory Background for the Aeroacoustics Branch

Flow Duct: Application Library path
Acoustics_Module/Aeroacoustics_and_Noise/flow_duct

Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Boundary Mode Interface

The Linearized Potential Flow, Boundary Mode Interface has these boundary, edge, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

These features are described for The Linearized Potential Flow, Frequency Domain Interface. The only difference for the Linearized Potential Flow, Boundary Mode interface is that you apply the features to boundaries instead of domains for 3D components. In the Linearized Potential Flow Model you can enter a **Unit normal** that defines the positive propagation direction.

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using.

- Continuity
- Initial Values
- Linearized Potential Flow Model
- Normal Mass Flow
- Sound Hard Boundary (Wall)
- Sound Soft Boundary
- Velocity Potential

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

The Compressible Potential Flow Interface

The **Compressible Potential Flow (cpf)** interface ( ), found under the **Acoustics>Aeroacoustics** branch ( ), is used to compute the velocity potential and density in a compressible potential flow model. Derived values include the associated pressure, velocity, and temperature in the flow. In a compressible potential flow model the flow is assumed to be represented by an ideal barotropic, irrotational fluid at constant entropy, that is, the fluid is also inviscid. The physics interface is used for modeling the background mean flow used as input to The Linearized Potential Flow, Frequency Domain Interface or The Linearized Potential Flow, Transient Interface.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Compressible Potential Flow**, **Slip Velocity**, and **Initial Values**. For axisymmetric models an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Compressible Potential Flow** to select physics features from the context menu.

The potential flow formulation for steady compressible flow is in general not suited for modeling shocks. In the region after a shock the flow is typically rotational, hence it is only suited for problems with a Mach number $\mathrm{Ma} < 1$.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `cpf`.

REFERENCE VALUES

Edit or enter the values as needed:

- **Reference pressure** p_{ref} (SI unit: Pa). The default is 1 atm.
- **Reference density** ρ_{ref} (SI unit: kg/m^3). The default is 1.2 kg/m^3.
- **Reference velocity** v_{ref} (SI unit: m/s).
- **Reference force potential** Ψ_{ref} (SI unit: J/kg).

DEPENDENT VARIABLES

This physics interface defines two dependent variables (field), the **Mean flow velocity potential** `Phi` and the **Density** `rho`. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**.

- Domain, Boundary, and Pair Nodes for the Compressible Potential Flow Interface
- Theory Background for the Aeroacoustics Branch

Flow Duct: Application Library path
Acoustics_Module/Aeroacoustics_and_Noise/flow_duct

Domain, Boundary, and Pair Nodes for the Compressible Potential Flow Interface

The Compressible Potential Flow Interface has these domain, boundary, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows

users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

- Compressible Potential Flow Model
- Initial Values
- Interior Wall (Slip Velocity)
- Mass Flow
- Mean Flow Velocity Potential
- Normal Flow
- Periodic Condition
- Slip Velocity
- Symmetry

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Compressible Potential Flow Model

The **Compressible Potential Flow Model** node adds equations for time-dependent or stationary modeling of compressible potential flow. The model will solve for the mean flow velocity potential and the density. The pressure and the velocity field are defined from these dependent variables.

COMPRESSIBLE POTENTIAL FLOW MODEL

For the **Ratio of specific heats** γ (dimensionless) select **From material** (the default) or **User defined**. For the **User defined** option the default value is 1.4.

Enter a **Force potential** ψ (SI unit: J/kg). To model a domain force or domain source acting on the fluid.

Select the **Calculate temperature** option it you need to also calculate the temperature field in the fluid. If this is selected, you also need to specify the **Specific gas constant** R_s (SI unit: J/(kg·K)). Select **From material** (the default) or **User defined.** Enabling this calculates the temperature according to

$$T = \frac{p_A}{\rho R_s}$$

where p_A is the pressure and ρ the density in the fluid.

Depending on the material selected in the **Materials** node for the fluid (typically Air) the **Model Inputs** may be active. Per default the **Absolute pressure** is taken from the model itself (grayed out). If the **Calculate temperature** option is enabled the **Temperature** is also taken from the model (grayed out). This will give a physically consistent solution. In both cases, you can click the **Make Model Inputs Editable** icon to enter a user defined expression.

Initial Values

The **Initial Values** node adds initial values for the mean flow velocity potential and density variables. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial values **Mean flow velocity potential** `Phi` (SI unit: m^2/s). Enter a **Density** `rho` (SI unit: kg/m^3) the default is the reference density `cpf.rhoref`.

Slip Velocity

The **Slip Velocity** node is the natural condition at a boundary impervious to the flow, meaning that the velocity normal to the boundary is zero. By multiplying with the density, this condition can alternatively be expressed as a vanishing mass flow through the boundary

$$-\mathbf{n} \cdot \rho \nabla \Phi = 0$$

Symmetry

The **Symmetry** condition is applied at planes where the model has symmetries. Using the intrinsic symmetries of a problem is a simple way to reduce the model size.

Normal Flow

The **Normal Flow** node implies that the flow is normal to the boundary and thus that the tangential velocity is zero. This corresponds to a constant velocity potential along

the boundary. Because the velocity potential is determined only up to a constant, imposing this condition fixes the arbitrary constant to zero where $\phi = 0$.

	Setting the Normal Flow condition on two or more disjoint boundaries can result in the wrong features unless symmetry implies that the velocity potential is equal on the boundaries in question.

Mass Flow

The **Mass Flow** node specifies the mass flow through the boundary. The mass flow is given by the product of the normal velocity v_n and the density at the boundary ρ_{bnd}. This results in a flux condition given by

$$-\mathbf{n} \cdot \rho \nabla \Phi = v_n \rho_{bnd}$$

MASS FLOW

Enter the **Normal velocity** v_n (SI unit: m/s) and **Fluid density at the boundary** ρ_{bnd} (SI unit: kg/m^3). The defaults are `cpf.vref` and `cpf.rhoref`, respectively. The values given in the Reference Values.

Mean Flow Velocity Potential

Use the **Mean Flow Velocity Potential** condition to prescribe the flow potential Φ at a boundary, $\Phi = \Phi_0$. This condition can be used to set up theoretical flow conditions at a boundary or couple to other physics interfaces.

MEAN FLOW VELOCITY POTENTIAL

Enter the Mean flow velocity potential Φ_0 (SI unit: m^2/s).

Periodic Condition

Use the **Periodic Condition** to define periodicities between boundaries in the model. From the **Type of periodicity** menu select **Continuity** (default) or **Antiperiodicity**.

Interior Wall (Slip Velocity)

The **Interior Wall (Slip Velocity)** is used to set up interior walls on interior boundaries. This simplifies modeling and meshing steps as the wall does not need to have a

thickness. The condition is simply applied to an interior boundary in the model. The condition enforces a vanishing mass flow on the up and down sides of the wall.

$$(-\mathbf{n} \cdot \rho \nabla \Phi)_{\text{up}} = 0 \qquad (-\mathbf{n} \cdot \rho \nabla \Phi)_{\text{down}} = 0$$

The dependent variables are defined as being discontinuous (so called slits) at this boundary.

The Linearized Euler, Frequency Domain Interface

The **Linearized Euler, Frequency Domain (lef)** interface ( ), found under the **Acoustics>Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in density, velocity, and pressure in the presence of a stationary background mean-flow that is well approximated by an ideal gas flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized Euler equations.

The equations defined by the Linearized Euler, Frequency Domain interface are the linearized continuity, momentum (Euler), and energy equations. The physics interface solves for the acoustic variations in the density ρ, velocity field $\mathbf{u}$, and pressure p. The equations are formulated in the frequency domain and assume harmonic variation of all sources and fields. The background mean flow can be any stationary gas flow that is well approximated by an ideal gas. The coupling between the acoustic field and the background flow does not include any predefined flow induced noise. Even though the equations do not include any loss mechanisms, only acoustic modes exist in the frequency domain as the driving frequency is predefined and real valued.

The equations are implemented in the so-called scattered field formulation. All equations and boundary conditions are formulated in the total acoustic fields (ρ_t, $\mathbf{u}_t$, p_t). The total fields are in the presence of the Background Acoustic Fields feature the sum of the background (ρ_b, $\mathbf{u}_b$, p_b) and the scattered field (ρ, $\mathbf{u}$, p):

$$\rho_t = \rho + \rho_b \qquad \mathbf{u}_t = \mathbf{u} + \mathbf{u}_b \qquad p_t = p + p_b$$

The scattered field variables are the variables solved for, that is, the dependent variables. When no background acoustic field is present the total field is simply equal to the scattered field

$$\rho_t = \rho \qquad \mathbf{u}_t = \mathbf{u} \qquad p_t = p$$

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Euler Model**, **Rigid Wall**, and **Initial Values**. For axisymmetric components an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Euler, Frequency Domain** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `lef`.

SOUND PRESSURE LEVEL SETTINGS

See Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface. Only **Use reference pressure for air** or **User-defined reference pressure** are available selections.

TYPICAL WAVE SPEED

See Typical Wave Speed for the Pressure Acoustics, Frequency Domain interface.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Density** `rho`, **Velocity field** `u` and its components, and **Pressure** `p`. The name can be changed but the names of fields and dependent variables must be unique within a model.

STABILIZATION

To display this section, click the **Show** button ( ) and select **Stabilization**. The **Streamline diffusion** check box is selected by default and the **Stabilization parameter** α_{SD} (dimensionless) default is $5 \cdot 10^{-3}$. Click to clear the check box as required.

The stabilization scheme implements the streamline upwind Petrov-Galerkin (SUPG) formulation of the weak form equations used for the finite element method. The stabilization parameter α_{SD} can be tuned depending on the problem solved, the nature of the background mean flow, and on the computational mesh. The implementation follows the one discussed in Ref. 10 and Ref. 14.

If the stabilization is turned off it is probably necessary to change the discretization to ensure a stable numerical scheme. Set the order of the density one lower than the

velocity and pressure dependent variables, for example, using a P1-P2-P2 discretization.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. From the list select the element order and type (Lagrange or serendipity) the default is **Linear** for all the dependent variables.

Choosing between Lagrange and Serendipity Shape Functions has influence on the number of DOFs solved for and on stability for distorted mesh.

- Domain, Boundary, and Pair Nodes for the Linearized Euler, Frequency Domain Interface
- Theory Background for the Aeroacoustics Branch

For modeling tips and tricks and good practice see the Modeling with the Aeroacoustics Branch section.

Domain, Boundary, and Pair Nodes for the Linearized Euler, Frequency Domain Interface

The Linearized Euler, Frequency Domain Interface has these domain, boundary, and pair nodes and subnodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Asymptotic Far-Field Radiation
- Axial Symmetry
- Background Acoustic Fields
- Continuity
- Domain Sources
- Impedance and Interior Impedance
- Initial Values
- Interior Wall
- Linearized Euler Model
- Moving Wall
- Outflow Boundary
- Prescribed Acoustic Fields
- Periodic Condition[1]
- Pressure (Isentropic)
- Rigid Wall
- Symmetry

[1]Described for the Pressure Acoustics, Frequency Domain interface

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Linearized Euler Model

Use the **Linearized Euler Model** to set up the governing equations, define the background mean flow, the ideal gas fluid properties, and select gradient term suppression stabilization, if needed. The governing equations solved are (in the time domain):

$$\begin{aligned}
&\frac{\partial \rho_t}{\partial t} + \nabla \cdot (\rho_t \mathbf{u}_0 + \rho_0 \mathbf{u}_t) = S_c \\
&\frac{\partial \mathbf{u}_t}{\partial t} + \nabla \cdot \left(\mathbf{u}_t \mathbf{u}_0^{\mathrm{T}} + \frac{p_t}{\rho_0} \mathbf{I} \right) + \frac{\rho_t}{\rho_0} (\mathbf{u}_0 \cdot \nabla) \mathbf{u}_0 - p_t \nabla (\rho_0^{-1}) \\
&\qquad + (\nabla \mathbf{u}_0 - (\nabla \cdot \mathbf{u}_0) \mathbf{I}) \mathbf{u}_t = S_m \\
&\frac{\partial p_t}{\partial t} + \nabla \cdot (\gamma p_0 \mathbf{u}_t + p_t \mathbf{u}_0) + (1-\gamma)(\mathbf{u}_t \cdot \nabla) p_0 - (1-\gamma)(\nabla \cdot \mathbf{u}_0) p_t = S_e
\end{aligned} \tag{4-1}$$

where ρ_t, $\mathbf{u}_t$, and p_t are the acoustic perturbations to the density, velocity, and pressure, respectively. The subscript "t" refers to the fact that the acoustic variables are the total fields, that is, the sum of possible Background Acoustic Fields and the scattered fields.

In the frequency domain the time derivatives of the dependent variables is replaced by multiplication with $i\omega$. The variables with a zero subscript are the background mean flow values, γ is the ratio of specific heats. The right-hand-side source terms S_c, S_m, and S_e are zero. They can be defined in the Domain Sources node. Details about the physics interfaces and references are found in the Theory Background for the Aeroacoustics Branch section.

MODEL INPUTS

In order to model the influence of the background mean flow on the propagation of the acoustic waves in the fluid the background mean flow temperature T_0, absolute pressure p_0, and velocity field $\mathbf{u}_0$ need to be defined.

All the background flow parameters can be functions of space. They can be either analytical expressions (user defined) or they can be picked up from a flow simulation performed using the CFD Module. By default they are set to the quiescent background conditions of air.

Enter **User defined** values for the **Background mean flow temperature** T_0 (SI unit: K), **Background mean flow pressure** p_0 (SI unit: Pa), and **Background mean flow velocity** $\mathbf{u}_0$ (SI unit: m/s). The defaults are 293.15 K, 1 atm, and 0 m/s, respectively.

Note that the **Background mean flow density** also needs to be defined or entered in the **Fluid Properties** section below.

When modeling aeroacoustics it is important how the Mapping Between CFD and Acoustics Mesh is done from a numerical perspective.

FLUID PROPERTIES

Select an option for the **Background mean flow density** ρ_0 (SI unit: kg/m^3) — **Ideal gas** (the default), **From material**, **User defined** (default value 1.2 kg/m^3), or it can be picked up from a flow interface, for example, from a High Mach Number Flow model as **Density (hmnf/fluid1)**. As the flow is assumed to be an ideal gas, the background density ρ_0 is readily defined as

$$\rho_0 = \frac{p_0}{R_s T_0}$$

where R_s is the specific gas constant, it can also be defined as a material input and is then defined as $\rho_0 = \rho_0(p_0, T_0)$, or it can be picked up from a flow model.

Define the remaining fluid properties necessary for defining an ideal gas. Select the **Gas constant type**: **Specific gas constant** (the default) or **Mean molar mass**. The defaults take values **From material** or for **User defined** enter another value or expression:

- **Specific gas constant** R_s (SI unit: J/(kg·K)). The default is 287.058 J/(kg·K)).
- **Mean molar mass** M_n (SI unit: g/mol). The default is 28.97 g/mol), which calculates $R_s = R/M_n$, where R is the gas constant.

Select an option from the **Specify Cp or γ** list: **Ratio of specific heats** (the default) or **Heat capacity at constant pressure**. The defaults take values **From material** or for **User defined** enter another value or expression:

- **Ratio of specific heats** γ (dimensionless). The default is 1.4.
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)). The default is 1005.4 J/(kg·K)), which calculates $\gamma = C_p/(C_p - R_s)$.

GRADIENT TERM SUPPRESSION STABILIZATION

When the linearized Euler (LE) equations are solved in the time domain (or in the frequency domain with an iterative solver), linear physical instability waves can develop, the so-called Kelvin-Helmholtz instabilities. They are instabilities that grow exponentially because no losses exist in the LE equations (no viscous dissipation and no heat conduction). Furthermore, they are limited by non-linearities in the full Navier-Stokes flow equations. It has been shown that the growth of these instabilities can be limited, while the acoustic solution is retained, by canceling terms involving gradients of the mean flow quantities. This is known as gradient terms suppression (GTS) stabilization.

More details in Ref. 9, Ref. 10, Ref. 11, and the Theory Background for the Aeroacoustics Branch section.

Select the following check boxes to activate the applicable gradient term suppression (GTS), which is a form of physical stabilization where certain terms involving gradients of the background mean flow properties are removed:

- **Suppression of mean flow density gradients**

  This option sets the following terms in the governing equations to zero:

$$p_t \nabla(\rho_0^{-1}) = 0$$

- **Suppression of mean flow velocity gradients**

  This option sets the following terms in the governing equations to zero:

$$\frac{\rho_t}{\rho_0}(\mathbf{u}_0 \cdot \nabla)\mathbf{u}_0 + (\nabla \mathbf{u}_0 - (\nabla \cdot \mathbf{u}_0)\mathbf{I})\mathbf{u}_t = 0$$

$$(1-\gamma)(\nabla \cdot \mathbf{u}_0)p_t = 0$$

- **Suppression of mean flow pressure gradients**

  This option sets the following terms in the governing equations to zero:

$$(1-\gamma)(\mathbf{u}_t \cdot \nabla)p_0 = 0$$

Rigid Wall

The **Rigid Wall** condition is used to model a rigid wall, corresponding to the sound hard wall condition in Pressure Acoustics. In the case of lossless flows, this reduces to the slip condition $\mathbf{u}_t \cdot \mathbf{n} = 0$ where $\mathbf{n}$ is the surface normal.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Initial Values

The **Initial Values** node adds initial values for the density, the velocity field, and pressure. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial values of the **Density** `rho` (SI unit: kg/m^3), the **Velocity field** `u` (SI unit: m/s), and the **Pressure** `p` (SI unit: Pa).

Axial Symmetry

In a 2D axisymmetric geometry the **Axial Symmetry** condition is automatically added on the axis of symmetry at $r = 0$. On this boundary the radial component of the total velocity is set equal to zero

$$u_t = 0 \qquad \mathbf{u}_t = (u_t, v_t, w_t)$$

Domain Sources

Add a **Domain Sources** node to define the mass source, and momentum and energy source types. This domain feature adds the right-hand side to the governing

Equation 4-1. This condition can be used to create any user defined source by entering expressions into the field. The sources can for example be point-like Gaussian sources to model a single vortex or two interacting vortices. Note that specifying the source term to the continuity equation often also requires specifying the source term to the energy equation. For example, for an isotropic pressure and density are related by $p = c_0^2\rho$, where c_0 is the local speed of sound.

DOMAIN SOURCES

Enter a **Mass source** S_c (SI unit: kg/(m^3·s)).

Select a **Momentum source type**: **Acceleration** (the default) or **Volume force**. For **Acceleration** enter vector expressions or values for the **Acceleration source** S_m (SI unit: m/s^2). For **Volume force** enter vector expressions or values for the **Volume force source** $\mathbf{F}_m$ (SI unit: N/m^3).

Select an **Energy source type**: **Pressure rate of change** (the default) or **Heat source**. For **Pressure rate of change** enter an expression or value for the **Pressure rate of change source** S_e (SI unit: Pa/s). For **Heat source** enter vector expressions or values for the **Heat source** Q_e (SI unit: W/m^3).

Background Acoustic Fields

The **Background Acoustic Fields** makes it possible to define a background field in a domain. This condition can be used to model scattering problems or as an inlet-like condition. In the latter case, when a perfectly matched layer is also present, you can set up a model with an incident field that also lets any reflected waves leave the computational domain. The feature is similar to the Background Pressure Field feature in Pressure Acoustics and the Background Acoustic Fields in Thermoviscous Acoustics.

This condition defines the background fields at the domain level ($\rho_b, \mathbf{u}_b, p_b$). The total acoustic field is now the sum of the scattered field (the dependent variables solved for) and the background field, such that

$$\rho_t = \rho + \rho_b \qquad \mathbf{u}_t = \mathbf{u} + \mathbf{u}_b \qquad p_t = p + p_b$$

On interior boundaries continuity in the total field is automatically applied. All boundary conditions are expressed in terms if the total fields.

BACKGROUND ACOUSTIC FIELDS

Enter values or expressions for:

- **Background acoustic density** ρ_b (SI unit: kg/m^3).
- **Background acoustic velocity** $\mathbf{u}_b$ (SI unit: m/s).
- **Background acoustic pressure** p_b (SI unit: Pa).

Several predefined variables exist for the total, background, and scattered field: the temperature, the entropy, the intensity, and the sound pressure level. They are located in the plot menu group **Background and scattered fields**, when postprocessing.

Prescribed Acoustic Fields

The **Prescribed Acoustic Fields** condition allows the user to prescribe one or more of the dependent variables at a boundary. When specifying (constraining) the dependent variables for the linearized Euler equations, it is necessary to not under-constrain the system. Typically, this requires defining both the density and pressure, defining the velocity, or defining all three at the same time.

For systems that can be assumed isentropic, where the acoustic changes in the entropy are zero, enter an expression for the pressure `p_user` and the density `p_user/lef.c0^2`. This corresponds to the classical relation $p = c_0^2\rho$. This is also achieved using the Pressure (Isentropic) condition.

PRESCRIBED VALUES FOR ACOUSTIC VARIABLES

By default no check boxes are selected. Click to select the following check boxes as needed.

- **Prescribed density** ρ_p (SI unit: kg/m^3).
- **Prescribed velocity** (SI unit: m/s). If this check box is selected, choose one or more of these additional check boxes: **Prescribed in x direction** u_{px}, **Prescribed in y direction** u_{py}, or **Prescribed in z direction** u_{pz} (for 3D components).
- **Prescribed pressure** p_p (SI unit: Pa).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Pressure (Isentropic)

The **Pressure (Isentropic)** node is used for prescribing a pressure p_p at a boundary assuming the isentropic relation between pressure and density (adiabatic relation). This means defining

$$p_t = p_p \qquad \rho_t = \frac{p_p}{c_0^2}$$

PRESSURE

Enter a value or expression for the prescribed pressure p_p (SI unit: Pa).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Symmetry

The **Symmetry** node adds a symmetry condition at a boundary; it acts the same as the Rigid Wall condition.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Impedance and Interior Impedance

Use the **Impedance** or **Interior Impedance** condition to specify an acoustic impedance on an exterior boundary or as a transfer impedance on an interior boundary. The impedance can be any expression and can, for example, be a function of the frequency (`freq`). The condition can be used to, for example, model a porous lining or specifying an outlet impedance. The condition is based on Myers' condition, see Ref. 1 and About the Impedance Boundary Condition located in the Theory Background for the Aeroacoustics Branch section. The condition is a so-called low frequency approximation; the viscous boundary layer of the background flow is assumed infinitely thin at the impedance wall.

IMPEDANCE/INTERIOR IMPEDANCE

Enter the value of the **Normal impedance** Z_n (SI unit: Pa·s/m). Click to select the **Wall impedance at solid boundary** check box if the impedance is at a wall with no normal

velocity component (slip condition) for the background flow. This simplifies the equations solved for.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Moving Wall

The **Moving Wall** condition is used to model a vibrating wall with harmonic variations. The condition is a variant of Myers' condition used for the **Impedance**. See Ref. 1 and About the Moving Wall Boundary Condition located in the Theory Background for the Aeroacoustics Branch section for further details.

MOVING WALL

Select a **Displacement**: **Inward normal displacement** v_n (SI unit: m) (the default), **Displacement field** $\mathbf{v}_w$ (SI unit: m), **Inward normal velocity** u_n (SI unit: m/s), or **Velocity field** $\mathbf{u}_w$(SI unit: m/s). Then enter values or expressions based on the selection.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Interior Wall

The **Interior Wall** boundary condition is used to model a wall condition on an interior boundary. It is similar to the Rigid Wall boundary condition available on exterior boundaries except that it applies on both sides (up and down) of an internal boundary. It allows discontinuities (in density, velocity, and pressure) across the boundary. Use the Interior Wall boundary condition to avoid meshing thin structures by applying this slip-like condition on interior curves and surfaces instead.

Asymptotic Far-Field Radiation

Use the **Asymptotic Far-Field Radiation** node to prescribe open non-reflecting radiation conditions. The boundary condition is based on the asymptotic behavior of the acoustic waves in the far field. The condition is also known as the condition of Tam and Dong (see Ref. 17). The condition applies at boundaries far from a compact acoustic source. The waves must be assumed spherical in 3D (and 2D axisymmetric) or cylindrical in 2D. This means that the condition is valid in the limit where

$$kr \gg 1 \qquad r = |\mathbf{r} - \mathbf{r}_0| \qquad k = \frac{\omega}{c_0}$$

where r is the distance from the source located at $\mathbf{r}_0$ and k is the wavenumber. The condition is formulated for the scattered field only, in problems involving an Background Acoustic Fields feature. The condition of Tam and Dong reads:

$$\left[\frac{i\omega}{V(\theta)} + \frac{\partial}{\partial r} + \frac{1}{\eta r}\right]\mathbf{U} = 0 \qquad \mathbf{U} = \begin{bmatrix} \rho \\ \mathbf{u} \\ p \end{bmatrix}$$

$$V(\theta) = \mathbf{u}_0 \cdot \mathbf{e}_r + \sqrt{c_0^2 - (\mathbf{u}_0 \cdot \mathbf{e}_\theta + \mathbf{u}_0 \cdot \mathbf{e}_\phi)}$$

where $\eta = 1$ in the 3D (and 2D axisymmetric) spherical case and $\eta = 2$ in the 2D cylindrical case.

The acoustic perturbations behave differently than the entropy and vorticity waves. The acoustic waves propagate in all directions at the speed of sound (correct by the background flow) while the entropy and vorticity waves are only convected by the background flow. The linearized Euler equations support both these waves so both have to be taken care off. At boundaries where the background flow leaves the computational domain, add the Outflow Boundary sub-feature. A default Outflow Boundary node is added with a cleared selections.

ASYMPTOTIC FAR-FIELD RADIATION

Enter a **Source location** $\mathbf{r}_0$ (SI unit: m).

Outflow Boundary

A default **Outflow Boundary** sub-node is added to the Asymptotic Far-Field Radiation. The outflow condition has to be added where the background mean flow leaves the computational domain to ensure that vorticity and entropy waves are not created here.

When solving, the interface can return an error if the condition is added where there is no background mean flow, $\mathbf{u}_0 = \mathbf{0}$. Here certain terms evaluate to zero. Adding the condition at boundaries where the background mean flow enters the computational domain can result in unphysical results.

Continuity

Continuity is available as an option at interfaces between parts in a pair. This condition gives continuity in all the dependent variables: density, velocity and pressure. It corresponds to a situation where the boundary has no direct effect on the acoustic fields (subscripts “src” and “dest” in the equation refers to the source and destination, the two sides of the pair):

$$\mathbf{U}_{\text{src}} - \mathbf{U}_{\text{dest}} = 0, \qquad \mathbf{U} = (\rho_t, \mathbf{u}_t, p_t)^T$$

PAIR SELECTION

Select an identity pair. A pair is automatically created in the geometry when **Create pairs** is checked in the **Form Union/Assembly** step.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

The Linearized Euler, Transient Interface

The **Linearized Euler, Transient (let)** interface ( ), found under the **Acoustics>Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in density, velocity, and pressure in the presence of a stationary background mean-flow that is well approximated by an ideal gas flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized Euler equations.

The equations defined by the Linearized Euler, Transient interface are the linearized continuity, momentum (Euler), and energy equations. The physics interface solves for the acoustic variations in the density ρ, velocity field $\mathbf{u}$, and pressure p. The equations are formulated in the time domain. The background mean flow can be any stationary gas flow that is well approximated by an ideal gas. The coupling between the acoustic field and the background flow does not include any predefined flow induced noise. As the equations do not include any loss mechanisms, non-acoustic modes and instabilities can be modeled in the time domain.

The equations are defined using a scattered-field formulation just as in The Linearized Euler, Frequency Domain Interface and allows the use of Background Acoustic Fields.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Euler Model**, **Rigid Wall**, and **Initial Values**. For axisymmetric components an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Euler** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `let`.

TRANSIENT SOLVER SETTINGS

Select the **Time stepping** (method) — **Manual** (default and recommended) or **Automatic/free** and then enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

Details about Time Stepping in Transient Models are found in the Modeling with the Aeroacoustics Branch section.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Density** *rho,* **Velocity field** u and its components, and **Pressure** p. The name can be changed but the names of fields and dependent variables must be unique within a model.

- Domain, Boundary, and Pair Nodes for the Linearized Euler, Transient Interface
- Theory Background for the Aeroacoustics Branch

For modeling tips and tricks and good practice see the Modeling with the Aeroacoustics Branch section.

Domain, Boundary, and Pair Nodes for the Linearized Euler, Transient Interface

The Linearized Euler, Transient Interface has these nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

This physics interface shares some of its node settings with The Linearized Euler, Frequency Domain Interface.

- Asymptotic Far-Field Radiation[1]
- Axial Symmetry[1]
- Background Acoustic Fields[1]
- Continuity[1]
- Domain Sources[1]
- Initial Values
- Interior Wall[1]
- Linearized Euler Model[1]
- Moving Wall
- Rigid Wall[1]
- Prescribed Acoustic Fields[1]
- Pressure (Isentropic)[1]
- Symmetry[1]

[1]This feature is described in the Linearized Euler, Frequency Domain interface.

Initial Values

The **Initial Values** node adds initial values for the density, the velocity field, pressure, and the first time derivatives for each variable. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial values of the:

- **Density** rho (SI unit: kg/m^3).
- **Density, first time derivative** $\partial rho/\partial t$ (SI unit: $kg/(m^3 \cdot s)$).
- **Velocity field** $\mathbf{u}$ (SI unit: m/s).
- **Velocity field, first time derivative** $\partial \mathbf{u}/\partial t$ (SI unit: m/s^2).

- **Pressure** p (SI unit: Pa).
- **Pressure, first time derivative** $\partial p/\partial t$ (SI unit: Pa/s).

Moving Wall

The **Moving Wall** condition can in the time domain only be defined in terms of the normal displacement. It is used to define a vibrating or moving wall. The displacement can be any time dependent expression. The condition can, for example, be used to model an actuator creating sound in a gas flow meter.

MOVING WALL

Enter a **Normal displacement** v_{n} (SI unit: m).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

The Linearized Navier-Stokes, Frequency Domain Interface

The **Linearized Navier-Stokes, Frequency Domain (lnsf)** interface ( ), found under the **Acoustics>Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in pressure, velocity, and temperature in the presence of any stationary isothermal or nonisothermal background mean-flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized Navier-Stokes equations.

The equations are formulated in the frequency domain and assume harmonic variation of all sources and fields. The equations include viscous losses and thermal conduction as well as the heat generated by viscous dissipation, if relevant. The coupling between the acoustic field and the background flow does not include any predefined flow induced noise.

The equations defined by the Linearized Navier-Stokes, Frequency Domain interface are the linearized continuity, momentum (Navier-Stokes), and energy equations. The physics interface solves for the acoustic variations in the pressure p, velocity field $\mathbf{u}$, and temperature T. The equations are formulated in the frequency domain for any fluid including losses due to viscosity and thermal conduction. The background mean flow can be any stationary flow.

The Linearized Navier-Stokes, Frequency Domain interface is formulated in the so-called scattered field formulation where the total acoustic field (subscript t) is the sum of the scattered field (the field solved for p, $\mathbf{u}$, and T) and a possible background acoustic field (subscript b), such that

$$p_\mathrm{t} = p + p_\mathrm{b} \qquad \mathbf{u}_\mathrm{t} = \mathbf{u} + \mathbf{u}_\mathrm{b} \qquad T_\mathrm{t} = T + T_\mathrm{b}$$

All governing equations and boundary conditions are formulated in the total field variables. When no Background Acoustic Fields feature is present (the background field values are zero per default) the total field is simply the field solved for

$$p_\mathrm{t} = p \qquad \mathbf{u}_\mathrm{t} = \mathbf{u} \qquad T_\mathrm{t} = T$$

Coupling the interfaces to structures enables detailed vibration analysis of structures in the presence of flow, such as FSI in the frequency domain. The coupling in the frequency domain and time domain is readily performed using the predefined

Aeroacoustic-Structure Boundary multiphysics coupling feature.

	For modeling tips and tricks and good practice see the Modeling with the Aeroacoustics Branch section.

	The *Helmholtz Resonator with Flow: Interaction of Flow and Acoustics* tutorial model gives an example of how to model the detailed interaction between flow and acoustics. The model requires both the Acoustics Module and the CFD Module. The Application Library path is **Acoustics_Module/Aeroacoustics_and_Noise/helmholtz_resnoator_with_flow**

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Navier-Stokes Model**, **Wall**, and **Initial Values**. For axisymmetric components, an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Navier-Stokes, Frequency Domain** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `lnsf`.

EQUATION

Expand the **Equation** section to see the equations solved for with the **Equation form** specified. The default selection is **Equation form** is set to **Study controlled**. The available studies are selected under **Show equations assuming**.

- For **Study controlled**, the scaling of the equations is optimized for the numerical performance of the different solvers and study types.
- For **Frequency domain** you can manually enter the scaling parameter Δ under **Linearized Navier-Stokes Equation Settings** section.

LINEARIZED NAVIER-STOKES EQUATION SETTINGS

For all component dimensions, and if required, click to expand the **Equation** section, then select **Frequency domain** as the **Equation form** and enter the settings as described below.

The default **Scaling factor** Δ is $1/(i\omega)$. This values correspond to the equations for a Frequency Domain study when the equations are study controlled. To get the equations corresponding to an Eigenfrequency study, change the **Scaling factor** Δ to 1. Changing the scaling factor influences the coupling to other physics.

SOUND PRESSURE LEVEL SETTINGS

See Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface.

TYPICAL WAVE SPEED

See Typical Wave Speed for the Pressure Acoustics, Frequency Domain interface.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Pressure** p, **Velocity field** u and its components, and **Temperature variation** T. The name can be changed but the names of fields and dependent variables must be unique within a model.

STABILIZATION

To display this section, click the **Show** button ( ) and select the **Stabilization Method** — **No stabilization applied**, **Galerkin least squares (GLS) stabilization** (the default), **Streamline upwind Petrov-Galerkin (SUPG) stabilization**, or **Streamline diffusion (legacy method)**. When stabilization is selected enter a value for the **Stabilization constant** α_{stab} (dimensionless). The default value is `1e-2` and should typically have a numerical value between `1` and `1e-3`.

The default GLS stabilization is the most efficient stabilization method as it operates on the convective, reactive, and diffusive parts of the governing equations. This is also the default method and the method suggested for most applications. The stabilization constant α_{stab} can be tuned depending on the problem solved, the nature of the background mean flow, and on the computational mesh.

- Stabilization in the Modeling with the Aeroacoustics Branch section.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. From the list select the element order and type (Lagrange or serendipity) the default is **Linear** for all the dependent variables.

	Choosing between Lagrange and Serendipity Shape Functions has influence on the number of DOFs solved for and on stability for distorted mesh.

	In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

	• Domain, Boundary, and Pair Nodes for the Linearized Navier-Stokes, Frequency Domain and Transient Interfaces • Theory Background for the Aeroacoustics Branch

	As a tutorial model showcasing acoustic-structure interaction in the presence of flow, that is FSI in the frequency domain, see the Application Gallery model: *Vibrating Plate in a 2D Viscous Parallel Plate Flow*. http://www.comsol.com/model/vibrating-plate-in-a-2d-viscous-parallel-plate-flow-18871 The model solves the Linearized Navier-Stokes equations coupled to Solid Mechanics in the presence of flow. The model requires the Acoustics Module and the CFD Module.

Domain, Boundary, and Pair Nodes for the Linearized Navier-Stokes, Frequency Domain and Transient Interfaces

The Linearized Navier-Stokes, Frequency Domain Interface has these domain, boundary, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon

toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Adiabatic
- Axial Symmetry
- Background Acoustic Fields
- Continuity
- Domain Sources
- First-Order Material Parameters
- Heat Flux
- Initial Values
- Interior Normal Impedance
- Interior Wall
- Isothermal
- Linearized Navier-Stokes Model
- Prescribed Pressure
- Prescribed Temperature
- Prescribed Velocity
- Pressure (Adiabatic)
- No Stress
- Normal Impedance
- Normal Stress
- No Slip
- Slip
- Stress
- Wall

These nodes are described for various other physics interfaces:

- Symmetry
- Periodic Condition

Linearized Navier-Stokes Model

The **Linearized Navier-Stokes Model** sets up the governing equations, defines the background mean flow, fluid properties, and the compressibility and thermal expansion properties of the fluid. The governing equations solved are the continuity, momentum, and energy equations:

$$
\begin{aligned}
&\frac{\partial \rho_t}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u}_t + \rho_t \mathbf{u}_0) = M \\
&\rho_0\left(\frac{\partial \mathbf{u}_t}{\partial t} + (\mathbf{u}_t \cdot \nabla)\mathbf{u}_0 + (\mathbf{u}_0 \cdot \nabla)\mathbf{u}_t\right) + \rho_t(\mathbf{u}_0 \cdot \nabla)\mathbf{u}_0 = \nabla \cdot \boldsymbol{\sigma} + \mathbf{F} \\
&\rho_0 C_p\left(\frac{\partial T_t}{\partial t} + (\mathbf{u}_t \cdot \nabla)T_0 + (\mathbf{u}_0 \cdot \nabla)T_t\right) + \rho C_p(\mathbf{u}_0 \cdot \nabla)T_0 \\
&-\alpha_p T_0\left(\frac{\partial p_t}{\partial t} + (\mathbf{u}_t \cdot \nabla)p_0 + (\mathbf{u}_0 \cdot \nabla)p_t\right) - \alpha_p T_t(\mathbf{u}_0 \cdot \nabla)p_0 = \nabla \cdot (k\nabla T_t) + \Phi + Q
\end{aligned}
\tag{4-2}
$$

where p_t, $\mathbf{u}_t$, and T_t are the acoustic perturbations to the pressure, velocity, and temperature, respectively. The subscript "t" refers to the fact that the acoustic variables are the total fields, that is, the sum of possible Background Acoustic Fields and the scattered fields.

In the frequency domain, the time derivatives of the dependent variables are replaced by multiplication with $i\omega$. The stress tensor is $\boldsymbol{\sigma}$ and Φ is the viscous dissipation function. The right-hand-side source terms M, $\mathbf{F}$, and Q are initially zero; they can be defined using the Domain Sources feature. The variables with a zero subscript are the background mean flow values. The material parameters are defined below. Details about the physics and references are found in the Theory Background for the Aeroacoustics Branch section.

The constitutive equations are the stress tensor and the linearized equation of state, while the Fourier heat conduction law is readily included in the above energy equation,

$$
\begin{aligned}
\boldsymbol{\sigma} &= -p_t\mathbf{I} + \mu(\nabla \mathbf{u}_t + (\nabla \mathbf{u}_t)^T) + \left(\mu_B - \frac{2}{3}\mu\right)(\nabla \cdot \mathbf{u}_t)\mathbf{I} \\
\rho_t &= \rho_0(\beta_T p_t - \alpha_p T_t)
\end{aligned}
\tag{4-3}
$$

The linearized viscous dissipation function is defined as

$$
\begin{aligned}
\Phi &= \nabla \mathbf{u}_t : \boldsymbol{\tau}(\mathbf{u}_0) + \nabla \mathbf{u}_0 : \boldsymbol{\tau}(\mathbf{u}_t) \\
\boldsymbol{\tau}(\mathbf{u}_t) &= \mu(\nabla \mathbf{u}_t + (\nabla \mathbf{u}_t)^T) + \left(\mu_B - \frac{2}{3}\mu\right)(\nabla \cdot \mathbf{u}_t)\mathbf{I} \\
\boldsymbol{\tau}(\mathbf{u}_0) &= \mu(\nabla \mathbf{u}_0 + (\nabla \mathbf{u}_0)^T) + \left(\mu_B - \frac{2}{3}\mu\right)(\nabla \cdot \mathbf{u}_0)\mathbf{I}
\end{aligned}
\tag{4-4}
$$

MODEL INPUTS

In order to model the influence of the background mean flow on the propagation of the acoustic waves in the fluid, the background mean flow temperature T_0, absolute

pressure p_0, and velocity field $\mathbf{u}_0$ need to be defined. The density is defined in the **Fluid Properties** section below, and is per default taken from the material. It is thus a function of the model inputs, that is, the background pressure and temperature. Enter **User defined** values for the:

- **Background mean flow temperature** T_0 (SI unit K). The default is 293.15 K.
- **Background mean flow pressure** p_0 (SI unit: Pa). The default is 1 atm.
- **Background mean flow velocity** $\mathbf{u}_0$ (SI unit: m/s). The defaults are 0 m/s.

When modeling aeroacoustics it is important how the Mapping Between CFD and Acoustics Mesh is done from a numerical perspective. Physically the Coupling to Turbulent Flows (Eddy Viscosity) is also important to model the attenuation of acoustics waves due to turbulence.

FLUID PROPERTIES

The defaults for the following are taken **From material**. For **User defined** edit the default values:

- **Background mean flow density** ρ_0 (p_0, T_0) (SI unit: kg/m^3).
- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Bulk viscosity** μ_B (SI unit: Pa·s).
- **Thermal conductivity** k (SI unit: W/(m·K)).
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)).

THERMAL EXPANSION AND COMPRESSIBILITY

Select an option from the **Coefficient of thermal expansion** α_p list: **From material** (the default) or **User defined**. For **User defined** enter a value for α_p (SI unit: 1/K). The subscript p refers to the fact that this is the isobaric coefficient of thermal expansion.

Select an option from the **Isothermal compressibility** β_T list: **From isentropic compressibility** (the default), **From speed of sound**, or **User defined**.

- For **From isentropic compressibility** the values for the **Isentropic compressibility** β_s (SI unit: 1/Pa) and **Ratio of specific heats** γ (dimensionless) are taken **From material**. For **User defined** enter a different values or expressions.

- For **From speed of sound** the values for the **Speed of sound** c (SI unit: m/s) and **Ratio of specific heats** γ (dimensionless) are taken **From material**. For **User defined** enter a different value or expression.
- For **User defined** enter a value for β_T (SI unit: 1/Pa).

VISCOUS DISSIPATION FUNCTION

Select the **Include viscous dissipation function** check box if you wish to include the heat source generated by the viscous losses. The viscous dissipation function Φ is defined in Equation 4-4.

Wall

The **Wall** node is the default boundary condition, which is used to model the most common conditions at solid surfaces. This condition contains both a mechanical and a thermal selection. The default is a no-slip and isothermal condition applicable in most cases. More advanced conditions at boundaries can be set-up by combining any of the Mechanical and Thermal conditions available.

The no-slip condition is the origin of the viscous boundary layer and the isothermal condition is the origin of the thermal boundary layer. It is within these acoustic boundary layers that the main dissipation happens. In some applications like for example, in large muffler systems, it may not be necessary to model these loss effects and thus switching to slip and adiabatic can be a good approximation. This also means that the mesh does not need to resolve the acoustic boundary layer which can save DOFs.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip**. See the No Slip and Slip conditions for further details.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

If **Slip** is selected for the **Mechanical condition** and **Use weak constraints** is enabled then the default discontinuous Galerkin (penalty like) formulation is switched to a Lagrange multiplier formulation instead. See the Slip condition for details. For the **No-slip** and

Isothermal selections a weak formulation is used instead of a point-wise constraint.

Axial Symmetry

In a 2D axisymmetric geometry the **Axial Symmetry** condition is automatically added on the axis of symmetry at $r = 0$. On this boundary the radial component of the total velocity is set equal to zero

$$u_t = 0 \qquad \mathbf{u}_t = (u_t, v_t, w_t)$$

Interior Wall

The **Interior Wall** boundary condition is used to model a wall condition on an interior boundary. It is similar to the Wall boundary condition available on exterior boundaries except that it applies on both sides (up and down) of an internal boundary. It allows discontinuities (in pressure, velocity, and temperature) across the boundary. Use the Interior Wall boundary condition to avoid meshing thin structures by applying this slip-like condition on interior curves and surfaces instead.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip**. See the No Slip and Slip conditions for further details.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

If **Slip** is selected for the **Mechanical condition** and **Use weak constraints** is enabled then the default discontinuous Galerkin (penalty like) formulation is switched to a Lagrange multiplier formulation instead. For the **No-slip** and **Isothermal** selections a weak formulation is used instead of a point-wise constraint.

No Slip

The **No Slip** sets up a no-slip condition for the flow on a hard wall:

$$\mathbf{u}_t = \mathbf{0}$$

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

	Mechanical and thermal boundary conditions contribute with each other such that a condition can be set on the velocity and temperature simultaneously. Thermal type condition override each other and the same is true for mechanical type conditions.

Isothermal

The **Isothermal** node sets up an isothermal condition:

$$T_t = 0$$

This is a good approximation on solid walls, as heat conduction is typically much higher in solids than in fluids.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Initial Values

The **Initial Values** node adds initial values for the pressure, the velocity field, and temperature. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial values of the **Pressure** p (SI unit: Pa), the **Velocity field** $\mathbf{u}$ (SI unit: m/s), and the **Temperature** T (SI unit: K).

Interior Normal Impedance

Use the **Interior Normal Impedance** node to apply a transfer impedance condition on an interior boundary. The condition relates the normal stress at the two sides of the boundary (up and down) to the normal velocity via the normal transfer impedance Z_n. The condition allows discontinuities (a slit) in pressure and temperature depending on the condition chosen for the temperature.

$$(\boldsymbol{\sigma}_{up}-\boldsymbol{\sigma}_{down})\mathbf{n} = -Z_n(\mathbf{u}_t \cdot \mathbf{n})\mathbf{n}$$

INTERIOR NORMAL IMPEDANCE

Enter a value or expression for the **Normal impedance** Z_n (SI unit: Pa·s/m).

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

First-Order Material Parameters

To display this node in the context menu, click the **Show** button ( ) and select **Advanced Physics Options**.

Use the **First Order Material Parameters** node to include subtle (acoustic) variations in the material parameters due to the acoustic variations of the dependent variables. The selected material parameters vary according to a linearization about their background values (at pressure p_0 and temperature T_0). The viscosity μ is, for example, replaced by:

$$\mu \rightarrow \mu + p\frac{\partial \mu}{\partial p}\bigg|_{T_0} + T\frac{\partial \mu}{\partial T}\bigg|_{p_0}$$

FIRST-ORDER MATERIAL PARAMETERS

By default no check boxes are selected. Click to select the following check boxes as needed.

- **Derivatives of dynamic viscosity** $\partial \mu / \partial p$ (SI unit: s) and $\partial \mu / \partial T$ (SI unit: kg/(m·s·K)).
- **Derivatives of bulk viscosity** $\partial \mu_B / \partial p$ (SI unit: s) and $\partial \mu_B / \partial T$ (SI unit: kg/(m·s·K)).
- **Derivatives of heat capacity at constant pressure** $\partial C_p / \partial p$ (SI unit: $m^3/(kg \cdot K)$) and $\partial C_p / \partial T$ (SI unit: $m^2/(s^2 \cdot K^2)$).
- **Derivatives of thermal conduction** $\partial k / \partial p$ (SI unit: $m^2/(s \cdot K)$) and $\partial k / \partial T$ (SI unit: $m \cdot kg / (s^2 \cdot K^2)$).
- **Derivatives of coefficient of thermal expansion** $\partial \alpha_p / \partial p$ (SI unit: $m \cdot s^2/(kg \cdot K)$) and $\partial \alpha_p / \partial T$ (SI unit: $1/K^2$).

VISCOUS DISSIPATION FUNCTION

Select the **Include viscous dissipation function** check box to include the effects of the redefined material parameters on the viscous dissipation function.

Domain Sources

Add a **Domain Sources** node to define the mass source, volume force source, and heat source, the right-hand-side of Equation 4-2. These can, for example, model varying thermal sources in a combustion chamber or a pulsating laser.

DOMAIN SOURCES

Enter values or expressions for the following:

- **Mass source** M (SI unit: kg/(m^3·s)).
- **Volume force source** $\mathbf{F}$ (SI unit: N/m^3).
- **Heat source** Q (SI unit: W/m^3).

Background Acoustic Fields

When the **Background Acoustic Fields** feature is added to a domain, it is possible to define the value of the background acoustic field variables p_b, $\mathbf{u}_b$, and T_b. Using this feature, it is possible to set up scattering problems as well as defining acoustic fields at an inlet of a waveguide (using a small domain at the inlet).

BACKGROUND ACOUSTIC FIELDS

Enter expressions for the **Background acoustic pressure** p_b, the **Background acoustic velocity** $\mathbf{u}_b$, and the **Background temperature variation** T_b. These can be analytical expressions or values of dependent variables solved in a previous study that defines the background field

Pressure (Adiabatic)

Add a **Pressure (Adiabatic)** node to give a pressure boundary condition with adiabatic conditions for the temperature. The condition is given by

$$p_t = p_p \qquad \boldsymbol{\sigma}\mathbf{n} = p_p\mathbf{n} \qquad -\mathbf{n}\cdot(-k\nabla T_t) = 0$$

where p_p is the desired pressure at the boundary, $\boldsymbol{\sigma}$ is the stress tensor, and $\mathbf{n}$ the surface normal. This is a good approximation when prescribing a pressure at an inlet or outlet.

PRESSURE (ADIABATIC)

Enter a value or expression for the **Pressure** p_p (SI unit: Pa).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Slip

Add a **Slip** node to define slip for the velocity defined by

$$\mathbf{n} \cdot \mathbf{u}_t = 0$$
$$\boldsymbol{\sigma}_n - (\boldsymbol{\sigma}_n \cdot \mathbf{n})\mathbf{n} = 0, \qquad \boldsymbol{\sigma}_n = \boldsymbol{\sigma}\mathbf{n}$$

where $\boldsymbol{\sigma}$ is the stress tensor. The last equation expresses that no tangential stress exists at that boundary.

This results in a so-called no-penetration condition where no viscous boundary layer is created. This condition can be used at boundaries where it is not necessary to model the losses in the viscous boundary layer. Used together with the Adiabatic condition, no acoustic boundary layer is modeled.

Per default, the **Slip** condition uses a so-called discontinuous Galerkin or penalty formulation to prevent locking problems on curved surfaces.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Select **Use weak constraints** if you want to switch from the default discontinuous Galerkin (penalty like) formulation to a Lagrange multiplier formulation for the slip condition (this formulation is not suited for an iterative solver). The Lagrange multiplier formulation also prevents locking problems.

Prescribed Velocity

Add a **Prescribed Velocity** node to define the velocity at a boundary.

PRESCRIBED VELOCITY

Select the components to prescribe and enter a value for the **Prescribed in the direction** (SI unit: m/s).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Prescribed Pressure

Add a **Prescribed Pressure** node to prescribe the pressure at a boundary (using a constraint).

PRESCRIBED PRESSURE

Enter a **Pressure** p_p (SI unit: Pa).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Normal Stress

Add a **Normal Stress** node to define the normal stress vector σ_n at a boundary:

$$\boldsymbol{\sigma}\mathbf{n} = \sigma_n$$

where $\boldsymbol{\sigma}$ is the stress tensor and $\mathbf{n}$ the surface normal.

NORMAL STRESS

Enter a **Normal stress** σ_n (SI unit: $\mathrm{N/m^2}$).

No Stress

Use the **No Stress** node to set the total surface stress equal to zero:

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + \nabla\mathbf{u}_t^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = \mathbf{0}$$

Normal Impedance

Use the **Normal Impedance** node to specify a normal impedance Z_n on a boundary. This feature is useful outside the viscous boundary layer, as this condition mimics the behavior of a corresponding **Pressure Acoustics Model** with a normal impedance condition. The boundary condition reads:

$$\left[-p_t\mathbf{I}+\mu(\nabla\mathbf{u}_t+\nabla\mathbf{u}_t^{\mathrm{T}})-\left(\frac{2\mu}{3}-\mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n}=-Z_n(\mathbf{u}_t\cdot\mathbf{n})\mathbf{n}$$

NORMAL IMPEDANCE

Enter a value or expression for the **Normal impedance** Z_n (SI unit: Pa·s/m).

Stress

Add a **Stress** node to define the prescribed stress $\boldsymbol{\sigma}_p$ at a boundary:

$$\boldsymbol{\sigma}\mathbf{n} = \boldsymbol{\sigma}_p\mathbf{n}$$

where $\boldsymbol{\sigma}$ is the stress tensor and $\mathbf{n}$ the surface normal.

STRESS

Select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the stress, and enter scalar or vector values or expressions for the **Stress** σ_p (SI unit: N/m^2).

Adiabatic

Add an **Adiabatic** node to set up an adiabatic condition for the temperature, that is, the natural condition for the temperature

$$-\mathbf{n}\cdot(-k\nabla T_t) = 0$$

Prescribed Temperature

Add a **Prescribed Temperature** node to define the temperature variation T_p at a boundary.

PRESCRIBED TEMPERATURE

Enter a value or expression for the **Temperature** T_p (SI unit: K).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Heat Flux

Add a **Heat Flux** node to define the inward normal heat flux q_n at a boundary:

$$-\mathbf{n} \cdot (-k\nabla T_t) = q_n$$

HEAT FLUX

Enter a value or expression for the **Inward normal heat flux** q_n (SI unit: W/m^2).

Continuity

Continuity is available as an option at interfaces between parts in a pair. This condition gives continuity in all the dependent variables: pressure, velocity, and temperature. It corresponds to a situation where the boundary has no direct effect on the acoustic fields (subscripts "src" and "dest" in the equation refers to the source and destination, the two sides of the pair):

$$\mathbf{U}_{src} - \mathbf{U}_{dest} = 0, \qquad \mathbf{U} = (p_t, \mathbf{u}_t, T_t)^T$$

The Linearized Navier-Stokes, Transient Interface

The **Linearized Navier-Stokes, Transient (Inst)** interface ( ), found under the **Acoustics>Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in pressure, velocity, and temperature in the presence of any stationary isothermal or nonisothermal background mean-flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized Navier-Stokes equations.

The equations are formulated in the time domain and include viscous losses and thermal conduction as well as the heat generated by viscous dissipation, if relevant. The coupling between the acoustic field and the background flow does not include any predefined flow induced noise.

The equations defined by the Linearized Navier-Stokes, Transient interface are the linearized continuity, momentum (Navier-Stokes), and energy equations. The physics interface solves for the acoustic variations in the pressure, velocity, and temperature. The equations are formulated in the time domain for any fluid including losses due to viscosity and thermal conduction. The background mean flow can be any stationary flow.

The governing equations are defined using a scattered field formulation just as in the The Linearized Navier-Stokes, Frequency Domain Interface and can use Background Acoustic Fields feature.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Navier-Stokes Model**, **Wall**, and **Initial Values**. For axisymmetric components, an **Axial Symmetry** node is also added. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Navier-Stokes, Transient** to select physics features from the context menu.

For modeling tips and tricks and good practice see the Modeling with the Aeroacoustics Branch section.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `lnst`.

TRANSIENT SOLVER SETTINGS

Select the **Time stepping** (method) as **Manual** (default and recommended) or **Automatic/free** and then enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

Details about Time Stepping in Transient Models are found in the Modeling with the Aeroacoustics Branch section.

The Stabilization, Dependent Variables, and Discretization settings are the same as for The Linearized Navier-Stokes, Frequency Domain Interface.

For boundary conditions and domain conditions see:

- Domain, Boundary, and Pair Nodes for the Linearized Navier-Stokes, Frequency Domain and Transient Interfaces
- Theory Background for the Aeroacoustics Branch

Modeling with the Aeroacoustics Branch

In this section:

Selecting an Aeroacoustics Interface

When modeling a muffler with an internal nonisothermal flow, a jet engine, or a flow sensor you should consider which physics interface to use. The influence the background mean flow has on the acoustic behavior in an aeroacoustic model can be modeled in several ways. The effects that need to be included typically depend on the Mach number (Ma). A rule of thumb says that for a Mach number below 0.1 ($\mathrm{Ma} < 0.1$) the convective effects of the background flow need not to be included, above they do. Other considerations are of course also important; for example, whether viscous and thermal losses are important, if the background flow has large gradients, or if the flow is turbulent or has vorticity.

- Mach number less than 0.1 ($\mathrm{Ma} < 0.1$)

  In this situation the convective flow effects can normally be neglected and Pressure Acoustics can be used. Only the background temperature distribution $T = T(\mathbf{x})$ and background pressure distribution $p_A = p_A(\mathbf{x})$ need to be included. This can be done directly in the Pressure Acoustics interface as a **Model Inputs**. The effects are included by making sure that the material properties depend on the local pressure and

temperature; the spatial variations in the speed of sound is $c = c(p_A,T) = c(\mathbf{x})$ and in the density $\rho = \rho(p_A,T) = \rho(\mathbf{x})$, respectively.

- Mach number greater than 0.1 ($\mathrm{Ma} > 0.1$):

  In this case the convective effects of the flow can probably not be neglected. Modeling this type of system requires the use of one of the Aeroacoustics interfaces: linearized potential flow, linearized Euler, or linearized Navier-Stokes. The choice depends on the assumptions about the flow that can be made.

Note that when the background mean flow velocity $\mathbf{u}_0$ is set to zero, the linearized Navier-Stokes equations reduce to the thermoviscous acoustic equations. However, in thermoviscous acoustics the default discretization is P1, P2, P2 and no stabilization is applied.

Meshing

When solving a model using one of the Aeroacoustic interfaces, it is important to remember that this is a wave problem. This means that the wavelength should be resolved by an appropriate number of mesh elements. Using the same guidelines as for Pressure Acoustics is reasonable, that is, a first good mesh should use at least 5 elements per wavelength for second order shape functions. In the linearized Euler interface the default shape functions are first order and here using at least 15 to 20 elements per wavelength is a first good choice.

Meshing guidelines for the Pressure Acoustics interface is given in Meshing (Resolving the Waves) under the Modeling with the Pressure Acoustics Branch section.

The Background Flow

Another important parameter to consider is resolving details in the background mean flow field. The acoustic mesh should capture gradients that exist in the background flow as these have a large influence on the acoustic propagation. Waves may, for example, be reflected and refracted in shear layers.

Meshing the Acoustic Boundary Layers in LNS

The Linearized Navier-Stokes interfaces just as the Thermoviscous Acoustics interface captures the physics of the acoustic boundary layer. When no-slip and isothermal conditions are used on walls a viscous and a thermal boundary layer will exist. In order

for the solution to be well behaved an capture losses correctly it is important to mesh this layer, for example, by using a boundary layer mesh. If the model is large using a single boundary layer mesh, with the approximate extend of the acoustic boundary layer, is the minimum requirement for a good solution.

Resolving Vorticity and Physics in LNS

In the LNS interface the GLS stabilization is very efficient and can ensure smooth and converged solution even without resolving details like vorticity generation (the propagation of vorticity waves). If these are important processes in the model the mesh should of course be able to resolve these details. Either refine the mesh or switch to (P2, P2, P2) discretization (keep the stabilization turned on). These phenomena are typically generated at walls where the no-slip condition generates vorticity when an acoustic wave is interacting with the background flow.

Stabilization

When solving the linearized Euler (LE) and linearized Navier-Stokes (LNS) equations using the finite element method (the standard Galerkin formulation). It can be shown that the method looses its good approximation characteristics when convective terms are present and when these terms locally dominate (Ref. 19). This can lead to spurious numerical oscillations. To remedy these oscillations stabilization is used.

It can be shown that if the cell Péclet number has a value larger than one $Pe^c > 1$ the solution will oscillate. In the linearized Navier-Stokes interface the variables `lnsf.CellPe_th` and `lnsf.CellPe_v` can be plotted to assess the characteristic number when comparing convection to viscous and thermal diffusion, respectively.

In this section:

- Linearized Navier-Stokes Stabilization
- Linearized Euler Stabilization

LINEARIZED NAVIER-STOKES STABILIZATION

In the linearized Navier-Stokes interfaces the two stabilization methods are implemented. The Galerkin least squares (GLS) and the streamline upwind Petrov-Galerkin (SUPG) stabilization. It is in genera recommended to use the GLS method. The SUPG method is implemented for completeness and can be used by

experienced users. There is also the option to not use stabilization or to use the legacy method from version 5.2a and older.

The implementation of the stabilization methods follows the one discussed in Ref. 19 and Ref. 21. A general overview about stabilization methods can be found in Ref. 22. The GLS method combines stability and accuracy and the method order of accuracy is $O(h^{2p+1})$ where p is the shape function order and h is the local mesh size. The SUPG method has an accuracy order of $O(h^{p+1/2})$ for convection dominated problems and $O(h^{p+1})$ for diffusion dominated problems.

The default discretization for the LNS interface is to use linear elements for all the dependent variables (P1-P1-P1). This effectively removes the stabilization on the diffusive parts of the equation. The GLS method is still superior to the SUPG method as it also stabilizes the reactive terms. These are the terms where gradients of the background fields enter. If no stabilization is used then set a (P1-P2-P2) discretization for the dependent variables as it ensures a stable numerical scheme.

The equation residuals can be visualized in the LE and the LNS interfaces by plotting, for example, `lnsf.res_e` (equation residual for the energy equation), `lnsf.res_mx` (equation residual for the momentum equation x-component), `lnsf.res_my`, `lnsf.res_mz`, and `lnsf.res_e` (equation residual for the energy equation).

LINEARIZED EULER STABILIZATION

In the linearized Euler interface the stabilization scheme implements the streamline upwind Petrov-Galerkin (SUPG) formulation of the weak form equations used for the finite element method. The stabilization constant α_{stab} can be tuned depending on the problem solved, the nature of the background mean flow, and on the computational mesh. The implementation follows the one discussed in Ref. 10 and Ref. 14.

See for example the model *Point Source in 2D Jet: Radiation and refraction of sound waves through a 2D shear layer* found in the Model Gallery:
www.comsol.com/model/point-source-in-2d-jet-radiation-and-refraction-of-sound-waves-through-a-2d-shea-16685

Solver Suggestions for Large Aeroacoustic Models

Studying aeroacoustic applications often lead to very large models that can contain many degrees of freedom (DOFs). Different steps can be taken to reduce the models size, for example, consider the use of slip instead of no-slip conditions in the LNS interface (avoiding the details in the acoustic boundary layer) or consider the choice between Lagrange and Serendipity Shape Functions.

When a model reaches a size where it can no longer fit in memory (when solved using the default direct solver) or if the solution procedure is slow with a direct solver, then it can be advantageous to switch to an iterative solver approach.

For the **Linearized Navier-Stokes** interfaces iterative solver suggestions are automatically generated when the default solver is generated. These can be seen by expanding the **Solver Configuration** tree under **Stationary Solver** or **Time-Dependent Solver**. Per default a direct solver is used and two iterative solvers are suggested and disabled (grayed out). To turn on one of these approached right-click the solver and select **Enable** (or press F4). The first suggestion *(GMRES with Direct Precon.)* uses an iterative solver with a direct preconditioner. This method is typically faster then the direct solver and uses 20% less memory. The second suggestion *(GMRES with DD)* uses an iterative solver with the domain decomposition method. This method is very robust (also for multiphysics applications) and very memory efficient, but it can be slow.

Further details on solving large models can be found in the Solver Suggestions for Large Thermoviscous Acoustics Models section. These suggestions also apply fir the linearized Navier-Stokes physics.

Lagrange and Serendipity Shape Functions

In most of the physics interfaces in the Acoustics Module and specifically in the Thermoviscous Acoustic interfaces, you can choose between two families of shape functions: *Lagrange* and *serendipity*. The current default is to use Lagrange shape functions. To display the **Discretization** section, click the **Show** button ( ) and select **Discretization**.

When using a structured mesh it may be advantageous to switch to the serendipity elements as they generate significantly fewer degrees of freedom (DOFs). The accuracy is in most cases almost as good as for the Lagrange elements. The Lagrange elements are however less sensitive to strong mesh distortions.

The serendipity shape functions differs from the Lagrange shape functions only for the following element shapes:

- 2D: Quadrilateral elements of discretization order higher than 1.
- 3D: Hexahedral, prism, and pyramid elements of discretization order higher than 1.

In the *COMSOL Multiphysics Reference Manual*:

- The Lagrange Element
- The Nodal Serendipity Element

When coupling two physics interfaces that have the same DOFs like, for example, displacement, the same type of shape functions should be used in both interfaces to ensure conformity. Since there is no difference between the two families of shape functions in 1D, this is not an issue when connecting edges.

Time Stepping in Transient Models

When solving transient wave problems it is recommended to use manual time stepping in the solver. In general there is no point in using an automatic time-step control which can be provided by the time-dependent solver. The tolerances in the automatic error control are difficult to tune in wave problems when there is weak but important high-frequency content. It is recommended to use the suggestion generated when the **Manual** method is chosen (the default) in the **Transient Solver Settings** section. The internal time step of the solver that is generated, when the **Maximal frequency to resolve** is set, will in most cases produce a solver that is adequate. It is assuming that the user has generated a mesh that properly resolves the same maximal frequency (minimal wavelength).

For further details see the Time Stepping in Transient Models section in Modeling with the Thermoviscous Acoustics Branch.

Mapping Between CFD and Acoustics Mesh

When the aeroacoustic and the CFD model are not solved on the same computational mesh, careful mapping of the CFD solution from the CFD mesh onto the acoustics mesh should be done. This step is crucial in order not to introduce non-physical numerical noise into the acoustic solution (Ref. 20). If the solution is not properly

mapped terms containing gradients of the background mean flow variables become very noisy and are typically the source of the error. If the acoustic problem is solved on the same mesh as the CFD then the mapping problem is less of an issue.

MAPPING STUDY

If two different mesh and/or different shape function orders are used in the acoustic and the CFD models a mapping study needs to be used. This is a study that solves an additional set of equations which maps and smooths the background flow CFD variables onto the acoustics mesh. The mapping equations can be set up using the **Weak Form PDE** interface from the **Mathematics** branch. In the most general case the mean background flow pressure p_0, velocity field $\mathbf{u}_0$, density ρ_0, temperature T_0, and turbulent viscosity μ_T variables should be mapped onto corresponding variables on the acoustics mesh, for example, $p_{0,\text{aco}}$, $\mathbf{u}_{0,\text{aco}}$, $\rho_{0,\text{aco}}$, $T_{0.\text{aco}}$, and $\mu_{T,\text{aco}}$. These new variables should then be used as the model inputs in the aeroacoustics model.

A simple but efficient mapping and smoothing is achieved by solving

$$
\begin{aligned}
p_{0,\,\text{aco}} - p_0 &= \delta h^2 \nabla \cdot (\nabla p_{0,\,\text{aco}}) \\
u_{i,\,0,\,\text{aco}} - u_{i,\,0} &= \delta h^2 \nabla \cdot (\nabla u_{i,\,0,\,\text{aco}}) \\
\rho_{0,\,\text{aco}} - \rho_0 &= \delta h^2 \nabla \cdot (\nabla \rho_{0,\,\text{aco}}) \\
T_{0,\,\text{aco}} - T_0 &= \delta h^2 \nabla \cdot (\nabla T_{0,\,\text{aco}}) \\
\mu_{T,\,\text{aco}} - \mu_T &= \delta h^2 \nabla \cdot (\nabla \mu_{T,\,\text{aco}})
\end{aligned}
$$

where the term on the right hand side adds smoothing using isotropic diffusion. The amount of diffusion is controlled by the parameter δ (a constant that can be tuned with a typical value of 0.01) and the mesh size squared h^2. The term corresponds to so-called source term stabilization as known from CFD.

In the **Weak Form PDE** interface define as many dependent variables as necessary and give them the same shape order as the order used for the acoustics (typically all linear). Assume that the source CFD variable for the x-velocity component is `u` and the new destination variable is `U0`, then the above mapping is achieved with the following **Weak Expression**, entered in the user interface:

```
(U0-withsol('sol1',u,setval(Ma,0.1)))*test(U0)+delta*h^2*(U0x*tes
t(U0x)+U0y*test(U0y)+U0z*test(U0z)
```

For the pressure map the total pressure `spf.pA` to `P0` using:

```
(P0-withsol('sol1',spf.pA,setval(Ma,0.1)))*test(P0)+delta*h^2*(P0
x*test(P0x)+P0y*test(P0y)+P0z*test(P0z))
```

Notice the use of the `withsol()` operator which is an extrusion-like coupling operator that can refer directly to a solution object and parameter value. In this example it is used to fetch data from the CFD mesh, in the solution generated by solver `sol1` and for the parameter value `Ma` = `0.1`. The `setval()` statement is optional. It is important to use such an operator such that the CFD solution is mapped and interpolated correctly to the integration (Gauss) points on the acoustics mesh.

In the *COMSOL Multiphysics Reference Manual*:

- The Weak Form PDE
- Operators, Functions, and Constants and Built-In Operators for details about the `withsol()` operator
- Numerical Stabilization (general introduction)

In some cases it can be necessary to extend the mapping equation with boundary conditions for the mapped variables. This should be done at symmetry planes for the mapped velocity field. This is also the cases when flow details near walls are important, for example, resolving the acoustic boundary layer. In this case add a no-slip condition on $\mathbf{u}_{0,aco}$. In general constraining the value of the mapped variables to the value in the CFD model values at boundaries can be necessary.

The mapping procedure as well as the coupling to a turbulent flow is studied in the application library model *Helmholtz Resonator with Flow: Interaction of Flow and Acoustics*. The Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/helmholtz_resnoator_with_flow**

Coupling to Turbulent Flows (Eddy Viscosity)

When acoustic waves propagate through a turbulent flow they will experience attenuation due to the turbulence. It can be shown theoretically (see Ref. 18 and Ref. 19) that the experienced attenuation corresponds to extending the value of the dynamic viscosity μ to include the eddy viscosity μ_τ, such that the total effective viscosity is:

$$\mu_t = \mu + \mu_\tau$$

This effect can be included in the linearized Navier-Stokes model by using the effective dynamic viscosity as taken from a CFD model `spf.mu_eff`. This variable includes the dynamic viscosity and the turbulent viscosity (`spf.muT`). Notice, however, that the value of the turbulent viscosity can be much larger than the true eddy viscosity and that it depends on the turbulence model selected. This means that the experienced attenuation can be too large. Typically, more advanced turbulence models like the SST model give less numerical diffusion and thus a better estimate of the eddy viscosity.

Eigenfrequency Studies

Both the linearized Euler and the linearized Navier-Stokes interfaces have the Eigenfrequency study as a predefined study type. It should be noted that this analysis type is not suited when a non-zero background mean flow is present. In this case, the current formulation of the equation system, being solved, becomes non-Hermitian. Mathematically this means that there are no discrete eigenvalues but only a continuous spectrum of eigenvalues. It can be very difficult to find the acoustic (physical) eigenvalues.

Postprocessing Variables

Intensity

The flow of energy is expressed by the acoustic intensity $\mathbf{I}$, which is formally defined by the time-averaged sound power per unit area (unit: $\mathrm{W/m^2}$). This quantity is available as a built-in postprocessing variable.

For a general fluid, including thermal and viscous losses (see Ref. 15 and Ref. 16 for details), the time-averaged intensity is given by

$$\mathbf{I} = \frac{1}{T}\int_0^T p\mathbf{u}\,\mathrm{d}t =$$

$$\frac{1}{2}\mathrm{Re}\left[(\rho_0\mathbf{u}+\rho\mathbf{u}_0)\left(\frac{p}{\rho_0}+\mathbf{u}_0\cdot\mathbf{u}\right)^* - \left[(\rho_0\mathbf{u}+\rho\mathbf{u}_0)^{\mathsf{T}}\cdot\left(\frac{1}{\rho_0}\boldsymbol{\tau}-\frac{\rho}{\rho_0^2}\boldsymbol{\tau}_0\right)^*\right]^{\mathsf{T}}\right.$$

$$\left. + \rho_0 T(s\mathbf{u}_0)^* + Tk\left[-\frac{\nabla T}{T_0}+\frac{T}{T_0^2}\nabla T_0\right]^*\right]$$

where the viscous stress tensors for the acoustic and background fields, $\boldsymbol{\tau}$ and $\boldsymbol{\tau}_0$ respectively, are given by (in index notation)

$$\tau_{ij} = \mu\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) - \left(\mu_B - \frac{2}{3}\mu\right)\left(\frac{\partial u_k}{\partial x_k}\right)\delta_{ij}$$

$$\tau_{0ij} = \mu\left(\frac{\partial u_{0j}}{\partial x_i} + \frac{\partial u_{0i}}{\partial x_j}\right) - \frac{2}{3}\mu\left(\frac{\partial u_{0k}}{\partial x_k}\right)\delta_{ij}$$

It is understood that all dependent variables in these equations are the complex amplitudes.

In the time domain, the equivalent quantity is the instantaneous intensity $\mathbf{i}$ given by

$$\mathbf{i} = p\mathbf{v} = (\rho_0\mathbf{u} + \rho\mathbf{u}_0)\left(\frac{p}{\rho_0} + \mathbf{u}_0 \cdot \mathbf{u}\right) - \left[(\rho_0\mathbf{u} + \rho\mathbf{u}_0)^{\mathsf{T}} \cdot \left(\frac{1}{\rho_0}\boldsymbol{\tau} - \frac{\rho}{\rho_0^2}\boldsymbol{\tau}_0\right)\right]^{\mathsf{T}}$$
$$+ \rho_0 T s \mathbf{u}_0 + T k\left[-\frac{\nabla T}{T_0} + \frac{T}{T_0^2}\nabla T_0\right]$$

where the dependent variables now include the explicit time dependence. The instantaneous expression is not defined in the frequency domain since it would represent effects happening at the double frequency.

See also the theory section The Energy Corollary for the Linearized Euler interface.

The intensity variables are defined for most Aeroacoustics physics where it makes sense. That is, the instantaneous quantities are defined in the time domain and the (time averaged) intensity variables are defined in the frequency domain. It is understood that the intensity vector is in general the time averaged quantity. For the Linearized Potential Flow interface, the variables are not defined in the time domain since it would require solving for an extra variable.

TABLE 4-1: INTENSITY VARIABLES IN 3D

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ix	x-component of the intensity vector (frequency domain only)
phys_id.Iy	y-component of the intensity vector (frequency domain only)
phys_id.Iz	z-component of the intensity vector (frequency domain only)
phys_id.Ii_mag	Magnitude of the instantaneous intensity vector (time domain only)

TABLE 4-1: INTENSITY VARIABLES IN 3D

VARIABLE	DESCRIPTION
phys_id.Iix	x-component of the instantaneous intensity vector (time domain only)
phys_id.Iiy	y-component of the instantaneous intensity vector (time domain only)
phys_id.Iiz	z-component of the instantaneous intensity vector (time domain only)

TABLE 4-2: INTENSITY VARIABLES IN 2D AXISYMMETRIC

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ir	r-component of the intensity vector (frequency domain only)
phys_id.Iz	z-component of the intensity vector (frequency domain only)
phys_id.Ii_mag	Magnitude of the instantaneous intensity vector (time domain only)
phys_id.Iir	r-component of the instantaneous intensity vector (time domain only)
phys_id.Iiz	z-component of the instantaneous intensity vector (time domain only)

TABLE 4-3: INTENSITY VARIABLES IN 2D

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ix	x-component of the intensity vector (frequency domain only)
phys_id.Iy	y-component of the intensity vector (frequency domain only)
phys_id.Ii_mag	Magnitude of the instantaneous intensity vector (time domain only)
phys_id.Iix	x-component of the instantaneous intensity vector (time domain only)
phys_id.Iiy	y-component of the instantaneous intensity vector (time domain only)

Theory Background for the Aeroacoustics Branch

The scientific field of *aeroacoustics* deals with the interaction between a background mean flow and an acoustic field propagating in this flow. In general, this concerns both the very complex description of the creation of sound by turbulence in the background flow, that is, flow induced noise and the influence the background mean flow has on the propagation of an externally created sound field, that is, flow borne noise or flow borne sound. The *computational aeroacoustics* (CAA) capabilities of the aeroacoustics interfaces in COMSOL Multiphysics only cover the flow borne noise/sound situation.

Aeroacoustic simulations would ideally involve solving the fully compressible continuity, momentum (Navier-Stokes equations), and energy equations in the time domain. The acoustic pressure waves would then form a subset of the fluid solution. This approach is often impractical for real-world *computational aeroacoustics* (CAA) applications due to the required computational time and memory resources. Instead, for solving many practical engineering problems, a decoupled two-step approach is used: first solve for the fluid flow, then the acoustic perturbations of the flow.

For the solving the acoustic problem, the governing equations are linearized around the background mean flow and only solved for the acoustic perturbation. Acoustic variables are assumed to be small and perturbation theory can be used, for example, the total pressure

$$p_{\text{tot}} = p_0 + p$$

is the sum of the background mean pressure p_0 and the acoustic pressure variations p (sometimes labeled p' or p_1).

This section presents the basic mathematical framework for the aeroacoustic equations solved in the aeroacoustic interfaces, starting with the general governing equations for fluid flow, that is, conservation equations, constitutive equations, and equations of state. Then the linearized potential flow equations, the equations for the compressional potential flow, the linearized Euler equations, and finally the linearized Navier-Stokes equations are presented.

In this section the theory background for:

- General Governing Equations
- Linearized Potential Flow
- Compressible Potential Flow
- Linearized Euler
- Linearized Navier-Stokes
- Scattered Field Formulation for LE and LNS

General Governing Equations

The equations governing the physics in any fluid are the foundation for deriving the linearized aeroacoustic equations and in general any acoustics equations. The governing equations for the motion of a compressible fluid are the continuity equation (mass conservation), the Navier-Stokes equation (momentum conservation), and the general heat transfer equation (energy conservation). In order to close the system of equations, constitutive equations are needed, along with the equation of state, and thermodynamic relations. See, for example, Ref. 3, Ref. 4, Ref. 5, Ref. 6, or Ref. 7 for details and further reading.

CONSERVATION EQUATIONS

The conservation equations are for mass, momentum, and energy:

$$\begin{aligned} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= M \\ \rho \frac{D\mathbf{u}}{Dt} = \rho\left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u}\right) &= \nabla \cdot \boldsymbol{\sigma} + \mathbf{F} \\ \rho T \frac{Ds}{Dt} = \rho T\left(\frac{\partial s}{\partial t} + (\mathbf{u} \cdot \nabla)s\right) &= -\nabla \cdot \mathbf{q} + \Phi + Q \end{aligned} \tag{4-5}$$

where $\mathbf{u}$ is the velocity field, ρ is the density, T is the temperature, s is the specific entropy, $\boldsymbol{\sigma}$ is the stress tensor, $\mathbf{q}$ is the local heat flux, Φ is the viscous dissipation function, and M, $\mathbf{F}$, and Q are source terms. The operator D/Dt is the material derivative (or advection operator) defined as

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + (\mathbf{u} \cdot \nabla)$$

THERMODYNAMIC RELATIONS

Some thermodynamic relations are necessary when reformulating the energy equations in terms of other sets of thermodynamic variables, like (p, T) or (ρ, p). They are the

density differential, the specific energy relation, a relation due to Helmholtz, and the fundamental entropy relation:

$$\frac{d\rho}{\rho} = -\alpha_p dT + \beta_T dp$$

$$du = \left(\frac{\partial u}{\partial T}\right)_\rho + \left(\frac{\partial u}{\partial \rho}\right)_T$$

$$\left(\frac{\partial u}{\partial \rho}\right)_T = \frac{1}{\rho^2}\left(p - \frac{\alpha_p}{\beta_T}T\right)$$

$$ds = \frac{1}{T}du + \frac{p}{T}d(\rho^{-1})$$

where u is the specific internal energy, α_p is the coefficient of thermal expansion (isobaric), and β_T is the isothermal compressibility. See, for example, Ref. 5 and Ref. 6 for details. They are defined together with the specific heat at constant pressure C_p and specific heat at constant volume C_v as

$$\alpha_p = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p \qquad \beta_T = \frac{1}{\rho}\left(\frac{\partial \rho}{\partial p}\right)_T$$

$$C_v = \left(\frac{\partial u}{\partial T}\right)_\rho \qquad C_p = \left(\frac{\partial u}{\partial T}\right)_p$$

Using the above thermodynamic relations, the entropy differential can be expressed as (used for the linearized Navier-Stokes equations)

$$\rho T ds = \rho C_p dT - \alpha_p T dp$$

while for an ideal gas it can be given as (used for the linearized Euler equations)

$$ds = C_v \frac{dp}{\rho} - C_p \frac{d\rho}{p}$$

CONSTITUTIVE EQUATIONS

The constitutive equations are the equations of state (density expressed in terms of any set of thermodynamic variables), the Stokes expression for the stress tensor, and the Fourier heat conduction law

$$\boldsymbol{\sigma} = -p\mathbf{I} + \boldsymbol{\tau} = -p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) + \left(\mu_B - \frac{2}{3}\mu\right)(\nabla \cdot \mathbf{u})\mathbf{I}$$

$$\mathbf{q} = -k\nabla T$$

where k is the thermal conduction, μ is the dynamic viscosity, and μ_B is the bulk viscosity. This then also defines the viscous dissipation function

$$\Phi = \nabla \mathbf{u} : \boldsymbol{\tau}$$

PERTURBATION THEORY

In the following, the governing equations are linearized and expanded to first order in the small parameters around the average stationary background solution. For details about perturbation theory see Ref. 3, Ref. 4, and Ref. 8. The small parameter variables (1'st order) represent the acoustic variations on top of the stationary background mean (or average) flow (0'th order solution). Note that, when solving the equations, the value of the acoustic field variables can also represent non-acoustic waves like thermal waves (entropy waves) and vorticity waves. In the time domain, these can be linear instabilities and can actually represent the onset of turbulence.

The dependent variables and sources are expanded according to

$$A = A_0(\mathbf{x}) + A_1(\mathbf{x}, t)$$

where A is any of the dependent variables or sources. In the frequency domain, the first order variables are assumed to be harmonic and expanded into Fourier components, such that

$$A = A_0(\mathbf{x}) + A_1(\mathbf{x}) e^{i\omega t}$$

The first order variation to material parameters, that are not treated as dependent variables, like, for example, the density ρ in the Linearized Navier-Stokes interface, are linearized using a Taylor expansion

$$\rho - \rho_0 = (p - p_0)\frac{\partial \rho_0}{\partial p}\bigg|_T + (T - T_0)\frac{\partial \rho_0}{\partial T}\bigg|_p$$

The above expansions are inserted into the governing equations and the linearized acoustic equations are derived retaining only first order linear terms.

Linearized Potential Flow

The equations presented here are the linearized potential flow equations. This restricts the applications of the physics interface to systems where the background flow is well described by a compressible potential flow, that is, a flow that is inviscid, barotropic, and irrotational. The sound sources also need to be external to the flow or at least they

need to be represented by simple well defined sources. Application areas typically include modeling of how jet engine noise is influenced by the mean flow.

The basic dependent variable is the velocity potential ϕ conventionally defined by the relationship

$$\mathbf{v} = \nabla\phi$$

where $\mathbf{v} = \mathbf{v}(\mathbf{r}, t)$ is the particle velocity associated with the acoustic wave motion. The *total* particle velocity is given by

$$\mathbf{v}_{\text{tot}}(\mathbf{r}, \mathbf{t}) = \mathbf{V}(\mathbf{x}) + \mathbf{v}(\mathbf{r}, t) \tag{4-6}$$

where $\mathbf{V}$ denotes the local mean velocity for the fluid motion. The dynamic equations for this mean-flow field are described in the next subsection. For now, just assume $\mathbf{V}$ to be a given *irrotational* background velocity field; hence, also the mean-flow velocity can be defined in terms of a potential field Φ, by $\mathbf{V} = \nabla\Phi$.

The linearized equation for the velocity potential ϕ, governing acoustic waves in a background flow with mean background velocity $\mathbf{V}$, mean background density ρ_0, and mean background speed of sound c_0, is

$$-\frac{\rho_0}{c_0^2}\frac{\partial}{\partial t}\left(\frac{\partial\phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right) + \nabla\cdot\left[\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right)\mathbf{V}\right] = 0 \tag{4-7}$$

In deriving this equation, all variables appearing in the full nonlinear fluid-dynamics equations were first split in time-independent and acoustic parts, in the manner of Equation 4-6. Then, linearizing the resulting equations in the acoustic perturbation and eliminating all acoustic variables except the velocity potential gives Equation 4-7. Thus, the density ρ in this equation is the time-independent part. The corresponding acoustic part is $\rho(\mathbf{r}, t) = p(\mathbf{r}, t)/{c_0}^2$ where p is the acoustic pressure, given by

$$p(\mathbf{r}, t) = -\rho_0\left(\frac{\partial\phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right)$$

Hence, once Equation 4-7 has been solved for the velocity potential, the acoustic pressure can easily be calculated.

When transformed to the frequency domain, the wave Equation 4-7 reads

$$-\frac{\rho_0}{c_0^2}i\omega(i\omega\phi + \mathbf{V}\cdot\nabla\phi) + \nabla\cdot\left[\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\mathbf{V}\right] = 0$$

while the acoustic pressure is

$$p(\mathbf{r}) = -\rho_0(i\omega\phi + \mathbf{V}\cdot\nabla\phi)$$

Typical boundary conditions include:

- Sound-hard boundaries or walls
- Sound-soft boundaries
- Impedance boundary conditions
- Radiation boundary conditions

FREQUENCY DOMAIN EQUATIONS

In the frequency domain the velocity potential ϕ is assumed to be a harmonic wave of the form

$$\phi(\mathbf{r}, t) = \phi(\mathbf{r})e^{i\omega t}$$

The governing frequency domain — or time-harmonic — equation is

$$-\frac{\rho_0}{c_0^2}i\omega(i\omega\phi + \mathbf{V}\cdot\nabla\phi) + \nabla\cdot\left[\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\mathbf{V}\right] = 0$$

In 2D, where

$$\phi(\mathbf{r}, t) = \phi(x, y)e^{i(\omega t - k_z z)}$$

the out-of-plane wave number k_z enters the equations when the ∇ operators are expanded:

$$-i\omega\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi - ik_z V_z\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi - ik_z V_z\phi)\right)$$

$$+ \rho_0 k_z^2\phi + ik_z V_z\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi - ik_z V_z\phi) = 0$$

The default value of the out-of-plane wave number is 0, that is, no wave propagation perpendicular to the 2D plane. In a mode analysis the equations are solved for k_z.

For 2D axisymmetric components

$$\phi(\mathbf{r}, t) = \phi(r, z)e^{i(\omega t - m\varphi)}$$

the circumferential wave number m similarly appears in the equation as a parameter:

$$-i\omega\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{V}\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\right) + \rho_0\frac{m^2}{r^2}\phi = 0$$

The background velocity field $\mathbf{V}$ cannot have a circumferential component because the flow is irrotational.

TIME-DEPENDENT EQUATION

In the time domain, the physics interface solves for the velocity potential ϕ with an arbitrary transient dependency. The following equation governs the acoustic waves in a mean potential flow:

$$-\frac{\rho_0}{c_0^2}\frac{\partial}{\partial t}\left(\frac{\partial\phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right) + \nabla\cdot\left[\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right)\mathbf{V}\right] = 0 \tag{4-8}$$

Here ρ_0 (SI unit: kg/m^3) is the background mean flow density, $\mathbf{V}$ (SI unit: m/s) denotes the background mean velocity, and c_0 (SI unit: m/s) refers to the speed of sound. The software solves the equation for the velocity potential ϕ, with SI unit m^2/s. The validity of this equation relies on the assumption that ρ_0, $\mathbf{V}$, and c_0 are approximately constant in time, while they can be functions of the spatial coordinates.

The background velocity field $\mathbf{V}$ cannot have a circumferential component because the flow is irrotational.

BOUNDARY MODE ANALYSIS

The boundary mode analysis type in 3D uses the eigenvalue solver to solve the equation

$$-i\omega\frac{\rho_0}{c_0^2}(i\omega\phi + (\mathbf{V}_t\cdot\nabla\phi) - ik_zV_n\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{V}_t\frac{\rho_0}{c_0^2}(i\omega\phi + (\mathbf{V}_t\cdot\nabla\phi) - ik_zV_n\phi)\right) + \rho_0k_z^2\phi + ik_zV_n\frac{\rho_0}{c_0^2}(i\omega\phi + (\mathbf{V}\cdot\nabla\phi) - ik_zV_n\phi) = 0 \tag{4-9}$$

for the eigenmodes, ϕ, and eigenvalues, $\lambda = -ik_z$, on a bounded two-dimensional domain, Ω, given well-posed edge conditions on $\partial\Omega$. In this equation, ϕ is the velocity

potential, ρ_0 is the background mean flow density, c_0 is the speed of sound, ω is the angular frequency, and k_z is the out-of-plane wave number or propagation constant. Furthermore, $\mathbf{V}_t$ denotes the background mean velocity in the tangential plane while V_n is the background mean velocity component in the normal direction.

Although the out-of-plane wave number is called k_z, the two-dimensional surface on which Equation 4-9 is defined does not necessarily have to be normal to the z-axis for 3D geometries.

Compressible Potential Flow

Consider a compressible and inviscid fluid in some domain Ω. The motion and state of the fluid is described by its velocity $\mathbf{V}$, density ρ, pressure p, and total energy per unit volume e. Its dynamics is governed by the Euler equations, expressing the conservation of mass, momentum, and energy:

$$\begin{aligned} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) &= 0 \\ \rho\left(\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla)\mathbf{V}\right) + \nabla p &= \mathbf{f} \\ \frac{\partial e}{\partial t} + \nabla \cdot ((e + p)\mathbf{V}) &= 0 \end{aligned} \tag{4-10}$$

Here a volume force $\mathbf{f}$ has been included on the right-hand side of the momentum equation, whereas a possible heat-source term on the right-hand side of the energy equation (the last one) has been set to zero.

To close this system of five equations with six unknowns, an equation of state is required. Here, this is taken to be the equation for an ideal barotropic fluid,

$$p = p_{\text{ref}}\left(\frac{\rho}{\rho_{\text{ref}}}\right)^{\gamma}$$

where $\gamma = C_p/C_V$ is the ratio between the specific heats at constant pressure and constant volume, while p_{ref} and ρ_{ref} are reference quantities for the pressure and the density, respectively, valid at some point in space. An alternative form of the ideal-fluid state equation is

$$p = \rho(\gamma - 1)e$$

The assumption that the fluid is barotropic means that $p = p(\rho)$. Taking the total time derivative and using the chain rule, leads to the relation

$$\frac{dp}{dt} = \frac{dp}{d\rho}\frac{d\rho}{dt} \equiv c^2 \frac{d\rho}{dt}$$

where, using the equation of state,

$$c = \sqrt{\gamma \frac{p}{\rho}}$$

defines the speed of sound in the ideal fluid.

Assuming the flow to be irrotational, there exists a *velocity potential* field Φ, such that $\mathbf{V} = \nabla\Phi$. If, in addition, the volume force is assumed to be given by $\mathbf{f} = -\rho\nabla\Psi$, where Ψ is referred to as the force potential, the second of Equation 4-10 can be integrated to yield the *Bernoulli equation*

$$\frac{\partial\Phi}{\partial t} + \left(\frac{1}{2}|\nabla\Phi|^2 + \frac{\gamma}{\gamma-1}\frac{p_{\text{ref}}}{\rho}\left(\frac{\rho}{\rho_{\text{ref}}}\right)^{\gamma} + \Psi\right) = \frac{1}{2}{v_{\text{ref}}}^2 + \frac{\gamma}{(\gamma-1)}\frac{p_{\text{ref}}}{\rho_{\text{ref}}} + \Psi_{\text{ref}}$$

In this equation, two additional reference quantities have entered: the velocity v_{ref}, and the force potential Ψ_{ref}, both valid at the same reference point as p_{ref} and ρ_{ref}. Note, in particular, that neither the pressure p, nor the energy per unit volume e, appears in this equation.

TIME DEPENDENT STUDY

Collecting the results, the equations governing the compressible, inviscid, irrotational flow of an ideal fluid are

$$\frac{\partial\Phi}{\partial t} + \left(\frac{1}{2}|\nabla\Phi|^2 + \frac{\gamma}{\gamma-1}\frac{p_{\text{ref}}}{\rho}\left(\frac{\rho}{\rho_{\text{ref}}}\right)^{\gamma} + \Psi\right) = \frac{1}{2}{v_{\text{ref}}}^2 + \frac{\gamma}{(\gamma-1)}\frac{p_{\text{ref}}}{\rho_{\text{ref}}} + \Psi_{\text{ref}}$$

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\nabla\Phi) = 0 \qquad c = \sqrt{\gamma\frac{p}{\rho}} \qquad \gamma \equiv C_p/C_V$$

where γ is the specific-heat ratio C_p/C_V and Ψ denotes the force potential, that is, the potential energy per unit mass measured in J/kg. In this equation, subscript **ref** signifies reference quantities that apply at a specific point or surface. Thus, p_{ref} is a reference pressure, ρ_{ref} is a reference density, v_{ref} is a reference velocity, and Ψ_{ref} is a reference force potential.

STATIONARY STUDY

In a stationary study, the same equation is used, but all time derivatives are set to zero, such that:

$$\left(\frac{1}{2}|\nabla\Phi|^2 + \frac{\gamma}{\gamma-1}\frac{p_{\mathrm{ref}}}{\rho}\left(\frac{\rho}{\rho_{\mathrm{ref}}}\right)^{\gamma} + \Psi\right) = \frac{1}{2}{v_{\mathrm{ref}}}^2 + \frac{\gamma}{(\gamma-1)}\frac{p_{\mathrm{ref}}}{\rho_{\mathrm{ref}}} + \Psi_{\mathrm{ref}}$$

$$\nabla\cdot(\rho\nabla\Phi) = 0$$

Linearized Euler

The linearized Euler equations are derived from Euler's equations, that is Equation 4-5 with no thermal conduction and no viscous losses. The fluid in the linearized Euler physics interface is assumed to be an ideal gas. This is the common approach in literature. A review of the linearized Euler equations is found in, for example, Ref. 12 and Ref. 13.

GOVERNING EQUATIONS

A linearization of the governing equations yield after some manipulation

$$\begin{aligned}
\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}_0 + \rho_0\mathbf{u}) &= S_{\mathrm{c}} \\
\frac{\partial\mathbf{u}}{\partial t} + \nabla\cdot\left(\mathbf{u}\mathbf{u}_0^{\mathrm{T}} + \frac{p}{\rho_0}\mathbf{I}\right) + \frac{\rho}{\rho_0}(\mathbf{u}_0\cdot\nabla)\mathbf{u}_0 - p\nabla(\rho_0^{-1}) & \\
+ (\nabla\mathbf{u}_0 - (\nabla\cdot\mathbf{u}_0)\mathbf{I})\mathbf{u} &= S_{\mathrm{m}} \\
\frac{\partial p}{\partial t} + \nabla\cdot(\gamma p_0\mathbf{u} + p\mathbf{u}_0) + (1-\gamma)(\mathbf{u}\cdot\nabla)p_0 - (1-\gamma)(\nabla\cdot\mathbf{u}_0)p &= S_{\mathrm{e}}
\end{aligned} \tag{4-11}$$

Here, the subscript "1" has been dropped on the acoustic perturbation variables. The time derivatives are replaced with multiplication by $i\omega$ in the frequency domain. Some ideal gas relations of interest are the equation of state, specific heat capacity and specific gas constant relations, and the compressibility and coefficients of thermal expansion:

$$c_0^2 = \frac{\gamma p_0}{\rho_0} = R_s \gamma T_0$$

$$\gamma = \frac{C_p}{C_v} \qquad C_p = C_v + R_s$$

$$\alpha_p = \frac{1}{T_0} \qquad \beta_T = \frac{1}{p_0}$$

The specific heat capacity at constant volume and constant pressure are often labeled with a lower case c, here we use upper case C's but they are the specific quantities.

It also follows from the governing equations and the thermodynamic relations that the acoustic variations in the specific entropy s and in the temperature T are given by

$$s = \frac{C_v}{p_0} p - \frac{C_p}{\rho_0} \rho$$

$$T = T_0 \left(\frac{p}{p_0} - \frac{\rho}{\rho_0} \right)$$

INSTABILITIES

When the linearized Euler (LE) equations are solved in the time domain (or in the frequency domain with an iterative solver), linear physical instability waves can develop, the so-called Kelvin-Helmholtz instabilities. They are instabilities that grow exponentially because no losses exist in the LE equations (no viscous dissipation and no heat conduction). Furthermore, they are limited by non-linearities in the full Navier-Stokes flow equations. It has been shown that in certain cases the growth of these instabilities can be limited, while the acoustic solution is retained, by canceling terms involving gradients of the mean flow quantities. This is known as gradient terms suppression (GTS) stabilization. See more details in Ref. 9, Ref. 10, and Ref. 11.

THE ENERGY COROLLARY

Expressions for the energy flux, that is, the acoustic intensity vector, are often referred to as Myers' energy corollary, see Ref. 15 and Ref. 16. The instantaneous intensity vector $\mathbf{I}_i$ is defined for both transient and frequency domain models as

$$\mathbf{I}_i = (\rho_0 \mathbf{u} + \rho \mathbf{u}_0) \left(\frac{p}{\rho_0} + \mathbf{u}_0 \cdot \mathbf{u} \right) + \rho_0 \mathbf{u}_0 T s$$

The (time averaged) intensity vector $\mathbf{I}$ is given in the frequency domain by

$$\mathbf{I} = \frac{1}{2}\mathrm{Re}\left((\rho_0\mathbf{u} + \rho\mathbf{u}_0)\left(\frac{p}{\rho_0} + \mathbf{u}_0 \cdot \mathbf{u}\right)^* + \rho_0\mathbf{u}_0 Ts^*\right)$$

ABOUT THE IMPEDANCE BOUNDARY CONDITION

The Linearized Euler, Frequency Domain Interface and The Linearized Euler, Transient Interface have an Impedance and Interior Impedance physics feature and its theory is included here.

In the frequency domain Myers' equation (Ref. 1) gives an expression for the normal velocity at a boundary with a normal impedance condition. It is a so-called low-frequency approximation condition in the limit of very thin flow boundary layers (compared to the wavelength). Such conditions are used, for example, for porous lining conditions in ducts (Ref. 2). The condition is given by:

$$\mathbf{u} \cdot \mathbf{n} = \frac{p}{Z_\mathrm{n}} + \frac{1}{i\omega}\mathbf{u}_0 \cdot \nabla\left(\frac{p}{Z_\mathrm{n}}\right) - \frac{p}{i\omega Z_\mathrm{n}}\mathbf{n} \cdot (\mathbf{n} \cdot \nabla\mathbf{u}_0) \tag{4-12}$$

where the surface normal $\mathbf{n}$ here points out of the domain and is the surface normal impedance.

If the flow is parallel to the impedance boundary condition $\mathbf{u}_0 \cdot \mathbf{n} = 0$, for example, slip flow over a mechanical impedance boundary condition (the same is true for the moving wall boundary condition described below), one can use a formulation with the tangential derivative ($\nabla_{||}$) for the second term on the right-hand side:

$$\mathbf{u}_0 \cdot \nabla A = \mathbf{u}_0 \cdot \nabla_{||} A + \mathbf{u}_0 \cdot \mathbf{n}(\mathbf{n} \cdot \nabla A) = \mathbf{u}_0 \cdot \nabla_{||} A$$

where A is an arbitrary scalar.

The last term on the right-hand side of Equation 4-12 can be reformulated as follows:

$$\mathbf{n} \cdot (\mathbf{n} \cdot \nabla\mathbf{u}_0) = (\mathbf{n} \cdot \nabla)(\mathbf{n} \cdot \mathbf{u}_0) - \mathbf{u}_0 \cdot ((\mathbf{n} \cdot \nabla)\mathbf{n})$$

Again these terms reduce significantly for the case where $\mathbf{n} \cdot \mathbf{u}_0 = 0$. If the boundary does not have curvature (planar boundary) then it is equal to zero. If the boundary is planar and the impedance condition is used inside the flow, for example at an outflow condition, then it reduces to the normal gradient of the velocity normal to the surface.

ABOUT THE MOVING WALL BOUNDARY CONDITION

The Linearized Euler, Frequency Domain Interface and The Linearized Euler, Transient Interface have a Moving Wall physics feature and its theory is included here.

Myers' equation (Ref. 1) gives the expressions used for a boundary condition at a moving wall. In the frequency domain it is given by

$$\mathbf{u}\cdot\mathbf{n} = -i\omega v_{\mathrm{n}} - \mathbf{u}_0\cdot\nabla(v_{\mathrm{n}}) + v_{\mathrm{n}}\mathbf{n}\cdot(\mathbf{n}\cdot\nabla\mathbf{u}_0)$$

The inward normal displacement is given by

$$v_{\mathrm{n}} = -n\cdot v_{\mathrm{n}} = (i\omega)^{-1}u_n \qquad u_n = -n\cdot u_{\mathrm{b}}$$

where u_n is the inward normal velocity is.

In the time domain, the condition on the normal velocity is given by

$$\mathbf{u}\cdot\mathbf{n} = -\frac{\partial v_{\mathrm{n}}}{\partial t} - \mathbf{u}_0\cdot\nabla(v_{\mathrm{n}}) + v_{\mathrm{n}}\mathbf{n}\cdot(\mathbf{n}\cdot\nabla\mathbf{u}_0)$$

where the inward normal displacement is given by $v_n = -\mathbf{n}\cdot\mathbf{v}$ (SI unit: m).

Linearized Navier-Stokes

The linearized Navier-Stokes equations are derived by linearizing the full set of fluid flow equations given in General Governing Equations. After some manipulation, the continuity, momentum, and energy equations and are:

$$\begin{aligned}
&\frac{\partial p}{\partial t} + \nabla\cdot(\rho_0\mathbf{u} + \rho\mathbf{u}_0) = M\\
&\rho_0\left(\frac{\partial\mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u}_0 + (\mathbf{u}_0\cdot\nabla)\mathbf{u}\right) + \rho(\mathbf{u}_0\cdot\nabla)\mathbf{u}_0 = \nabla\cdot\boldsymbol{\sigma} + \mathbf{F}\\
&\rho_0 C_{\mathrm{p}}\left(\frac{\partial T}{\partial t} + \mathbf{u}\cdot\nabla T_0 + \mathbf{u}_0\cdot\nabla T\right) + \rho C_{\mathrm{p}}(\mathbf{u}_0\cdot\nabla T_0)\\
&-\alpha_{\mathrm{p}}T_0\left(\frac{\partial p}{\partial t} + \mathbf{u}\cdot\nabla p_0 + \mathbf{u}_0\cdot\nabla p\right) - \alpha_{\mathrm{p}}T(\mathbf{u}_0\cdot\nabla p_0) = \nabla\cdot(\mathrm{k}\nabla T) + \Phi + Q
\end{aligned} \tag{4-13}$$

where p, $\mathbf{u}$, and T are the acoustic perturbations to the pressure, velocity, and temperature, respectively. In the frequency domain, the time derivatives of the dependent variables are replaced by multiplication with $i\omega$. The stress tensor is $\boldsymbol{\sigma}$ and Φ is the viscous dissipation function. The variables with a zero subscript are the

background mean flow values and the subscript "1" is dropped on the acoustic variables.

The constitutive equations are the stress tensor and the linearized equation of state, while the Fourier heat conduction law is readily included in the above energy equation,

$$\boldsymbol{\sigma} = -p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) + \left(\mu_{\mathrm{B}} - \frac{2}{3}\mu\right)(\nabla\cdot\mathbf{u})\mathbf{I}$$

$$\rho = \rho_0(\beta_{\mathrm{T}}p - \alpha_{\mathrm{p}}T)$$

The linearized viscous dissipation function is defined as

$$\Phi = \nabla\mathbf{u}:\boldsymbol{\tau}(\mathbf{u}_0) + \nabla\mathbf{u}_0:\boldsymbol{\tau}(\mathbf{u})$$

$$\boldsymbol{\tau}(\mathbf{u}) = \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) + \left(\mu_{\mathrm{B}} - \frac{2}{3}\mu\right)(\nabla\cdot\mathbf{u})\mathbf{I}$$

$$\boldsymbol{\tau}(\mathbf{u}_0) = \mu(\nabla\mathbf{u}_0 + (\nabla\mathbf{u}_0)^{\mathrm{T}}) + \left(\mu_{\mathrm{B}} - \frac{2}{3}\mu\right)(\nabla\cdot\mathbf{u}_0)\mathbf{I}$$

The terms in the governing equations presented in Equation 4-13 can be divided into four categories. The time derivative (or frequency dependent) term, convective terms like $\mathbf{u}_0\cdot\nabla p$, reactive terms like $\mathbf{u}\cdot\nabla p_0$, diffusive terms, and source terms. In many aeroacoustic formulations the reactive terms are removed (or simplified) from the governing equations in order to avoid the Kelvin-Helmholtz instabilities. This is sometimes referred to as gradient term stabilization (GTS).

Scattered Field Formulation for LE and LNS

The governing equations, given above in Equation 4-11 and Equation 4-13, are defined in the general scattered field formulation. Assuming that all the acoustic fields are the sum of a background (prescribed) field and the resulting scattered field, the total fields can be written as

$$p_{\mathrm{t}} = p_{\mathrm{s}} + p_{\mathrm{b}} \qquad \mathbf{u}_{\mathrm{t}} = \mathbf{u}_{\mathrm{s}} + \mathbf{u}_{\mathrm{b}} \qquad T_{\mathrm{t}} = T_{\mathrm{s}} + T_{\mathrm{b}} \qquad \rho_{\mathrm{t}} = \rho_{\mathrm{s}} + \rho_{\mathrm{b}}$$
$$p \equiv p_{\mathrm{s}} \qquad \mathbf{u} \equiv \mathbf{u}_{\mathrm{s}} \qquad T \equiv T_{\mathrm{s}} \qquad \rho \equiv \rho_{\mathrm{s}}$$

where subscript "t" stands for total, subscript "s" for scattered, and subscript "b" for background. The scattered field is the field solved for, that is the dependent variables p, $\mathbf{u}$, T, and ρ (depending on the interface). If no background acoustic field is defined (the default) the scattered field is equal to the total field. The governing equations and all boundary conditions are expressed in therms of the total fields. The background

acoustic fields are defined by adding the Background Acoustic Fields feature in the LE interface or the Background Acoustic Fields feature in the LNS interface. The features have a user defined option where all fields can be entered, either as analytical expressions or defined in terms of a solution to another study or model.

References for the Aeroacoustics Branch Interfaces

1. M. K. Myers, "On the basic boundary condition in the presence of flow", *J. of Sound and Vibration*, vol. 71, pp. 429, 1980.

2. W. Eversman, "The boundary condition at an impedance wall in a non-uniform duct with potential mean flow", *J. of Sound and Vibration*, vol. 246, pp. 63, 2001.

3. D.T. Blackstock, *Fundamentals of Physical Acoustics,* John Wiley & Sons, 2000.

4. H. Bruus, *Theoretical Microfluidics*, Oxford University Press, 2010.

5. G.K. Bachelor, *An Introduction to Fluid Dynamics*, Cambridge University Press, 2000.

6. L. D. Landau and E. M. Lifshitz, *Fluid Mechanics, Course on Theoretical Physics volume 6*, Butterworth-Heinemann, 2003.

7. B. Lautrup, *Physics of Continuous Matter, Exotic and Every Day Phenomena in the Macroscopic World,* 2nd ed., CRC Press, 2011.

8. C. M. Bender and S. A. Orszag, *Advanced Mathematical Methods for Scientists and Engineers: Asymptotic Methods and Perturbation Theory*, Springer, 1999.

9. C. Bogey, C. Bailly, and D. Juvé, "Computation of Flow Using Source Terms in Linearized Euler's Equations", *AIAA Journal*, vol. 40, pp. 235, 2002.

10. P. P. Rao and P. J. Morris, "Use of Finite Element Methods in Frequency Domain Aeroacoustics", *AIAA Journal*, vol. 44, pp. 1643, 2006.

11. A. Agarwal, P. J. Morris, and R. Mani, "Calculation of Sound Propagation in Nonuniform Flows: Suppression of Instability Waves", *AIAA Journal*, vol. 42, pp.80, 2004.

12. C. Bailly and D. Juvé, "Numerical Solution of Acoustic Propagation Problems Using Linearized Euler Equations", *AIAA Journal*, vol. 38, pp. 22, 2000.

13. C. W. Tam, "Computational Aeroacoustics: Issues and Methods", *AIAA Journal*, vol. 33, 1995.

14. G. Hauke and T. J. R. Hughers, "A comparative study of different sets of variables for solving compressible and incompressible flows," *Comput. Methods Appl. Mech. Engrg.*, vol. 153, pp. 1–44, 1998.

15. M. K. Myers, "An Exact Energy Corollary for Homentropic Flows," *J. Sound. Vib.*, vol. 109, pp. 277–284, 1986.

16. M. K. Myers, "Transport of energy disturbances in arbitrary steady flows," *J. Fluid Mech.*, vol. 226, pp. 383–400, 1991.

17. C. K. W. Tam and Z. Dong, "Radiation and Outflow Boundary Conditions for Direct Computation of Acoustic and Flow Disturbances in Nonuniform Mean Flow," J. *of Comp. Acoustics* 4, pp. 175–201, 1996.

18. L. Du, A. Holmberg, M. Karlsson, and M Åbom, "Sound amplification at a rectangular T-junction with merging mean flows," *J. Sound Vib.*, vol. 367, pp. 69-83 2016.

19. J. Gikadi, S. Föller, T. Sattelmayer, "Impact of turbulence on the prediction of linear aerodynamic interactions: Acoustic response of a turbulent shear layer," *J. Sound Vib.*, vol. 333, pp. 6548-6559, 2014.

20. S. Redonnet and G. Cunha, "An advanced hybrid method for the acoustic prediction," *Adv. Eng. Softw.*, vol. 88, pp. 30-52, 2015.

21. J. Gikadi, "Prediction of Acoustic Modes in Cumbustors using Linearized Navier-Stokes Equations in the Frequency Domain," Ph.D. thesis, TUM, 2013.

22. O. C. Zienkiewicz, R. L. Taylor, and P. Nithiarasu, "The Finite Element Method for Fluid Dynamics," Butterworth-Heinemann, 7th edition, 2014.

5

Thermoviscous Acoustics Interfaces

This chapter describes the physics interfaces found under the **Thermoviscous Acoustics** branch ( ).

The Thermoviscous Acoustics, Frequency Domain Interface

The **Thermoviscous Acoustics, Frequency Domain (ta)** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations of pressure, velocity, and temperature. This physics interface is required to accurately model acoustics in geometries of small dimensions. Near walls, viscous losses and thermal conduction become important because a boundary layer exists. The thicknesses of the boundary layers is known as the viscous and thermal penetration depth. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations. It is, for example, used when modeling the response of transducers like microphones, miniature loudspeakers and receivers. Other applications include analyzing feedback in hearing aids and in mobile devices, or studying the damped vibrations of MEMS structures.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. Linear acoustics is assumed.

The equations defined by the Thermoviscous Acoustics, Frequency Domain interface are the linearized Navier-Stokes equations in quiescent background conditions solving the continuity, momentum, and energy equations. Thermoviscous acoustics is also known as *viscothermal acoustics* or sometimes *thermoacoustics* (not to be confused with the field discussing heating and cooling using acoustics). Due to the detailed description necessary when modeling thermoviscous acoustics, the model simultaneously solves for the acoustic pressure p, the acoustic velocity variation $\mathbf{u}$ (particle velocity), and the acoustic temperature variations T. It is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

The Thermoviscous Acoustics, Frequency Domain interface is formulated in the so-called scattered field formulation where the total field (subscript t) is the sum of the scattered field (the field solved for, p, $\mathbf{u}$, and T) and a possible background acoustic field (subscript b), such that

$$p_\mathrm{t} = p + p_\mathrm{b} \qquad \mathbf{u}_\mathrm{t} = \mathbf{u} + \mathbf{u}_\mathrm{b} \qquad T_\mathrm{t} = T + T_\mathrm{b}$$

When no Background Acoustic Fields feature is present (the background field values are zero per default) the total field is simply the field solved for, $p_\mathrm{t} = p$, $\mathbf{u}_\mathrm{t} = \mathbf{u}$, and

$T_t = T$. All governing equations and boundary conditions are formulated in the total field variables.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Thermoviscous Acoustics Model**, **Wall**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Thermoviscous Acoustics** to select physics features from the context menu.

For good modeling strategies, solver suggestions, postprocessing information, as well as tips and tricks, see the Modeling with the Thermoviscous Acoustics Branch section.

On the Thermoviscous Acoustics Physics Interface

The Thermoviscous Acoustics, Frequency Domain interface solves, as mentioned, the full linearized Navier-Stokes (momentum), continuity, and energy equations. It solves for the propagation of compressible linear waves in a general viscous and thermally conductive fluid. The length scale at which the thermoviscous acoustic description is necessary is given by the thickness of the, above-mentioned, viscous boundary layer (the viscous penetration depth), which is

$$\delta_v = \sqrt{\frac{\mu}{\pi f \rho_0}}$$

and the thickness of the thermal boundary layer (the thermal penetration depth) is

$$\delta_t = \sqrt{\frac{k}{\pi f \rho_0 C_p}}$$

where the definition of the symbols f, μ, ρ_0, k, and C_p may be found in Table 5-1. The thickness of both boundary layers depends on the frequency f and decreases with increasing frequency. The ratio of the two length scales is related to the nondimensional Prandtl number Pr, by

$$\frac{\delta_v}{\delta_t} = \sqrt{\frac{\mu C_p}{k}} = \sqrt{\mathrm{Pr}}$$

which define the relative importance of the thermal and viscous effects for a given material. In air at 20 ^{o}C and 1 atm the viscous boundary layer thickness is 0.22 mm at

100 Hz while it is only 55 µm in water under the same conditions. The Prandtl number is 0.7 in air and 7 in water.

Evaluate the value of the viscous and thermal boundary layer thickness as well as the Prandtl number in postprocessing. They are defined by the variables `ta.d_visc`, `ta.d_therm`, and `ta.Pr`, respectively.

The physical quantities commonly used in the thermoviscous acoustics interfaces are defined in Table 5-1 below.

TABLE 5-1: THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN PHYSICAL QUANTITIES

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Pressure (acoustic)	p	pascal	Pa
Total acoustic pressure	p_t	pasca	Pa
Scattered acoustic pressure	p_s	pasca	Pa
Temperature variation (acoustic)	T	kelvin	K
Total temperature variation	T_t	kelvin	K
Scattered temperature variation	T_s	kelvin	K
Acoustic velocity field	$\mathbf{u} = (u, v, w)$	meter/second	m/s
Total acoustic velocity field	$\mathbf{u}_t$	meter/second	m/s
Scattered acoustic velocity field	$\mathbf{u}_s$	meter/second	m/s
Dynamic viscosity	μ	pascal-second	Pa·s
Bulk viscosity	μ_B	pascal-second	Pa·s
Thermal conductivity	k	watt/meter-kelvin	W/(m·K)
Heat capacity at constant pressure	C_p	joule/meter^3-kelvin	J/(m^3·K)
Isothermal compressibility	β_T	1/pascal	1/Pa
Coefficient of thermal expansion (isobaric)	α_p	1/kelvin	1/K
Ratio of specific heats	γ	(dimensionless)	1
Frequency	f	hertz	Hz
Wave number	k	1/meter	1/m
Equilibrium pressure	p_0	pascal	Pa

TABLE 5-1: THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN PHYSICAL QUANTITIES

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Equilibrium density	ρ_0	kilogram/meter^3	kg/m^3
Equilibrium temperature	T_0	kelvin	K
Speed of sound	c	meter/second	m/s
Acoustic impedance	Z	pascal-second/meter	Pa·s/m

As the thermoviscous acoustics physics interface solves for both pressure, velocity, and temperature, models can easily become large and contain many DOFs. See Solver Suggestions for Large Thermoviscous Acoustics Models for suggestions on how to solve large thermoviscous acoustic models.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `ta`.

EQUATION

Expand the **Equation** section to see the equations solved for with the **Equation form** specified. The default selection for **Equation form** is set to **Study controlled**. The available studies are selected under **Show equations assuming**.

- For **Study controlled**, the scaling of the equations is optimized for the numerical performance of the different solvers and study types.
- For **Frequency domain** you can manually enter the scaling parameter Δ under the **Thermoviscous Acoustics Equation Settings** section.

THERMOVISCOUS ACOUSTICS EQUATION SETTINGS

For all component dimensions, and if required, click to expand the **Equation** section, then select **Frequency domain** as the **Equation form** and enter the settings as described below.

The default **Scaling factor** Δ is $1/(i\omega)$. This value corresponds to the equations for a Frequency Domain study when the equations are study controlled. To get the equations corresponding to an Eigenfrequency study, change the **Scaling factor** Δ to 1. Changing the scaling factor influences the coupling to other physics.

SOUND PRESSURE LEVEL SETTINGS

See the settings for Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface.

TYPICAL WAVE SPEED FOR PERFECTLY MATCHED LAYERS

Enter a value or expression for the typical wave speed for perfectly matched layers c_{ref} (SI unit: m/s). The default is 343 m/s.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. From the list select the element order and type (Lagrange or serendipity) for the **Pressure**, the **Velocity field**, and the **Temperature variation**, respectively. The default is **Linear** for the pressure and **Quadratic Lagrange** for the velocity and the temperature.

- For numerical stability reasons the element order for the pressure should one less than the element order for the velocity.
- In fluids where the thermal and viscous boundary layer thickness are of the same order of magnitude (where the Prandtl number Pr is of the order 1, like in air), it is recommended to use the same shape order for the temperature and the velocity. Both fields vary equally over the same length scale in the acoustic boundary layers near walls.

Choosing between Lagrange and Serendipity Shape Functions has influence on the number of DOFs solved for and on stability for distorted mesh.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Pressure** p, the **Velocity field** $\mathbf{u}$ and its components, and the **Temperature variation** T. The names can

be changed but the names of fields and dependent variables must be unique within a model.

- Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface
- Theory Background for the Thermoviscous Acoustics Branch

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- *Uniform Layer Waveguide*: Application Library path **Acoustics_Module/Verification_Examples/uniform_layer_waveguide**
- *Generic 711 Coupler—An Occluded Ear-Canal Simulator*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/generic_711_coupler**

Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface

The Thermoviscous Acoustics, Frequency Domain Interface has these domain, boundary, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

	The Continuity node with this physics interface is available as a pair boundary condition. This gives continuity in pressure, temperature variation, velocity, and in the flux on a pair boundary between thermoviscous acoustic domains.

Some of the following nodes are available from the **Mechanical** and **Thermal** submenus (listed in alphabetical order):

- Adiabatic
- Axial Symmetry
- Background Acoustic Fields
- Heat Flux
- Heat Source
- Initial Values
- Interior Normal Impedance
- Interior Temperature Variation
- Interior Velocity
- Interior Wall
- Isothermal
- Normal Impedance
- Normal Stress
- No Slip
- No Stress
- Periodic Condition[1]
- Pressure (Adiabatic)
- Slip
- Stress
- Symmetry
- Temperature Variation
- Thermoviscous Acoustics Model
- Velocity
- Wall

[1]Described for the Pressure Acoustics, Frequency Domain interface

Thermoviscous Acoustics Model

Use the **Thermoviscous Acoustics Model** node to define the model inputs (the background equilibrium temperature and pressure) and the material properties of the fluid (dynamic viscosity, bulk viscosity, thermal conductivity, heat capacity at constant pressure, and equilibrium density) necessary to model the propagation of acoustic compressible waves in a thermoviscous acoustic context. Extended inputs are available for the coefficient of thermal expansion and the compressibility, which enables modeling of any constitutive relation for the fluid.

MODEL INPUTS

This section contains field variables that appear as model inputs. The fields are always active as the equilibrium (background) temperature enters the governing energy equation explicitly. From the **Equilibrium temperature** T_0 (SI unit: K) list, select an existing temperature variable (from another physics interface) if available, or select **User defined** to define a different value or expression. The default is **User defined** and set to 293.15 K (that is, 20°C). From the **Equilibrium pressure** p_0 (SI unit: Pa) list, select an existing absolute pressure variable (from another physics interface) if available, or select **User defined** to define a different value or expression. The default is **User defined** and set to 1 atm.

THERMOVISCOUS ACOUSTICS MODEL

Define the material parameters of the fluid by selecting an **Equilibrium density** — **Ideal gas**, **From material**, or **User defined**.

- If **From material** is selected (the default) the equilibrium density, and its dependence on the equilibrium pressure p_0 and temperature T_0, is taken from the defined material. Make sure that the Thermal Expansion and Compressibility settings are correct.
- For **Ideal gas** also select the Gas constant type — select Specific gas constant R_s (SI unit: J/(kg·K) or Mean molar mass M_n (SI unit: kg/mol)
- For **User defined** enter a value or expression for the **Equilibrium density** $\rho_0(p_0, T_0)$ (SI unit: kg/m^3). The default is `ta.p0/(287[J/kg/K]*ta.T0)`, which is the ideal gas law.

The other thermoviscous acoustic model parameters defaults use values **From material.** For **User defined** enter another value or expression for:

- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Bulk viscosity** μ_B (SI unit: Pa·s). The bulk viscosity parameter describes the difference between the mechanical and thermodynamic pressures. It is associated with losses doe to expansion and compression. Its value is difficult to measure and typically require absorption experiments to be determined. Its numerical value is of the same order as the dynamic viscosity. See, for example, Ref. 8 for fluids and Ref. 9 for gases.
- **Thermal conductivity** k (SI unit: W/(m·K)).
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)). This is the specific heat capacity or heat capacity per unit mass.

THERMAL EXPANSION AND COMPRESSIBILITY

One of the main characteristics of an acoustic wave is that it is a compressional wave. In the detailed thermoviscous acoustic description, this property is closely related to the constitutive relation between the density, the pressure, and the temperature. This results in the important (linear) relation for the acoustic density variation

$$\rho_t = \rho_0(\beta_T p_t - \alpha_p T_t)$$

where ρ_t is the total density variation, p_t is the total acoustic pressure, T_t is the total acoustic temperature variations, β_T is the (isothermal) compressibility of the fluid, and α_p the (isobaric) coefficient of thermal expansion (sometimes named α_0). If this constitutive relation is not correct, then no waves propagate or possibly they propagate at an erroneous speed of sound. When the **From equilibrium density** option (the default) is selected for the coefficient of thermal expansion and the compressibility, both values are derived from the equilibrium density $\rho_0(p_0,T_0)$ using their defining relations

$$\beta_T = \frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial p_0}\right]_{T_0} \qquad \alpha_p = -\frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial T_0}\right]_{p_0}$$

If the equilibrium density ρ_0 is a user-defined constant value or the material model does not define both a pressure and temperature dependence for ρ_0, the coefficient of thermal expansion and the compressibility need to be set manually, or they evaluate to 0.

For most materials, selected from the material library, it is necessary to set the coefficient of thermal expansion and the compressibility using one of the nondefault options.

If the material is air, the **From equilibrium density** option works well as the equilibrium density $\rho_0 = \rho_0(p_0,T_0)$ is a function of both pressure and temperature.

For water the coefficient of thermal expansion is well defined as $\rho_0 = \rho_0(T_0)$, while the compressibility can easily be defined using the **From speed of sound** option.

The **Thermal Expansion and Compressibility** section displays if **From material** or **User defined** is selected as the **Equilibrium density** under **Thermoviscous Acoustics Model**.

Select an option from the **Coefficient of thermal expansion** α_p list — **From equilibrium density** (the default), **From material**, or **User defined**. For **User defined** enter a value for α_p (SI unit: $1/K = K^{-1}$).

Select an option from the **Isothermal compressibility** β_T lists — **From equilibrium density** (the default), **From speed of sound**, **From isentropic compressibility**, or **User defined**. For **User defined**, enter a value for β_T (SI unit: $1/Pa = Pa^{-1}$).

For each of the following, and based on the above selection, the default is taken **From material**. For **User defined** enter another value or expression in the text field.

- **Speed of sound** c (SI unit: m/s).
- **Ratio of specific heats** γ (dimensionless). The default is 1.
- **Isentropic compressibility** β_s (SI unit: $1/Pa = Pa^{-1}$).

See the Theory Background for the Thermoviscous Acoustics Branch section for a detailed description of the governing equations and the constitutive relations.

To visualize the dissipated energy due to viscosity and thermal conduction in postprocessing. Three postprocessing variables exist:

- The viscous power dissipation density `ta.diss_visc`.
- The thermal power dissipation density `ta.diss_therm`.
- The total thermo-viscous power dissipation density `ta.diss_tot`.

In certain cases it can be interesting not to include thermal conduction in the model and treat all processes as adiabatic (isentropic). This is, for example, relevant for fluids where the thermal boundary layer is much thinner than the viscous. Not solving for the temperature field T also saves some degrees of freedom (DOFs).

This is achieved by setting the **Isothermal compressibility** to **User defined** and here enter the adiabatic (isotropic) value β_s (remember that for fluids $\beta_s = \gamma \cdot \beta_T$). Then, in the solver sequence under **Solver Configuration>Solver 1> Dependent Variables** select **Define by study step** to **User defined** and under **>Temperature variation (mod1.T)** click to clear the **Solver for this field** box.

See also Solver Suggestions for Large Thermoviscous Acoustics Models for suggestions on how to set up the solver for large problems.

Background Acoustic Fields

When the **Background Acoustic Fields** feature is added to a domain it is possible to define the value of the background acoustic field variables p_b, $\mathbf{u}_b$, and T_b. Using this feature it is possible to set up scattering problems as well as defining acoustic fields at an inlet of a waveguide (using a small domain at the inlet).

MODEL INPUTS

This section contains field variables that appear as model inputs. The fields are always active as the equilibrium (background) temperature T_0 enters the governing energy equation explicitly. It is used for the **Plane wave** option.

BACKGROUND ACOUSTIC FIELDS

Select the **Acoustic field type** — **User defined** (the default) or **Plane wave**.

When **User defined** is selected enter expressions for the **Background acoustic pressure** p_b, the **Background acoustic velocity** $\mathbf{u}_b$, and the **Background temperature variation** T_b. These can be analytical expressions or values of dependent variables solved in a previous study that defines the background field.

When **Plane wave** is selected enter the **Pressure Amplitude** $|p_b|$ (given at $\mathbf{x} = \mathbf{0}$) and the wave direction vector $\mathbf{e}_k$ (this vector is automatically normalized). Finally, select where the **Material data** should be taken from, the default is **From material model** (the same as the domain model material) or select a material from the list. This will set up the

background pressure, velocity, and temperature variations for a plane wave with the correct thermo-viscous attenuation compatible with the governing equations.

	By using the The Thermoviscous Acoustics, Boundary Mode Interface it is possible to define sources and ports at the inlet of waveguides. In combination with the Background Acoustic Fields it is possible to set up an incident field at the inlet of a waveguide. Define the background fields in a small domain at the inlet backed by a PML.

	A model showcasing the plane wave background field option: *Transfer Impedance of a Perforate*. Application Library path **Acoustics_Module/Tutorials/transfer_impedance_perforate**

Wall

Use the **Wall** node to model the most common conditions at solid surfaces. This is the default boundary condition. This condition contains both a mechanical and a thermal selection. The default is a no-slip and isothermal condition applicable in most cases. More advanced conditions at boundaries can be set-up by combining any of the Mechanical and Thermal conditions available.

The no-slip condition is the origin of the viscous boundary layer and the isothermal condition is the origin of the thermal boundary layer. It is within these acoustic boundary layers that the main dissipation happens. See the Theory Background for the Thermoviscous Acoustics Branch section for more details.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip**. See the No Slip and Slip conditions for further details.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. If **Slip** is selected for the **Mechanical condition** and **Use weak constraints** is enabled then

the default discontinuous Galerkin (penalty like) formulation is switched to a Lagrange multiplier formulation instead. See the Slip condition for details. For the **No-slip** and **Isothermal** selections a weak formulation is used instead of a point-wise constraint.

Initial Values

The **Initial Values** node adds initial values for the sound pressure, velocity field, and temperature variation. If necessary add more **Initial Values** nodes from the **Physics** toolbar to give initial values in, for example, other domains.

INITIAL VALUES

Enter values or expressions for the **Pressure** p (SI unit: Pa) (the default is 0 Pa), **Velocity field** $\mathbf{u}$ (SI unit: m/s) (the defaults are 0 m/s), and **Temperature variation** T (SI unit: K) (the default is 0 K).

Axial Symmetry

In a 2D axisymmetric geometry the **Axial Symmetry** condition is automatically added on the axis of symmetry at $r = 0$. On this boundary the radial component of the total velocity is set equal to zero

$$u_t = 0 \qquad \mathbf{u}_t = (u_t, v_t, w_t)$$

Interior Wall

The **Interior Wall** boundary condition is used to model a wall condition on an interior boundary. It is similar to the Wall boundary condition available on exterior boundaries except that it applies on both sides (up and down) of an internal boundary. It allows discontinuities (in pressure, velocity, and temperature) across the boundary. Use the Interior Wall boundary condition to avoid meshing thin structures by applying this slip-like conditions on interior curves and surfaces instead.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip**. See the No Slip and Slip conditions for further details.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

If **Slip** is selected for the **Mechanical condition** and **Use weak constraints** is enabled the default discontinuous Galerkin (penalty like) formulation is switched to a Lagrange multiplier formulation instead. For the **No-slip** and **Isothermal** selections a weak formulation is used instead of a point-wise constraint.

Interior Normal Impedance

Use the **Interior Normal Impedance** node to apply a transfer impedance condition on an interior boundary. The condition relates the normal stress at the two sides of the boundary (up and down) to the normal velocity via the normal transfer impedance Z_n. The condition allows discontinuities (a slit) in pressure and temperature depending on the condition chosen for the temperature.

$$\left(\left[-p_t\mathbf{I}+\mu(\nabla\mathbf{u}_t+(\nabla\mathbf{u}_t)^T)-\left(\frac{2\mu}{3}-\mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]_{up}\right.$$
$$\left.-\left[-p_t\mathbf{I}+\mu(\nabla\mathbf{u}_t+(\nabla\mathbf{u}_t)^T)-\left(\frac{2\mu}{3}-\mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]_{down}\right)\mathbf{n}=-Z_n(\mathbf{u}_t\cdot\mathbf{n})\mathbf{n}$$

INTERIOR NORMAL IMPEDANCE

Enter a value or expression for the **Normal impedance** Z_n (SI unit: Pa·s/m).

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Interior Velocity

Use the **Interior Velocity** node to specify a velocity on an interior boundary in thermoviscous acoustics. The condition can be used to specify sources, for example, the velocity of a diaphragm in a miniature transducer that is modeled using a lumped circuit model. The velocity components can be prescribed independently, and there is an option to force continuity in pressure across the boundary, as well as options for the thermal conditions.

MECHANICAL

	To define a prescribed velocity for each space direction (x and y, plus z for 3D), select one or more of the **Prescribed in x direction**, **Prescribed in y direction**, and **Prescribed in z direction** check boxes. Then enter a value or expression for the prescribed velocities u_0, v_0, or w_0 (SI unit: m/s).

	To define a prescribed velocity for each space direction (r and z), select one or both of the **Prescribed in r direction** and **Prescribed in z direction** check boxes. Then enter a value or expression for the prescribed velocities u_0, or w_0 (SI unit: m/s).

Select the **Continuous pressure** check box if the velocity condition is to be used as a source where the pressure should not be slit.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Interior Temperature Variation

Use the **Interior Temperature Variation** condition to model a thermal type of source on an interior boundaries, $T = T_{\text{bnd}}$. This can, for example, be a thin sheet that has a harmonically varying temperature, given by T_{bnd}, for example, due to Ohmic heating. This temperature fluctuation will give rise to acoustic waves that can be modeled using the Thermoviscous Acoustics interface and this boundary condition.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip**. See the No Slip and Slip conditions for further details.

THERMAL

Enter a value or expression for the **Temperature variation** T_{bnd} (SI unit: K) that has to be applied at the interior boundary.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

No Slip

The **No Slip** node sets up a no-slip condition for the flow on a hard wall:

$$\mathbf{u}_t = \mathbf{0}$$

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

	Mechanical and thermal boundary conditions contribute such that a condition can be set on the velocity and temperature simultaneously (they do not override). Thermal type condition override each other and the same is true for mechanical type conditions.

Isothermal

Use the **Isothermal** node to model a wall that is assumed to be a good thermal conductor and backed by a large heat reservoir kept at constant temperature. This implies that the harmonic temperature variations vanish: $T_t = 0$.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Heat Source

Use the **Heat Source** node to define the heat source for the thermoviscous acoustics model. This adds a domain heat source Q to the right-hand side of the energy equation.

HEAT SOURCE

Enter a value for the **Heat source** Q (SI unit: $\mathrm{W/m^3}$).

Pressure (Adiabatic)

Use the **Pressure (Adiabatic)** node to specify a prescribed pressure p_{bnd}, that acts as a pressure source at the boundary, typically an inlet or outlet. In the frequency domain

p_{bnd} is the amplitude of a harmonic pressure source. The adiabatic condition states that no heat flows into or out of the boundary:

$$p_t = p_{bnd}$$

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -p_{bnd}\mathbf{n}$$

$$-\mathbf{n}\cdot(-k\nabla T_t) = 0$$

This condition is in general not physically correct on a solid wall because solids are generally better thermal conductors than air.

PRESSURE

Enter the value of the **Pressure** p_{bnd} (SI unit: Pa) at the boundary. The default is 0 Pa.

Symmetry

The **Symmetry** node for The Thermoviscous Acoustics, Frequency Domain Interface adds a boundary condition that represents symmetry. It corresponds to the Slip condition for the mechanical degrees of freedom and the Adiabatic condition for the temperature variation.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Velocity

Use the **Velocity** node to define the prescribed velocities $\mathbf{u}_0$ on the boundary: $\mathbf{u}_t = \mathbf{u}_0$. This condition is useful, for example, when modeling a vibrating wall.

VELOCITY

	To define a prescribed velocity for each space direction (x and y, plus z for 3D), select one or more of the **Prescribed in x direction**, **Prescribed in y direction**, and **Prescribed in z direction** check boxes. Then enter a value or expression for the prescribed velocities u_0, v_0, or w_0 (SI unit: m/s).

To define a prescribed velocity for each space direction (r and z), select one or both of the **Prescribed in r direction** and **Prescribed in z direction** check boxes. Then enter a value or expression for the prescribed velocities u_0, or w_0 (SI unit: m/s).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Slip

Use the **Slip** node to prescribe zero normal velocity, and therefor also zero tangential stress, condition on the boundary

$$\mathbf{n} \cdot \mathbf{u}_t = \mathbf{0}$$

$$\boldsymbol{\sigma}_n - (\boldsymbol{\sigma}_n \cdot \mathbf{n})\mathbf{n} = \mathbf{0}$$

$$\boldsymbol{\sigma}_n = \left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla \cdot \mathbf{u}_t)\mathbf{I}\right]\mathbf{n}$$

This results in a so-called no-penetration condition where no viscous boundary layer is created. Use the condition in places where the viscous losses in the boundary layer are nonimportant. In this way it is not necessary to mesh the boundary layer resulting in fewer mesh elements.

Per default, the **Slip** condition uses a so-called discontinuous Galerkin or penalty formulation to prevent locking problems on curved surfaces.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Select **Use weak constraints** if you want to switch from the default discontinuous Galerkin (penalty like) formulation to a Lagrange multiplier formulation for the slip condition (this formulation is not suited for an iterative solver). The Lagrange multiplier formulation also prevents locking problems.

Stress

Use the **Stress** node to define the components of the resulting stresses vector $\boldsymbol{\sigma}$ on the boundary

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = \boldsymbol{\sigma}$$

Here the resulting stress σ represents the product of the stress tensor $\boldsymbol{\sigma}$ with the surface normal $\mathbf{n}$. These are the resulting local stresses in the three spatial directions.

STRESS

Enter the **Stress** σ (SI unit: $\mathrm{N/m^2}$) components for each space direction (x, y, and z or r and z for 2D axisymmetric components).

No Stress

Use the **No Stress** node to set the total surface stress equal to zero

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = \mathbf{0}$$

Normal Stress

Use the **Normal Stress** node to define the inward normal stress σ_n on the boundary

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -\sigma_n\mathbf{n}$$

where σ_n is a scalar corresponding to a pressure (sometimes referred to as the hydrostatic stress).

NORMAL STRESS

Enter a value or expression for the **Inward normal stress** σ_n (SI unit: $\mathrm{N/m^2}$).

Normal Impedance

Use the **Normal Impedance** node to specify a normal specific impedance Z_0 on a boundary. This feature is useful outside the viscous boundary layer, as this condition mimics the behavior of a corresponding **Pressure Acoustics Model** with a normal impedance condition. The boundary condition reads:

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -Z_0(\mathbf{u}_t\cdot\mathbf{n})\mathbf{n}$$

NORMAL IMPEDANCE

Enter a value or expression for the **Normal impedance** Z_0 (SI unit: Pa·s/m).

Adiabatic

Use the **Adiabatic** node to define a situation with no heat flow into or out of the boundary

$$-\mathbf{n} \cdot (-k\nabla T_t) = 0$$

Temperature Variation

Use the **Temperature Variation** node to define the temperature variation on the boundary T_{bnd}. In the frequency domain this is the amplitude of a harmonic temperature variation

$$T_t = T_{bnd}$$

TEMPERATURE VARIATION

Enter a value or expression for the **Temperature variation** T_{bnd} (SI unit: K).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Heat Flux

Use the **Heat Flux** node to define a thermal source given by an inward normal heat flux $q_n = -\mathbf{n}\cdot\mathbf{q}$ through the boundary

$$-\mathbf{n} \cdot (-k\nabla T_t) = q_n$$

HEAT FLUX

Enter a value for the **Inward normal heat flux** q_n (SI unit: W/m^2).

The Thermoviscous Acoustics, Transient Interface

The **Thermoviscous Acoustics, Transient (tatd)** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, is used to compute the transient evolution of the acoustic variations in pressure, velocity, and temperature. The interface is the time domain equivalent of The Thermoviscous Acoustics, Frequency Domain Interface. This physics interface is required to accurately model acoustics in geometries of small dimensions. Near walls, viscous losses and thermal conduction become important because a boundary layer exists. The thicknesses of the boundary layers is known as the viscous and thermal penetration depth. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations. It is, for example, used when modeling the response of transducers like microphones, miniature loudspeakers and receivers. Other applications include analyzing feedback in hearing aids, smart phones and in mobile devices, or studying the damped vibrations of MEMS structures.

The physics interface solves the equations in the time domain and linear acoustics is assumed.

The equations defined by the Thermoviscous Acoustics, Transient interface are the linearized Navier-Stokes equations in quiescent background conditions solving the continuity, momentum, and energy equations. Due to the detailed description necessary when modeling thermoviscous acoustics, the model simultaneously solves for the acoustic pressure p, the acoustic velocity variation $\mathbf{u}$ (particle velocity), and the acoustic temperature variations T. It is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

The Thermoviscous Acoustics, Transient interface is, as the Frequency Domain variant, formulated in the so-called scattered field formulation where the total field (subscript t) is the sum of the scattered field (the field solved for, p, $\mathbf{u}$, and T) and a possible background acoustic field (subscript b), such that

$$p_\mathrm{t} = p + p_\mathrm{b} \qquad \mathbf{u}_\mathrm{t} = \mathbf{u} + \mathbf{u}_\mathrm{b} \qquad T_\mathrm{t} = T + T_\mathrm{b}$$

When no Background Acoustic Fields feature is present (the background field values are zero per default) the total field is simply the field solved for, $p_\mathrm{t} = p$, $\mathbf{u}_\mathrm{t} = \mathbf{u}$, and

$T_t = T$. All governing equations and boundary conditions are formulated in the total field variables.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Thermoviscous Acoustics Model**, **Wall**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Thermoviscous Acoustics** to select physics features from the context menu.

For good modeling strategies, solver suggestions, postprocessing information, as well as tips and tricks, see the Modeling with the Thermoviscous Acoustics Branch section.

For more details about the physics interface see On the Thermoviscous Acoustics Physics Interface sub-section, under The Thermoviscous Acoustics, Frequency Domain Interface section. Details about the governing equations are found in the Theory Background for the Thermoviscous Acoustics Branch section.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `tatd`.

EQUATION

Expand the **Equation** section to see the equations solved for with the **Equation form** specified. The default selection for **Equation form** is set to **Study controlled**. The available studies are selected under **Show equations assuming**.

- For **Study controlled**, the scaling of the equations is optimized for the numerical performance of the different solvers and study types.
- For **Frequency domain** you can manually enter the scaling parameter Δ under the **Thermoviscous Acoustics Equation Settings** section.

TRANSIENT SOLVER SETTINGS

Select the **Time stepping** (method) as **Manual** (default and recommended) or **Automatic/free** and then enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

Details about Transient Solver Settings are found in the section Modeling with the Thermoviscous Acoustics Branch.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. From the list select the element order and type (Lagrange or serendipity) for the **Pressure**, the **Velocity field**, and the **Temperature variation**, respectively. The default is **Linear** for the pressure and **Quadratic Lagrange** for the velocity and the temperature.

- For numerical stability reasons the element order for the pressure should one less than the element order for the velocity.
- In fluids where the thermal and viscous boundary layer thickness are of the same order of magnitude (where the Prandtl number Pr is of the order 1, like in air), it is recommended to use the same shape order for the temperature and the velocity. Both fields vary equally over the same length scale in the acoustic boundary layers near walls.

Choosing between Lagrange and Serendipity Shape Functions has influence on the number of DOFs solved for and on stability for distorted mesh.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Pressure** p, the **Velocity field** $\mathbf{u}$ and its components, and the **Temperature variation** T. The names can be changed but the names of fields and dependent variables must be unique within a model.

Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Boundary Mode Interface

The Thermoviscous Acoustics, Transient Interface has these domain, boundary, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

	In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

	The Continuity node with this physics interface is available as a pair boundary condition. This gives continuity in pressure, temperature variation, velocity, and in the flux on a pair boundary between thermoviscous acoustic domains.

All of the boundary conditions available with the Thermoviscous Acoustics, Transient interface are the same as for the Frequency Domain version, and are described there

(see the following list). Some of the following nodes are available from the **Mechanical** and **Thermal** submenus (listed in alphabetical order):

- Adiabatic[1]
- Axial Symmetry[1]
- Background Acoustic Fields
- Heat Flux[1]
- Heat Source[1]
- Initial Values[1]
- Interior Normal Impedance[1]
- Interior Temperature Variation[1]
- Interior Velocity[1]
- Interior Wall[1]
- Isothermal[1]
- Normal Impedance[1]
- Normal Stress[1]
- No Slip[1]
- No Stress[1]
- Periodic Condition[2]
- Pressure (Adiabatic)[1]
- Slip[1]
- Stress[1]
- Symmetry[1]
- Temperature Variation[1]
- Thermoviscous Acoustics Model
- Velocity[1]
- Wall[1]

[1]Described for the Thermoviscous Acoustics, Frequency Domain interface

[2]Described for the Pressure Acoustics, Frequency Domain interface

Thermoviscous Acoustics Model

Use the **Thermoviscous Acoustics Model** node to define the model inputs (the background equilibrium temperature and pressure) and the material properties of the fluid (equilibrium density, dynamic viscosity, bulk viscosity, thermal conductivity, and heat capacity at constant pressure) necessary to model the transient propagation of acoustic compressible waves in a thermoviscous acoustic context. Extended inputs are available for the coefficient of thermal expansion and the compressibility, which enables modeling of any constitutive relation for the fluid.

The **Model Inputs**, the **Thermoviscous Acoustics Model**, and the **Thermal Expansion and Compressibility** sections are the same as for the frequency domain interface. They are described under the Thermoviscous Acoustics Model section in The Thermoviscous Acoustics, Frequency Domain Interface.

Note that for the transient interface it is equally important, as for the frequency domain interface, to set the material parameters correctly. The values of the coefficient of

thermal expansion and the isothermal compressibility are prone to errors, so make sure to take a thorough look at the Thermal Expansion and Compressibility section.

Background Acoustic Fields

When the **Background Acoustic Fields** feature is added to a domain it is possible to define the value of the background acoustic field variables p_b, $\mathbf{u}_b$, and T_b. Using this feature it is possible to set up scattering problems as well as defining acoustic fields at an inlet of a waveguide (using a small domain at the inlet).

BACKGROUND ACOUSTIC FIELDS

Enter expressions for the **Background acoustic pressure** p_b, the **Background acoustic velocity** $\mathbf{u}_b$, and the **Background temperature variation** T_b. These can be analytical expressions or values of dependent variables solved in a previous study that defines the background field. The time domain version of the background acoustics field feature does not include the plane wave option available fore the frequency domain interface.

The Thermoviscous Acoustics, Boundary Mode Interface

The **Thermoviscous Acoustics, Boundary Mode (tabm)** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, is used to compute and identify propagating and non-propagating modes in waveguides and ducts. The interface performs a boundary mode analysis on a given boundary including the thermal and viscous loss effects that are important in the acoustic boundary layer near walls.

The interface is applied at boundaries which represent the cross section or the inlet of a waveguide or duct of small dimensions. It solves for the acoustic variations of pressure p, velocity $\mathbf{u}$, and temperature T, as well as the out-of-plane wave number k_{n} of the modes. Near walls, viscous losses and thermal conduction become important because boundary layers exists. The thickness of these layers is known as the viscous and thermal penetration depth. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations. The Thermoviscous Acoustics, Boundary Mode interface is, for example, used when setting up sources in systems with small ducts like hearing aids or mobile devices. It can also be used to identify the propagating wave number and characteristic impedance of a duct cross section and use that information in the homogenized Narrow Region Acoustics model in The Pressure Acoustics, Frequency Domain Interface.

The Thermoviscous Acoustics, Boundary Mode interface solves the equations defined by the linearized Navier-Stokes equations (linearized continuity, momentum, and energy equations), in quiescent background conditions, on boundaries, searching for the out-of-plane wave numbers at a given frequency. All gradients in the governing equations are expressed in terms of the in-plane gradient and the out-of-plane wave number that is being solved for. Due to the detailed description necessary when modeling thermoviscous acoustics, the model simultaneously solves for the acoustic pressure p, the velocity variation $\mathbf{u}$ (particle velocity), and the acoustic temperature variations T. The interface is available on boundaries for 3D and on edges for 2D axisymmetric geometries.

For details about the governing equations see the Theory Background for the Thermoviscous Acoustics Branch.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Thermoviscous Acoustics Model**, **Wall**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Thermoviscous Acoustics, Boundary Mode** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `tabm`.

EQUATION

Expand the **Equation** section to see the equations solved for with the **Equation form** specified. The default selection is **Equation form** is set to **Study controlled**. The available studies are selected under **Show equations assuming**.

- For **Study controlled**, the frequency used for the mode analysis study is given in the study.
- For **Mode analysis** you can set the frequency manually. The default **Mode analysis frequency** f is 100 Hz.

SOUND PRESSURE LEVEL SETTINGS

See the settings for Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. From the list select the element order for the **Pressure**, the **Velocity field**, and the **Temperature variation**. The default uses **Linear** elements for the pressure and **Quadratic** for the velocity field and the temperature variations.

In order for the system to be numerically stable, it is important that the order for the pressure degree of freedom (DOF) is one lower than the velocity field. Per default, the velocity components and the temperature share the same element order as they vary

similarly over the same length scale in the acoustic boundary layer. Therefore, both typically require the same spatial accuracy.

Increase the element order to, for example, quadratic for the pressure and cubic for the velocity and temperature dofs. This can add additional accuracy but it also adds additional degrees of freedom compared to the default setting and will be computationally more expensive.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Pressure** p, the **Velocity field** $\mathbf{u}$ and its components, and the **Temperature variation** T. The names can be changed but the names of fields and dependent variables must be unique within a model.

- Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface
- Theory Background for the Thermoviscous Acoustics Branch

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Boundary Mode Interface

The Thermoviscous Acoustics, Boundary Mode Interface has these boundary and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

	In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

	The Continuity node with this physics interface is available as a pair boundary condition. This gives continuity in pressure, temperature variation, velocity and in the flux on a pair boundary between thermoviscous acoustic domains.

All of the boundary conditions available with the Thermoviscous Acoustics, Boundary Mode interface are the same as for the Frequency Domain and Transient version, and are described there (see the following list). The Thermoviscous Acoustics Model feature is described here. Some of the following nodes are available from the **Mechanical** and **Thermal** submenus (listed in alphabetical order):

- Adiabatic[1]
- Heat Flux[1]
- Heat Source[1]
- Initial Values[1]
- Isothermal[1]
- Normal Impedance[1]
- Normal Stress[1]
- No Slip[1]
- No Stress[1]
- Pressure (Adiabatic)[1]
- Slip[1]
- Stress[1]
- Symmetry[1]
- Temperature Variation[1]
- Thermoviscous Acoustics Model
- Velocity[1]
- Wall[1]

[1]Described for the Thermoviscous Acoustics, Frequency Domain interface

	In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Thermoviscous Acoustics Model

Use the **Thermoviscous Acoustics Model** node to define the model inputs (the background equilibrium temperature and pressure) and the material properties of the fluid (dynamic viscosity, bulk viscosity, thermal conductivity, heat capacity at constant pressure, and equilibrium density) necessary to model the propagation of acoustic compressible waves in a thermoviscous acoustic context. Extended inputs are available for the coefficient of thermal expansion and the compressibility, which enables modeling of any constitutive relation for the fluid.

The **Model Inputs**, the **Thermoviscous Acoustics Model**, and the **Thermal Expansion and Compressibility** sections are the same as for the frequency domain and transient interfaces. They are described under the Thermoviscous Acoustics Model section in The Thermoviscous Acoustics, Frequency Domain Interface.

Note that for the boundary mode interface it is equally important as for the frequency domain and transient interfaces to set the material parameters correctly. The values of the coefficient of thermal expansion and the isothermal compressibility are prone to errors, so make sure to take a thorough look at the Thermal Expansion and Compressibility section.

BOUNDARY MODE SETTINGS

This section contains an input for the **Propagation direction** $\mathbf{n}$ of the modes analyzed with the interface. Typically, this direction is equal to the surface or boundary normal. The default value is (`tabm.nx`, `tabm,ny`, `tabm,nz`) in 3D and (`tabm.nr`, `tabm.nz`) in 2D axisymmetric. When setting up models, this vector defines the positive propagation direction for the waves and thus the sign of the wave number.

The Acoustic-Thermoviscous Acoustic Interaction, Frequency Domain Interface

The **Acoustic-Thermoviscous Acoustic Interaction, Frequency Domain** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, combines the Thermoviscous Acoustics, Frequency Domain and Pressure Acoustics, Frequency Domain interfaces together with the Acoustic-Thermoviscous Acoustic Boundary coupling feature. To couple the two physics and use Thermoviscous Acoustics only where necessary can reduce the computational cost of a model.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. Linear acoustics is assumed.

When a predefined **Acoustic-Thermoviscous Acoustic Interaction, Frequency Domain** interface is added from the **Acoustics>Thermoviscous Acoustics** branch of the **Model Wizard** or the **Add Physics** windows, the **Thermoviscous Acoustics, Frequency Domain** and **Pressure Acoustics, Frequency Domain** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Acoustic-Thermoviscous Acoustic Boundary**.

	For details about all the multiphysics couplings in the Acoustics Module, see Multiphysics Couplings.

	The use of a predefined multiphysics coupling approach helps improve the flexibility and design options for your modeling. For for details, see The Multiphysics Node and Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

On the Constituent Physics Interfaces

The Pressure Acoustics, Frequency Domain interface is used to compute the pressure variations when modeling the propagation of acoustic waves in fluids at quiescent background conditions. The physics interface solves the Helmholtz equation and is suited for all linear frequency-domain acoustic simulations with harmonic variations of

the pressure field. It includes domain conditions to model losses in a homogenized way, so-called fluid models for porous materials, as well as losses in narrow regions. The domain features also include background incident acoustic fields.

The Thermoviscous Acoustics, Frequency Domain interface is used to compute the propagation of acoustic waves including thermal and viscous losses. The interface solves for the acoustic variations in pressure, velocity, and temperature. Thermoviscous acoustics is required to accurately model acoustics in geometries with small dimensions. Near walls a viscous and thermal boundary layer exists. Here viscous losses due to shear and thermal conduction become important because of large gradients. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations. The equations defined by the Thermoviscous Acoustics, Frequency Domain interface are the linearized Navier-Stokes equations in quiescent background conditions solving the continuity, momentum, and energy equations.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics are added using the predefined multiphysics interface, for example **Acoustic-Thermoviscous Acoustic Interaction, Frequency Domain**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Thermoviscous Acoustics, Frequency Domain** on one side and **Pressure Acoustics, Frequency Domain** on the other.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

See Acoustic-Thermoviscous Acoustic Boundary in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Thermoviscous Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface.
- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface

- Theory Background for the Pressure Acoustics Branch
- Theory Background for the Thermoviscous Acoustics Branch

The Thermoviscous Acoustic-Solid Interaction, Frequency Domain Interface

The **Thermoviscous Acoustic-Solid Interaction, Frequency Domain** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, combines the Thermoviscous Acoustics, Frequency Domain and Solid Mechanics interfaces. The physics interface solves for, and has predefined couplings between, the displacement field in the solid and the acoustic variations in the fluid domains.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. Linear acoustics is assumed.

When a predefined **Thermoviscous Acoustic-Solid Interaction, Frequency Domain** interface is added from the **Acoustics>Thermoviscous Acoustics** branch of the **Model Wizard** or the **Add Physics** windows, the **Thermoviscous Acoustics, Frequency Domain** and **Solid Mechanics** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Thermoviscous Acoustic-Structure Boundary**.

For details about all the multiphysics couplings in the Acoustics Module, see Multiphysics Couplings.

On the Constituent Physics Interfaces

The Thermoviscous Acoustics, Frequency Domain interface is used to compute the acoustic variations of pressure, velocity, and temperature. The physics interface is required to accurately model acoustics in geometries of small dimensions. Near walls, viscosity and thermal conduction become important because it creates a viscous and a thermal boundary layer where losses are significant. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations.

The Solid Mechanics interface is intended for general structural analysis of 3D, 2D or 2D axisymmetric bodies. The physics interface is based on solving Navier's equations, and results such as displacements, stresses, and strains are computed.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics are added using the predefined multiphysics interface, for example **Thermoviscous Acoustic-Solid Interaction, Frequency Domain**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Thermoviscous Acoustics, Frequency Domain** on one side and **Solid Mechanics** on the other.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

See Thermoviscous Acoustic-Structure Boundary in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Thermoviscous Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface.

- Theory Background for the Thermoviscous Acoustics Branch

- Regarding the available structural-mechanics features and their theory background, see The Solid Mechanics Interface.

The Thermoviscous Acoustic-Shell Interaction, Frequency Domain Interface

This physics interface requires a Structural Mechanics Module license. For theory and physics interface feature descriptions relating to the Shell interface, see the *Structural Mechanics Module User's Guide*.

The **Thermoviscous Acoustic-Shell Interaction, Frequency Domain** interface ( ), found under the **Acoustics>Thermoviscous Acoustics** branch ( ) when adding a physics interface, combines the Thermoviscous Acoustics, Frequency Domain and Shell interfaces. The physics interface solves for and has a predefined coupling between the displacement field of the shell and the acoustic variations in the fluid domains. It can be used, for example, for modeling the vibrating response of micromirrors in MEMS applications. The physics interface is available for 3D geometries, and it is capable of modeling the coupled thermoviscous acoustics and shell vibrations in the frequency domain.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. Linear acoustics is assumed.

When a predefined **Thermoviscous Acoustic-Shell Interaction, Frequency Domain** interface is added from the **Acoustics>Thermoviscous Acoustics** branch of the **Model Wizard** or the **Add Physics** windows, the **Thermoviscous Acoustics, Frequency Domain** and **Shell** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling features **Thermoviscous Acoustic-Structure Boundary**. This feature automatically couples the thermoviscous acoustic domain to exterior shells (on exterior boundaries) and to interior shells (on interior boundaries).

For details about all the multiphysics couplings in the Acoustics Module, see Multiphysics Couplings.

On the Constituent Physics Interfaces

The Thermoviscous Acoustics, Frequency Domain interface is used to compute the acoustic variations of pressure, velocity, and temperature. The physics interface is required to accurately model acoustics in geometries of small dimensions. Near walls, viscosity and thermal conduction become important because it creates a viscous and a thermal boundary layer where losses are significant. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations.

The Shell interface is used to model structural shells on 3D boundaries. Shells are thin flat or curved structures, having significant bending stiffness. The physics interface uses shell elements of the MITC type, which can be used for analyzing both thin (Kirchhoff theory) and thick (Mindlin theory) shells. Geometric nonlinearity can be taken into account. The material is assumed to be linearly elastic.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics are added using the predefined multiphysics interface, for example **Thermoviscous Acoustic-Shell Interaction, Frequency Domain**, the **Selection** on the coupling feature is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Thermoviscous Acoustics, Frequency Domain** on one side (exterior shells) or on both sides (interior shells) of **Shell** boundaries.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND COUPLING FEATURES

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

See Thermoviscous Acoustic-Structure Boundary in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Thermoviscous Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface.
- The available physics features for Results Evaluation are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*

- Theory Background for the Thermoviscous Acoustics Branch
- Theory for Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*

Modeling with the Thermoviscous Acoustics Branch

This section contains modeling tips and tricks, information about solvers, postprocessing variables, and meshing.

In this section:

Meshing the Boundary Layer

When modeling using the Thermoviscous Acoustics interfaces several length scales become important when setting up the mesh.

First, there is the wavelength which should still be resolved as in pressure acoustics, see Meshing (Resolving the Waves) in the Pressure Acoustics Interfaces chapter.

Secondly, there is the thickness of the viscous and thermal boundary layers. In order for the model to include the correct amount of damping the boundary layers need to be resolved. Ideally this is done using a **Boundary Layers** mesh. The **Thickness of first boundary layer** and the **Number of boundary layers** should be set such that they resolve the boundary layer at the specific modeling frequency. Remember that the boundary layer thickness scales as one over the square root of the frequency.

A great deal of information about meshing and modeling with the Thermoviscous acoustics interface can be found in two COMSOL Blog posts. They are:

- www.comsol.com/blogs/theory-thermoacoustics-acoustics-thermal-viscous-losses/
- www.comsol.com/blogs/modeling-thermoacoustic-interface-comsol/

See the Boundary Layers section in the *COMSOL Multiphysics Reference Guide* for more details.

Finally, it is important to consider the thickness of the boundary layer compared to the physical dimensions of the model, for example, the viscous boundary layer thickness δ_v compared to the tube radius a. This is sometimes known as the Womersley number

$$\mathrm{Wo} = \sqrt{\frac{\omega\rho a^2}{\mu}} = \frac{a\sqrt{2}}{\delta_v} = \frac{a}{\delta_t}\sqrt{\frac{2}{\mathrm{Pr}}}.$$

where δ_v is the viscous boundary layer thickness and Pr is the Prandtl number. If the Womersley number is very small, say $\mathrm{Wo} < 0.1$, the effect associated with the losses in the viscous boundary layer can normally be disregarded. In this case the boundary layer need not be meshed and a **Slip** condition can be used instead of a **No-slip** condition. The same is true for the thermal boundary layer thickness compared to the tube radius. Here an **Isothermal** condition can be replaced by an **Adiabatic** condition.

If the **No-slip** and/or **Isothermal** conditions are kept (when $\mathrm{Wo} < 0.1$) then remember to at least add one boundary layer mesh which is of roughly the size of the acoustic boundary layer. If this is not done erroneous losses can be introduced in the model.

Solver Suggestions for Large Thermoviscous Acoustics Models

Solving thermoviscous acoustic problems can easily involve solving for many degrees of freedom (DOFs) as the model solves for the acoustic variations in both pressure, velocity field (2 or 3 components), and temperature. First of all, it is important to restrict the use of the thermoviscous acoustics model to domains and regions of the model where it is necessary. Couple to Pressure Acoustics using the Multiphysics coupling or use the Narrow Region Acoustics feature of Pressure Acoustics to reduce the number of DOFs. Secondly, care should be taken when meshing the computational domain. If these two things have been carefully considered, the solver can be changed from its default *Direct* setting to use an *Iterative* solver. Depending on the model size and involved physics two options are described below.

In both cases a good starting point for setting up a new solver configuration is to right click the study node and select **Show Default Solver**, then expand the **Solver Configuration** tree under **Stationary Solver** or **Time-Dependent Solver**. Predefined iterative solver suggestions are automatically generated. Per default a direct solver is used and two iterative solvers are suggested and disabled (grayed out). To turn on one of these

approached right-click the solver and select **Enable** (or press F4). The first suggestion *(GMRES with Direct Precon.)* uses an iterative solver with a direct preconditioner. This method is typically faster then the direct solver and uses 20% less memory. The second suggestion *(GMRES with DD)* uses an iterative solver with the domain decomposition method. This method is very robust (also for multiphysics applications) and very memory efficient, but it can be slow. Both suggestions are described below as well as how to set them up manually. In liquids where thermal effects can be neglected the model can be solved in the adiabatic case and DOFs saved. Finally, choosing different shape functions can also reduce the memory consumption.

- Iterative Solver with Direct Preconditioner using Hybridization
- Iterative Solver with Domain Decomposition
- Solving in the Adiabatic Case
- Choosing Shape Functions

ITERATIVE SOLVER WITH DIRECT PRECONDITIONER USING HYBRIDIZATION

For large 2D problems and 3D problems, that only involve thermoviscous acoustics, using the following approach will save around 20% memory and can speed up the solution procedure by a factor 2 or 3. Under the *Stationary Solver* node take the following steps: Add an **Iterative** solver with the GMRES solver. As preconditioner add the **Direct Preconditioner** and switch the solver to PARDISO. Expand the **Hybridization** section and select **Multi preconditioner** in the **Preconditioner variables** list add the *Pressure* and *Velocity field*. Add a second direct preconditioner with the same settings but now select only the *Temperature* as preconditioner variables. The reason for splitting the equations up in this manner is that the energy equation is only loosely coupled to the momentum and continuity equations.

See the Direct Preconditioner section in the *COMSOL Multiphysics Reference Guide* for more details.

The direct preconditioner approach is used in the model: *Transfer Impedance of a Perforate*. Application Library path **Acoustics_Module/Tutorials/transfer_impedance_perforate**

ITERATIVE SOLVER WITH DOMAIN DECOMPOSITION

A more advanced approach, to handle very large 3D models, is again to use the GMRES iterative solver but now with the domain decomposition preconditioner. This approach can also be used for multiphysics problems involving several physics.

Start by adding an **Iterative** solver and select GMRES as the solver. Then right click the iterative node and select **Domain Decomposition.** A good starting point, to work with this solver, is to use the default settings with only a few changes:

- If you set **Recompute and clear subdomain** data to **On** you will get a very memory lean solver (but the solution may take longer). **Recompute and clear subdomain data** is a selector with the options **Automatic**, **Off**, **On**. If the option **Automatic** is chosen, the recompute and clear mechanism is activated if there is an out-of-memory-error during the domain decomposition setup phase. The setup is then repeated with recompute and clear activated (which can be costly in terms of time, but is better than failure). A warning is given in this case.
- In the **Coarse Level** section change the **Use coarse level** to **Algebraic**.
- In the direct solver under the **Coarse Solver** and **Domain Solvers** sub-nodes use the PARDISO solver.

This type of approach should allow you to solve large thermoviscous acoustics models, also including multiphysics interactions, using a minimum of RAM. Possibly increase the value of the **Maximum number of DOFs per subdomain** option to use a larger amount of the RAM at your disposition.

See the Domain Decomposition section in the *COMSOL Multiphysics Reference Guide* for more details

SOLVING IN THE ADIABATIC CASE

In certain cases, it can be interesting not to include thermal conduction in the model and treat all processes as adiabatic (isentropic). This is, for example, relevant for fluids where the thermal boundary layer is much thinner than the viscous. Not solving for the temperature field T also saves some degrees of freedom (DOFs).

This is achieved by setting the **Isothermal compressibility** to **User defined** and here enter the adiabatic value β_0 (remember that $\beta_0 = \gamma \cdot \beta_T$). Then, in the solver sequence under **Solver Configuration > Solver 1 > Dependent Variables** select **Define by study step** to **User defined** and under **> Temperature variation (mod1.T)** click to clear the **Solver for this field** check box.

CHOOSING SHAPE FUNCTIONS

In models with structured mesh it can be advantageous to switch to the serendipity shape functions instead of the default Lagrange, see Lagrange and Serendipity Shape Functions below. In general if a boundary layer mesh is used (to resolve the thermal and viscous boundary layers) and/or if a PML us used in the model, the mesh contains structured mesh regions.

Lagrange and Serendipity Shape Functions

In most of the physics interfaces in the Acoustics Module and specifically in the Thermoviscous Acoustic interfaces, you can choose between two families of shape functions: *Lagrange* and *serendipity*. The current default is to use Lagrange shape functions. To display the **Discretization** section, click the **Show** button ( ) and select **Discretization**.

When using a structured mesh it may be advantageous to switch to the serendipity elements as they generate significantly fewer degrees of freedom (DOFs). The accuracy is in most cases almost as good as for the Lagrange elements. The Lagrange elements are however less sensitive to strong mesh distortions.

The serendipity shape functions differs from the Lagrange shape functions only for the following element shapes:

- 2D: Quadrilateral elements of discretization order higher than 1.
- 3D: Hexahedral, prism, and pyramid elements of discretization order higher than 1.

In the *COMSOL Multiphysics Reference Manual*:

- The Lagrange Element
- The Nodal Serendipity Element

When coupling two physics interfaces that have the same DOFs like, for example, displacement, the same type of shape functions should be used in both interfaces to ensure conformity. Since there is no difference between the two families of shape functions in 1D, this is not an issue when connecting edges.

Transient Solver Settings

When solving transient wave problems it is recommended to use manual time stepping in the solver. In general there is no point in using an automatic time-step control which

can be provided by the time-dependent solver. The tolerances in the automatic error control are difficult to tune in wave problems when there is weak but important high-frequency content. It is recommended to use the suggestion generated when the **Manual** method is chosen (the default) in the **Transient Solver Settings** section. The internal time step of the solver that is generated, when the **Maximal frequency to resolve** is set, will in most cases produce a solver that is adequate. It is assuming that the user has generated a mesh that properly resolves the same maximal frequency (minimal wavelength).

For further details see the Time Stepping in Transient Models section in Modeling with the Thermoviscous Acoustics Branch.

Postprocessing Variables

This section contains information about variables for:

- Viscous and Thermal Boundary Layer Variables
- Material Properties
- Stress Tensor Components
- Power Dissipation and Intensity Variables
- In and Out-of-Plane Variables for the Boundary Mode Interface

In the variables names, in the tables below, *phys_id* represents the interface name. For example, `ta` for the frequency domain interface, `tatd` for the transient interface, and `tabm` for the boundary mode interface.

VISCOUS AND THERMAL BOUNDARY LAYER VARIABLES

The viscous and thermal boundary layer thickness (thermal and viscous penetration depth) as defined in Theory Background for the Thermoviscous Acoustics Branch can be evaluated in postprocessing for the frequency domain models. The same is the case for the Prandtl number relating the two length scales (available in both frequency and time domain).

TABLE 5-2: BOUNDARY LAYER VARIABLES

VARIABLE	DESCRIPTION
phys_id.d_visc	Viscous boundary layer thickness

TABLE 5-2: BOUNDARY LAYER VARIABLES

VARIABLE	DESCRIPTION
phys_id.d_therm	Thermal boundary layer thickness
phys_id.Pr	Prandtl number

MATERIAL PROPERTIES

Material properties are readily available for postprocessing in plots. In plots click the **Replace Expression** icon and browse to the **Material properties** list under the thermoviscous acoustics interface. Important parameters to plot are the coefficient of thermal expansion *phys_id*.alpha0 and the isothermal compressibility *phys_id*.betaT. These should not evaluate to zero.

STRESS TENSOR COMPONENTS

The stress tensor components are defined as variables and can be evaluated in postprocessing or used to create exotic couplings between physics. In the table below only the xx component of the stress tensor and the x component of the stress are shown. Change the spatial reference accordingly,

TABLE 5-3: STRESS VARIABLES ONE COMPONENT IN 3D

VARIABLE	DESCRIPTION
phys_id.K_stress_tensorxx	Viscous stress tensor, xx-component
phys_id.T_stress_tensorxx	Total stress tensor, xx-component
phys_id.K_stressx	Viscous stress, x-component (on boundaries)
phys_id.T_stressx	Total stress, x-component (on boundaries)

POWER DISSIPATION AND INTENSITY VARIABLES

The thermoviscous acoustics interface models the energy dissipation process which stem from viscous and thermal dissipation processes. The amount of dissipated energy can be of interest as a results analysis variable or as a source term for a multiphysics problem.

The energy conservation-dissipation corollary describes the transport and dissipation of energy in a system (see Ref. 1 pp. 516 or Ref. 6). In linear acoustics, this equation is derived by taking the dot product (scalar product) of the momentum and the velocity **v**, adding it to the continuity equation, and then adding the entropy. After some manipulation and integration, the use of Gauss' theorem yields Equation 5-1

$$\frac{\partial}{\partial t}\int_{\Omega} w\,dV + \int_{\partial\Omega} (\mathbf{i}\cdot\mathbf{n})\,dA = -\int_{\Omega}\Delta\,dV \qquad \text{or} \qquad \frac{\partial w}{\partial t} + \nabla\cdot\mathbf{i} = -\Delta$$

$$w = \frac{1}{2}\rho_0 u^2 + \frac{1}{2}\frac{p^2}{\rho_0 c_0^2} + \frac{1}{2}\frac{\rho_0 T_0}{C_p}s^2$$

$$\mathbf{i} = p\mathbf{u} - [\mathbf{u}^{\mathsf{T}}\cdot\boldsymbol{\tau}]^{\mathsf{T}} - \frac{k}{T_0}T\nabla T \tag{5-1}$$

$$\Delta = \boldsymbol{\tau}:\nabla\mathbf{u} + \frac{k}{T_0}(\nabla T)^2 = \Delta_v + \Delta_t$$

where w is the disturbance energy of the control volume, $u = |\mathbf{u}|$ is the velocity, T is the temperature variation, p is the acoustic pressure variations, p_0 is the background equilibrium pressure, T_0 the background equilibrium temperature, ρ_0 the background density, c_0 the (isentropic) speed of sound, C_p the heat capacity at constant pressure (per unit mass), k the coefficient of thermal conduction, $\mathbf{i}$ is the instantaneous intensity (flux of energy out of a control volume), Δ is the dissipated energy per unit volume and time (SI unit: Pa/s = J/(m^{3}s) = W/m^3), s is the entropy, $\boldsymbol{\tau}$ is the viscous stress tensor, $\boldsymbol{\tau}:\nabla\mathbf{u}$ is the viscous dissipation function, and T indicates transpose of vector. Δ_v and Δ_t are the viscous and thermal contributions to the dissipation function. In Equation 5-1 we have made use of Ref. 6 for the expression for the intensity $\mathbf{I}$.

In the Thermoviscous Acoustics, Frequency Domain interface, the dissipation term Δ is directly given by the RMS value of the tensor expression

$$\Delta_v = \langle\boldsymbol{\tau}:\nabla\mathbf{u}\rangle = \frac{1}{4}(\boldsymbol{\tau}^*:\nabla\mathbf{u} + \boldsymbol{\tau}:(\nabla\mathbf{u})^*) \tag{5-2}$$

where "`:`" in Equation 5-2 is the double dot operator (or total inner product) and `*` is the complex conjugate. In the above expressions, the time averaged expressions for a product in the frequency domain is defined as:

$$\langle AB\rangle = \langle \mathrm{Re}(Ae^{i\omega t})\mathrm{Re}(Be^{i\omega t})\rangle = \frac{1}{4}(A^*B + AB^*)$$

The power dissipation variables are defined in Table 5-4.

TABLE 5-4: POWER DISSIPATION VARIABLES

VARIABLE	DESCRIPTION
phys_id.diss_therm	Thermal power dissipation density
phys_id.diss_visc	Viscous power dissipation density
phys_id.diss_tot	Total thermal and viscous power dissipation density

In the Thermoviscous Acoustics, Frequency Domain interface the (time averaged) intensity $\mathbf{I}$ is given by averaging the instantaneous intensity vector $\mathbf{i}$ in Equation 5-1 using the same time averaged products defined above. The intensity and intensity magnitude are defined in Table 5-5.

TABLE 5-5: INTENSITY VARIABLES

VARIABLE	DESCRIPTION
phys_id.Ix	Intensity x-component (in 1D, 2D, and 3D)
phys_id.Iy	Intensity y-component (in 2D, and 3D)
phys_id.Iz	Intensity z-component (2D axisymmetric and 3D)
phys_id.Ir	Intensity r-component (2D axisymmetric)
phys_id.I_mag	Intensity magnitude

In the Thermoviscous Acoustics, Transient interface the instantaneous intensity $\mathbf{i}$ variables is available for post processing. The instantaneous intensity and instantaneous intensity magnitude are defined in Table 5-6.

TABLE 5-6: INSTANTANEOUS INTENSITY VARIABLES

VARIABLE	DESCRIPTION
phys_id.Iix	Instantaneous intensity x-component (in 1D, 2D, and 3D)
phys_id.Iiy	Instantaneous intensity y-component (in 2D, and 3D)
phys_id.Iiz	Instantaneous intensity z-component (2D axisymmetric and 3D)
phys_id.Iir	Instantaneous intensity r-component (2D axisymmetric)
phys_id.Ii_mag	Instantaneous intensity magnitude

IN AND OUT-OF-PLANE VARIABLES FOR THE BOUNDARY MODE INTERFACE

Several dedicated variables exist for the The Thermoviscous Acoustics, Boundary Mode Interface where quantities are defined in terms of their in-plane and out-of-plane values. For example, the intensity variable $\mathbf{I}$ has the following derived values

$$\mathbf{I}_{\text{ip}} = \mathbf{I} - (\mathbf{I} \cdot \mathbf{n})\mathbf{n}$$
$$\mathbf{I}_{\text{op}} = (\mathbf{I} \cdot \mathbf{n})\mathbf{n}$$

where ip stands for in-plane and op for out-of-plane. These two variables are named `tabm.Iip` and `tabm.Iop` (with spatial components `x`, `y`, and `z`). The magnitude of these two variables is given by `tabm.Iip_mag` and `tabm.Iop_mag`. In the same manner variables exist for the acceleration and the velocity.

Theory Background for the Thermoviscous Acoustics Branch

The Thermoviscous Acoustics, Frequency Domain Interface, The Thermoviscous Acoustics, Transient Interface, and The Thermoviscous Acoustics, Boundary Mode Interface are designed for the analysis of acoustics in viscous and thermally conducting, compressible Newtonian fluids. The physics interface solves the linearized Navier-Stokes equation, the continuity equation, and the energy equation. This corresponds to a small parameter expansion of the dependent variables. The physics interface solves for the acoustic pressure variations p, the fluid velocity variations $\mathbf{u}$, and the acoustic temperature variations T. The interface uses a scattered field formulation where it is possible to define background acoustic fields. All equations and boundary conditions are formulated in the total fields and solve for the scattered fields.

The Thermoviscous Acoustics, Frequency Domain Interface is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries. The physics interfaces solves problems in the frequency domain, that is, Frequency Domain, Frequency-Domain Modal, and Eigenfrequency type analysis. In 2D and 1D axisymmetric systems, a Mode Analysis study is also available for the out-of-plane component.

The Thermoviscous Acoustics, Transient Interface is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries. The physics interfaces solves problems in the time domain using the Time Dependent analysis type.

The The Thermoviscous Acoustics, Boundary Mode Interface is available on boundaries in 3D and 2D axisymmetric geometries. The physics interface helps identify and solve propagating and non-propagating modes using the Mode Analysis study.

In this section:

- Formulation for Mode Analysis in 2D and 1D axisymmetric
- Formulation for the Boundary Mode Interface
- References for the Thermoviscous Acoustics, Frequency Domain Interface

The Viscous and Thermal Boundary Layers

In general, a tangential harmonic oscillation of amplitude u_0 and frequency f applied to a wall at $z = 0$ creates a viscous wave of the form

$$u(z) = u_0 e^{-\sqrt{\frac{\pi f \rho_0}{\mu}}(1+i)z}$$

where, f is the frequency, ρ_0 is the static density, and μ is the dynamic viscosity. The viscous shear waves are therefore dispersive with wavelength

$$L_v = 2\pi\sqrt{\frac{\mu}{\pi\rho_0 f}} = 2\pi\delta_v$$

and highly damped since their amplitude decays exponentially with distance from the boundary (see Ref. 3). In fact, in just one wavelength, the amplitude decreases to about 1/500 of its value at the boundary. Therefore, the viscous boundary layer thickness can for most purposes be considered to be less than L_v. The length scale δ_v is the so-called viscous penetration depth or viscous boundary layer thickness.

Similarly, a harmonically oscillating temperature with amplitude T_0 and frequency f at $z = 0$ creates a thermal wave of the form

$$T(z) = T_0 e^{-\sqrt{\frac{\pi f \rho_0 C_p}{k}}(1+i)z}$$

where C_p is the heat capacity at constant pressure and k is the thermal conductivity. The wavelength is here

$$L_t = 2\pi\sqrt{\frac{k}{\pi\rho_0 f C_p}} = 2\pi\delta_t$$

and a decay behavior similar to the viscous waves. The length scale δ_t is here the thermal penetration depth.

The ratio of viscous wavelength to thermal wavelength is a nondimensional number related to the Prandtl number Pr, as

$$\frac{L_v}{L_t} = \sqrt{\frac{\mu C_p}{k}} = \sqrt{\mathrm{Pr}}$$

In air, this ratio is roughly 0.8, while in water, it is closer to 2.7. Thus, at least in these important cases, the viscous and thermal boundary layers are of the same order of magnitude. Therefore, if one effect is important for a particular geometry, so is probably the other.

General Linearized Compressible Flow Equations

In general, the motion of a viscous compressible Newtonian fluid, including the energy equation, is governed by the set of equations listed in Equation 5-3 below.

$$\begin{gathered}
\frac{d\rho}{dt} + \rho(\nabla \cdot \mathbf{u}) = 0 \\
\rho\frac{d\mathbf{u}}{dt} = \nabla \cdot \boldsymbol{\sigma} + \mathbf{F} \\
\rho C_p \frac{dT}{dt} - \alpha_p T\frac{dp}{dt} = -\nabla \cdot \mathbf{q} + \phi + Q \\
\boldsymbol{\sigma} = -p\mathbf{I} + \boldsymbol{\tau} = -p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla \cdot \mathbf{u})\mathbf{I} \\
\mathbf{q} = -k\nabla T \\
\rho = \rho(p, T)
\end{gathered} \tag{5-3}$$

where the dependent variables are pressure p, velocity $\mathbf{u}$, temperature T, and density ρ. The first three equations are the continuity equation, the momentum equation (the Navier-Stokes equation), and the energy equation, respectively. The last three equation are the constitutive equations. They define the total stress tensor $\boldsymbol{\sigma}$ and the viscous stress tensor $\boldsymbol{\tau}$ through Stokes expression, the Fourier heat conduction law, and an equation of state. See, for example, Ref. 1 to 7 for further details.

The material time derivatives (or total derivatives) d/dt are in the following expanded according to

$$\frac{dA(\mathbf{x}, t)}{dt} = \frac{\partial A}{\partial t} + \mathbf{u} \cdot \nabla A$$

where A is a dummy variable. The equation of state relates local values of pressure, density and temperature and is therefore an algebraic equation or an ODE, rather than

a PDE. A common form of state equations is to know the density as function of pressure and temperature, $\rho = \rho(p, T)$. In the following, it is assumed that the state equation has this form.

The basic properties of the fluid are the dynamic viscosity μ and thermal conductivity $\mathbf{k}$. The coefficient μ_B is the bulk (or second) viscosity and describes losses due to compressibility (expansion and contraction of the fluid), where μ describes losses due to shear friction. The bulk viscosity can in some cases be used to model an empirically observed deviation from Stokes' assumption but is usually negligible compared to μ unless the motion is irrotational; see Ref. 3 and Ref. 4. These three properties are taken to be constant or at most weakly temperature-dependent. The specific heat at constant pressure C_p (per unit mass) and the (isobaric) coefficient of volumetric thermal expansion α_p (sometimes denoted α_0),

$$\alpha_p = -\frac{1}{\rho}\frac{\partial \rho}{\partial T}\bigg|_p$$

are both possibly functions of pressure and temperature.

In the energy equation

$$\phi = \boldsymbol{\tau}(\mathbf{u}):S(\mathbf{u}) = \boldsymbol{\tau}(\mathbf{u}):\nabla\mathbf{u}$$

is the viscous dissipation function — that is, the scalar contraction of the viscous stress tensor $\boldsymbol{\tau}$ with the rate of strain tensor S. Both tensors are seen as functions of a velocity vector. If the mean velocity is zero, this term vanishes in the following linearization because it is homogeneous of second order in the velocity gradients. Otherwise, it acts as an oscillating source/sink (the term is included in the full Linearized Navier-Stokes interfaces). In the right-hand sides of Equation 5-3, $\mathbf{F}$ and Q are a volume force and a heat source, respectively.

Acoustic Perturbation and Linearization

For small perturbations around steady-state solution, the dependent variables and sources can be assumed to take on the following form:

$$
\begin{aligned}
\mathbf{u} &= \mathbf{u}_0(\mathbf{x}) + \mathbf{u}_1(t, \mathbf{x}) \\
p &= p_0(\mathbf{x}) + p_1(t, \mathbf{x}) \\
T &= T_0(\mathbf{x}) + T_1(t, \mathbf{x}) \\
\rho &= \rho_0(\mathbf{x}) + \rho_1(t, \mathbf{x}) \\
\mathbf{F} &= \mathbf{F}_0(\mathbf{x}) + \mathbf{F}_1(t, \mathbf{x}) \\
Q &= Q_0(\mathbf{x}) + Q_1(t, \mathbf{x})
\end{aligned}
$$

where the subscript "1" variables represent the acoustic perturbations (1st order perturbation) and subscript "0" the background mean flow quantities. Assuming zero mean flow $\mathbf{u}_0 = \mathbf{0}$ and after inserting into the governing Equation 5-3, the steady-state equations can be subtracted from the system, which is subsequently linearized to first order by ignoring terms quadratic in the acoustic variables. Dropping the subscript "1" for readability yields the thermoviscous acoustic equations:

$$
\begin{aligned}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u}) &= 0 \\
\rho_0 \frac{\partial \mathbf{u}}{\partial t} &= \nabla \cdot \left[-p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_{\mathrm{B}} \right) (\nabla \cdot \mathbf{u}) \mathbf{I} \right] + \mathbf{F} \\
\rho_0 C_{\mathrm{p}} \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T_0 \right) - T_0 \alpha_{\mathrm{p}} \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p_0 \right) &= -\nabla \cdot (-k \nabla T) + Q
\end{aligned}
\tag{5-4}
$$

where the unprimed variables are now the acoustic deviation from the steady state.

The density ρ is expressed in terms of the pressure and the temperature variations using the density differential (Taylor expansion about the steady quiescent values)

$$
\rho = p \left[\frac{\partial \rho}{\partial p} \right]_{T_0} + T \left[\frac{\partial \rho}{\partial T} \right]_{p_0} = \rho_0 (p \beta_{\mathrm{T}} - T \alpha_{\mathrm{p}})
\tag{5-5}
$$

The two thermodynamic quantities (the coefficient terms in square brackets) define the isobaric coefficient of thermal expansion α_{p} (sometimes named α_0) and the isothermal compressibility β_{T}, according to the following relations

$$
\begin{aligned}
\beta_{\mathrm{T}} &= \frac{1}{\rho_0} \left[\frac{\partial \rho_0}{\partial p} \right]_T = \frac{1}{K_{\mathrm{T}}} = \frac{1}{\rho_0 c^2} \frac{\gamma}{} = \gamma \beta_{\mathrm{s}} \\
\gamma &= \frac{C_{\mathrm{p}}}{C_{\mathrm{v}}} = \frac{K_{\mathrm{s}}}{K_{\mathrm{T}}} \\
\alpha_{\mathrm{p}} &= -\frac{1}{\rho_0} \left[\frac{\partial \rho_0}{\partial T} \right]_p
\end{aligned}
\tag{5-6}
$$

where K_s is the isentropic bulk modulus (sometimes named K_0), K_T the isothermal bulk modulus, C_v is the heat capacity at constant volume (per unit mass), c is the (isentropic) speed of sound, and γ is the ratio of specific heats (the adiabatic index). The isothermal compressibility β_T is related to the isentropic (or adiabatic) compressibility β_s (sometimes named β_0) and the coefficient of thermal expansion α_p via the thermodynamic relations

$$\beta_s = \beta_T - \frac{\alpha_p^2 T_0}{\rho_0 C_p} \qquad \beta_T = \gamma\beta_s \tag{5-7}$$

It is derived using the Maxwell relations; see, for example, Ref. 5 and Ref. 7.

From Equation 5-6 and Equation 5-7 the isothermal compressibility and the isobaric coefficient of thermal expansion can be expressed in terms of the speed of sound as

$$\beta_T = \frac{1}{\rho_0 c^2}\frac{\gamma}{} \qquad \alpha_p = \frac{1}{c}\sqrt{\frac{C_p(\gamma-1)}{T_0}} \tag{5-8}$$

The equations presented in Equation 5-4 and Equation 5-5 are the ones solved in the time domain in the The Thermoviscous Acoustics, Transient Interface. Assuming small harmonic oscillations about a steady-state solution, the dependent variables and sources can be assumed to take on the following form

$$\begin{aligned} \mathbf{u} &= \mathbf{u}_0 + \mathbf{u}_1 e^{i\omega t} & p &= p_0 + p_1 e^{i\omega t} \\ T &= T_0 + T_1 e^{i\omega t} & \rho &= \rho_0 + \rho_1 e^{i\omega t} \\ \mathbf{F} &= \mathbf{F}_0 + \mathbf{F}_1 e^{i\omega t} & Q &= Q_0 + Q_1 e^{i\omega t} \end{aligned}$$

Inserting this into the governing equations and performing the linearization yield the equations solved in the frequency domain in the The Thermoviscous Acoustics, Frequency Domain Interface:

$$\begin{gathered} i\omega\rho + \nabla\cdot(\rho_0\mathbf{u}) = 0 \\ i\omega\rho_0\mathbf{u} = \nabla\cdot\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right] + \mathbf{F} \\ \rho_0 C_p(i\omega T + \mathbf{u}\cdot\nabla T_0) - T_0\alpha_p(i\omega p + \mathbf{u}\cdot\nabla p_0) = -\nabla\cdot(-k\nabla T) + Q \\ \rho = \rho_0(p\beta_T - T\alpha_p) \end{gathered} \tag{5-9}$$

The system of equations implemented in the thermoviscous acoustics interfaces is further given in a scattered field formulation, as described below in Scattered Field Formulation and Background Acoustic Fields.

IDEAL GAS

For an ideal gas, the equation of state $p = \rho RT$, where R is the specific gas constant, leads to

$$\beta_T = \frac{1}{p_0} \qquad \alpha_0 = \frac{1}{T_0}$$

and the density

$$\rho = \rho_0\left(\frac{p}{p_0} - \frac{T}{T_0}\right)$$

Inserting these expressions and dividing the continuity equation by the reference density, Equation 5-9 in the frequency domain take on the following simplified form

$$i\omega\rho_0\mathbf{u} = \nabla\cdot\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_{\mathrm{B}}\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right]$$

$$i\omega\left(\frac{p}{p_0} - \frac{T}{T_0}\right) + \nabla\cdot\mathbf{u} = 0$$

$$i\omega(\rho_0 C_{\mathrm{p}} T - p) + \mathbf{u}\cdot\nabla p_0 + \rho_0 C_{\mathrm{p}}(\mathbf{u}\cdot\nabla T_0) = -\nabla\cdot(-\mathrm{k}\nabla T) + Q$$

This is, for example, the system of equations implemented in the Thermoviscous Acoustics, Frequency Domain interface when the ideal gas law is selected.

ISENTROPIC (ADIABATIC) CASE

If the process is assumed to be adiabatic and reversible — that is, isentropic — the thermal conductivity is effectively zero. Then also the temperature can be eliminated, giving for an ideal gas:

$$\begin{aligned} i\omega\rho_0\mathbf{u} &= \nabla\cdot\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_{\mathrm{B}}\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right] \\ i\omega\left(\frac{1}{p_0} - \frac{1}{\rho_0 C_{\mathrm{p}} T_0}\right)p + \nabla\cdot\mathbf{u} &= 0 \end{aligned} \tag{5-10}$$

where we have assumed constant background properties. Defining the speed of sound c in analogy with the standard assumptions for linear acoustics (term in front of the pressure in the continuity equation), it is found that

$$\frac{1}{\rho_0 c^2} = \frac{1}{p_0} - \frac{1}{\rho_0 C_{\mathrm{p}} T_0} = \frac{1}{p_0}\left(1 - \frac{R}{C_{\mathrm{p}}}\right) = \frac{1}{\gamma p_0}$$

or

$$c = \sqrt{\frac{\gamma p_0}{\rho_0}}$$

In the case with a general fluid, the corresponding relation is using Equation 5-6 and Equation 5-7:

$$\frac{1}{\rho_0 c^2} = \frac{1}{K_0} = \frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial p}\right]_T - \frac{T_0}{\rho_0 C_p}\left(\frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial T}\right]_p\right)^2 = \frac{1}{K_T} - \frac{T_0 \alpha_p^2}{\rho_0 C_p}$$

where K_0 is the adiabatic bulk modulus, K_T the isothermal bulk modulus, and α_0 the coefficient of thermal expansion.

ISOTHERMAL CASE

If, on the other hand, the thermal conductivity is high, or the thermoviscous acoustic waves propagate in a narrow space between highly conductive walls, the temperature can be assumed to be constant (isothermal assumption) and the system of equations for an ideal gas becomes:

$$i\omega\rho_0 \mathbf{u} = \nabla \cdot \left[-p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla \cdot \mathbf{u})\mathbf{I}\right]$$

$$i\omega\left(\frac{1}{p_0}\right)p + \nabla \cdot \mathbf{u} = 0$$

which, again comparing to standard assumptions, gives

$$\frac{1}{\rho_0 c^2} = \frac{1}{p_0}$$

or equivalently

$$c = \sqrt{\frac{p_0}{\rho_0}}$$

Therefore, thermal conductivity and/or conducting walls decrease the apparent speed of sound in narrow domains.

THE HELMHOLTZ EQUATION

If the thermodynamic processes in the system are assumed to be adiabatic and viscosity can be neglected Equation 5-10 reduces, for constant background properties, to

$$i\omega\rho_0\mathbf{u} = \nabla\cdot(-p\mathbf{I}) = \nabla p$$

$$i\omega\frac{1}{\rho_0 c^2}p + \nabla\cdot\mathbf{u} = 0$$

Now, taking the divergence of the momentum equations and inserting the expression for the divergence of the velocity, taken from the continuity equations, yields the Helmholtz equation for constant material properties:

$$\nabla^2 p + \frac{\omega^2}{c^2}p = 0$$

Scattered Field Formulation and Background Acoustic Fields

The governing equations, given above in Equation 5-6, are defined in the general scattered field formulation. Assuming that all the acoustic fields are the sum of a background (prescribed) field and the resulting scattered field, the total fields can be written as

$$p_t = p_s + p_b \qquad \mathbf{u}_t = \mathbf{u}_s + \mathbf{u}_b \qquad T_t = T_s + T_b$$
$$p \equiv p_s \qquad \mathbf{u} \equiv \mathbf{u}_s \qquad T \equiv T_s$$

where subscript `t` stands for total, subscript `s` for scattered, and subscript `b` for background. The scattered field is the field solved for, that is the dependent variables p, $\mathbf{u}$, and T. If no background acoustic field is defined (the default) the scattered field is equal to the total field. The governing equations and all boundary conditions are expressed in therms of the total fields. The equations solved for are thus, in general, for the frequency domain case given by

$$\begin{gathered}
i\omega\rho_t + \nabla\cdot(\rho_0\mathbf{u}_t) = 0 \\
i\omega\rho_0\mathbf{u}_t = \nabla\cdot\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right] \\
\rho_0 C_p(i\omega T_t + \mathbf{u}_t\cdot\nabla T_0) - T_0\alpha_p(i\omega p_t + \mathbf{u}_t\cdot\nabla p_0) = -\nabla\cdot(-k\nabla T_t) + Q \\
\rho_t = \rho_0(\beta_T p_t - \alpha_p T_t)
\end{gathered} \tag{5-11}$$

The background acoustic fields are defined by adding the Background Acoustic Fields feature. The feature has a user defined option where all fields can be entered, either as analytical expressions or defined in terms of a solution to another study or model. The background acoustic fields feature also has a plane-wave option (in the frequency domain interface) that defines a plane propagating attenuated wave.

PLANE WAVE SOLUTION

The plane wave option in the Background Acoustic Fields feature (in the frequency domain only) defines the following fields

$$
\begin{aligned}
p_\mathrm{b} &= |p_\mathrm{b}| e^{-k_\mathrm{b}(\mathbf{n}_k \cdot \mathbf{x})} \\
\mathbf{u}_\mathrm{b} &= \frac{\omega(\beta_\mathrm{T} p_\mathrm{b} - \alpha_\mathrm{p} T_\mathrm{b})}{k_\mathrm{b}} \mathbf{n}_k \\
T_\mathrm{b} &= \frac{i\omega\alpha_\mathrm{p} T_0 p_\mathrm{b}}{i\omega\rho_0 C_p + \mathrm{k} k_\mathrm{b}^2}
\end{aligned}
\qquad (5\text{-}12)
$$

$$
k_\mathrm{b} = \frac{\omega}{c}\left(1 + \frac{i\omega b_\mathrm{tv}}{\rho_0 c^2}\right)^{\frac{1}{2}} \qquad b_\mathrm{tv} = \frac{4}{3}\mu + \mu_\mathrm{B} + \frac{(\gamma - 1)\mathrm{k}}{C_\mathrm{p}}
$$

where k_b is the is the wave number of a plane propagating wave (in the direction $\mathbf{n}_k$) with viscous and thermal attenuation, see Ref. 3. The expressions given in Equation 5-12 are an exact solution to the governing equations Equation 5-11.

Formulation for Eigenfrequency Studies

When performing an eigenfrequency study the governing equations (Equation 5-6) are on the form:

$$
\begin{gathered}
-\lambda\rho + \nabla \cdot (\rho_0 \mathbf{u}) = 0 \\
-\lambda\rho_0 \mathbf{u} = \nabla \cdot \left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^\mathrm{T}) - \left(\frac{2\mu}{3} - \mu_\mathrm{B}\right)(\nabla \cdot \mathbf{u})\mathbf{I}\right] \\
-\rho_0 C_\mathrm{p}(\lambda T_\mathrm{t} + \mathbf{u}_\mathrm{t} \cdot \nabla T_0) + T_0 \alpha_\mathrm{p}(\lambda p_\mathrm{t} + \mathbf{u}_\mathrm{t} \cdot \nabla p_0) = -\nabla \cdot (-\mathrm{k}\nabla T) + Q \\
\rho = \rho_0(\beta_\mathrm{T} p - \alpha_\mathrm{p} T)
\end{gathered}
$$

where the eigenvalue is $\lambda = -i\omega$. It is important to note that there is a difference between regular pressure acoustics and thermoviscous acoustics in terms of what modes can exist and which modes are found during an eigenfrequency study. In pressure acoustics only the pure acoustic modes exist; here the equations and

assumptions made ensure this. In thermoviscous acoustics, on the other hand, the equations are formulated for all small signal components that can exist. This means that other nonacoustic modes also exist, that is, thermal and vorticity modes.

VORTICITY AND THERMAL MODES

When solving an eigenfrequency problem in thermoviscous acoustics, it is important to take a close look at the obtained eigenfrequencies and assess if they are acoustic or not. The nature of the solution is of the form

$$p(\mathbf{x},t) \sim p(\mathbf{x})e^{-\lambda t} = p(\mathbf{x})e^{i\omega t} \qquad \lambda = \alpha + i\beta \qquad \omega = -\beta + i\alpha$$

where λ is the eigenvalue. Typically, eigenvalues exist near the positive real axis, where $\beta \approx 0$. These are exponentially decaying nonacoustic (nonoscillating) modes that stem from the thermal equation or the deviatoric part of the momentum equation (the nonpressure and nonvolume part of the stress tensor) also called the vorticity modes. The acoustic eigenvalues on the other hand lie close to the imaginary axis and are oscillating and slightly damped.

OTHER SPURIOUS MODES

Note that other spurious and nonacoustic modes can also exist when, for example, a PML layer is used to model an open boundary. These modes stem from nonphysical phenomena and the scaling inside the PML layer. In all cases it is a good idea to have an a priori knowledge of the location/type of the eigenvalues, maybe from solving an lossless pressure acoustics model, and also to look at the modes in terms of, for example, the pressure field.

Formulation for Mode Analysis in 2D and 1D axisymmetric

The Mode Analysis study type is available for The Thermoviscous Acoustics, Frequency Domain Interface in 2D and 1D axisymmetric models. This type of study is used to determine the form of the *propagating acoustic modes* in *waveguide* structures. The analyzed 2D and 1D axisymmetric geometries can be thought of as the cross sections of a waveguide. This study is really restricted to the cross section of the waveguides. To calculate the modes in 3D structures at the inlet of the waveguide use the The Thermoviscous Acoustics, Boundary Mode Interface, described below.

The spatial dependency in the (out-of-plane) axial z-direction along the waveguide is assumed to be of the form of a traveling wave with wave number k_z. The dependent variables in 2D are rewritten as

$$p = p(x,y)e^{-ik_z z}$$
$$\mathbf{u} = \mathbf{u}(x,y)e^{-ik_z z}$$
$$T = T(x,y)e^{-ik_z z}$$

and in 1D axisymmetric as

$$p = p(r)e^{-ik_z z}$$
$$\mathbf{u} = \mathbf{u}(r)e^{-ik_z z}$$
$$T = T(r)e^{-ik_z z}$$

Using this form of the dependent variables, differentiation with respect to z reduces to a multiplication with $-ik_z$. The *propagating modes* are determined by solving an eigenvalue problem in the variable $\lambda = -ik_z$.

The expression for the pressure can now be written retaining the harmonic time dependency, as

$$k_z = \beta + i\alpha$$
$$p = p(\mathbf{x})e^{-ik_z z}e^{i\omega t} = p(\mathbf{x})e^{\alpha z}e^{i(\omega t - \beta z)}$$

where $\mathbf{x}$ is the in plane coordinate(s). The axial wave number is split into a real and an imaginary part. The imaginary part α of the wave number describes how fast the propagating modes decay along the waveguide, it is often referred to as the attenuation coefficient. The real part β is related to the phase speed c_{ph} of the propagating mode by $c_{ph} = \beta/\omega$. In thermoviscous acoustics, the obtained wave numbers always have an imaginary part as the modeled system always includes losses. The relation between the angular frequency ω and the axial wave number k_z is called the dispersion relation.

- Mode Analysis in the *COMSOL Multiphysics Reference Manual*

Formulation for the Boundary Mode Interface

The Thermoviscous Acoustics, Boundary Mode Interface interface adds extended functionality to the above mentioned mode analysis available in 2D and 1D axisymmetric. The interface can be applied on boundaries in 3D (and 2D axisymmetric) and used to compute the propagating modes and out-of-plane wave

number k_n on a (flat) surface of any orientation in 3D. This makes it possible to more easily set up boundary conditions at inlets of waveguides using the mode information.

The equations solved are the same as for The Thermoviscous Acoustics, Frequency Domain Interface but using modified expressions for the gradients. We assume propagation in the out-of-plane direction, that is, in the normal direction $\mathbf{n}$, and the fields can then we written as

$$p = p(\mathbf{x}_{ip})e^{-ik_n\mathbf{x}_{op}}$$
$$\mathbf{u} = \mathbf{u}(\mathbf{x}_{ip})e^{-ik_n\mathbf{x}_{op}}$$
$$T = T(\mathbf{x}_{ip})e^{-ik_n\mathbf{x}_{op}}$$

where $\mathbf{x}_{ip}$ is the in-plane coordinate, $\mathbf{x}_{op}$ is the out-of-plane coordinate, and k_n is the wave number we are seeking. Because of the above assumption about the acoustic fields, the gradient operator can be split into a tangential in-plane component ($||$) and an normal out-of-plane component ($\perp$), such that

$$\nabla p = \nabla_{||} p - ik_n p\mathbf{n}$$
$$\nabla \mathbf{u} = \nabla_{||} \mathbf{u} - ik_n(\mathbf{u} \cdot \mathbf{n}^T)$$
$$\nabla T = \nabla_{||} T - ik_n T\mathbf{n}$$

The in-plane gradient is defined in COMSOL Multiphysics by the tangential derivative operator `dtang()`. The divergence of the velocity field is defined as the trace of the Jacobian, that is

$$\nabla \cdot \mathbf{u} = \text{trace}(\nabla \mathbf{u})$$

With the above modifications and redefinition of the gradient operators, solving for the propagating modes reduces to an eigenvalue problem solving for the acoustic field and the out-of-plane wave number for a given frequency.

References for the Thermoviscous Acoustics, Frequency Domain Interface

1. W.M. Beltman, P.J.M. van der Hoogt, R.M.E.J. Spiering, and H. Tijdeman, "Implementation and Experimental Validation of a New Viscothermal Acoustic Finite Element for Acousto-Elastic Problems," *J. Sound and Vibration*, vol. 216, pp. 159–185, 1998.

2. M. Malinen, M. Lyly, and others, "A Finite Element Method for the Modeling of Thermo-Viscous Effects in Acoustics," *Proc. ECCOMAS 2004*, Jyväskylä, 2004.

3. D.T. Blackstock, *Fundamentals of Physical Acoustics,* John Wiley & Sons, 2000.

4. H. Bruus, *Theoretical Microfluidics*, Oxford University Press, 2010.

5. G.K. Bachelor, *An Introduction to Fluid Dynamics*, Cambridge University Press, 2000.

6. M. K. Myers, "Transport of energy by disturbances in arbitrary steady flows," J. Fluid Mech., vol. 226, pp. 383–400, 1991.

7. B. Lautrup, *Physics of Continuous Matter, Exotic and Every Day Phenomena in the Macroscopic World,* 2nd ed., CRC Press, 2011.

8. A. S. Dukhin and P. J. Goetz, "Bulk viscosity and compressibility measurements using acoustic spectroscopy," J. Chem. Phys., vol. 130, pp. 124519-1, 2009.

9. M. S. Cramer, "Numerical estimate for the bulk viscosity of ideal gases," Phys. Fluids, vol. 30, pp. 066102, 2012,

6

Ultrasound Interfaces

This chapter describes the physics interfaces found under the **Acoustics>Ultrasound** branch ()

The Convected Wave Equation, Time Explicit Interface

The **Convected Wave Equation, Time Explicit (cwe)** interface ( ), found under the **Acoustics > Ultrasound** subbranch ( ) when adding a physics interface, is used to solve large transient linear acoustic problems containing many wavelengths in a stationary background flow. It is suited for time-dependent simulations with arbitrary time-dependent sources and fields. In general, the interface is suited for modeling the propagation of acoustic signals over large distances relative to the wavelength, for example, linear ultrasound problems. The interface includes absorbing layers that are used to set up effective nonreflecting like boundary conditions. The interface exists in 2D, 2D axisymmetric, and 3D.

The interface is based on the discontinuous Galerkin (DG or DG-FEM) method and uses a time explicit solver. The method is very memory efficient and can solve problems with many million degrees of freedom (DOFs). Application areas include ultrasound flow meters and other ultrasound sensors where time of flight is an important parameter. The applications are not restricted to ultrasound, but also include, for example, transient propagation of audio pulses in room acoustics or car cabins.

The interface solves the linearized Euler equations assuming an adiabatic equation of state. The dependent variables are the acoustic pressure p and the acoustic velocity perturbation $\mathbf{u}$. The background mean flow can be any stationary flow with small to moderate velocity gradients. No physical loss mechanisms are included in the interface.

When solving models that are based on the DG method, optimizing the mesh is important. For further details see Optimizing the Mesh for DG in the Meshing, Discretization, and Solvers section.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Convected Wave Equation Model**, **Sound Hard Wall**, **Initial Values**, and **Axial Symmetry** (in case of 2D axisymmetric). Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Convected Wave Equation, Time Explicit** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `cwe`.

FILTER PARAMETERS FOR ABSORBING LAYERS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. In the **Filter Parameters for Absorbing Layers** section you can change and control the values set for the filter used in the Absorbing Layers. The values of the filter parameters defined here are used in all absorbing layers added to the model and they override the value of filter parameters enabled in the material model (Convected Wave Equation Model). The default values of the filter parameters α, η_c, and s are set to 0.1, 0.01, and 2, respectively. Inside the absorbing layer it is important to use a filter that is not too aggressive since this will result in spurious reflections.

For general information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Guide*.

EQUATION SETTINGS (2D AXISYMMETRIC ONLY)

This section is only available in **2D Axisymmetric**. Per default the acoustic velocity perturbation $\mathbf{u}$ is assumed to have only two components: the radial component u_r, and the axial component u_z. You can select the **Enable out-of-plane components** check box to enable the third — circumferential component u_φ, which physically can only be non-zero, if the background mean flow has the corresponding component.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. In this section you can select the discretization for the **Acoustic pressure** and **Acoustic velocity**. Per default both are set to **Quartic** (4th order). Using quartic elements together with a mesh size equal to approximately half the wavelength to be resolved, leads to the best performance when using the DG method. For further details see the Meshing, Discretization, and Solvers section.

DEPENDENT VARIABLES

The dependent variables (field variables) are the **Acoustic pressure**, **Acoustic velocity**, and **Acoustic velocity**, **components**. The names can be changed, but the names of fields and dependent variables must be unique within a model.

- For information about modeling strategies, meshing, the absorbing layer and more, see Modeling with the Convected Wave Equation Interface.
- For the theoretical background of the model solved, see Theory for the Convected Wave Equation Interface.

- *Gaussian Pulse in 2D Uniform Flow: Convected Wave Equation and Absorbing Layers*. The Application Library path: **Acoustics_Module/Tutorials/gaussian_pulse_absorbing_layers**
- *Ultrasound Flow Meter with Generic Time-of-Flight Configuration*. The Application Library path: **Acoustics_Module/Ultrasound/ultrasound_flow_meter_generic**
- *Ultrasonic Flow Meter with Piezoelectric Transducers: Coupling between FEM and DG*. The Application Library path: **Acoustics_Module/Ultrasound/flow_meter_piezoelectric_transducers**

Domain, Boundary, Edge, Point, and Pair Nodes for the Convected Wave Equation Interface

The Convected Wave Equation, Time Explicit Interface has these domain, boundary, edge, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users). Continuity in the total pressure and velocity is the default condition on interior boundaries.

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Acoustic Impedance
- Convected Wave Equation Model
- Domain Sources
- General Flux/Source
- Initial Values
- Normal Velocity
- Pressure
- Sound Hard Wall
- Symmetry

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Convected Wave Equation Model

The **Convected Wave Equations** node adds the equations for modeling the propagation of acoustic waves in a stationary background flow. Adiabatic behavior is assumed, meaning that this in some sense represents pressure acoustics in the presence of flow. The convected wave equation model solves the linearized Euler equations also referred to as linear acoustic equations for moving media. The equations are valid for any stationary background mean flow as long as there are not too large gradients in the background properties.

The linear continuity equation, momentum equation, and equation of state solved are given by:

$$\frac{\partial \rho}{\partial t} + (\mathbf{u}_0 \cdot \nabla)\rho + (\mathbf{u} \cdot \nabla)\rho_0 + \rho(\nabla \cdot \mathbf{u}_0) + \rho_0(\nabla \cdot \mathbf{u}) = f_\mathrm{p}$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}_0 \cdot \nabla)\mathbf{u} + (\mathbf{u} \cdot \nabla)\mathbf{u}_0 + \frac{1}{\rho_0}\nabla p - \frac{\rho}{\rho_0^2}\nabla p_0 = \mathbf{f}_\mathrm{v}$$

$$\rho = \frac{p}{c_0^2}$$

where $\mathbf{u}_0$ is the background mean flow velocity, p_0 is the background mean flow pressure, ρ_0 is the background mean flow density, and c_0 is the speed of sound. All background properties can vary in space. The right-hand side sources f_p and $\mathbf{f}_\mathrm{v}$ can be defined by the Domain Sources. More information about the governing equations is given in the Theory for the Convected Wave Equation Interface section.

In the **Settings** window, define the properties for the acoustics model and model inputs including the background mean flow, pressure, and velocity.

MODEL INPUTS

In order to model the influence the background mean flow has on the propagation of the acoustic waves in the fluid, the **Background mean flow pressure** p_0 and **Background mean flow velocity** $\mathbf{u}_0$ need to be defined. If a material that is temperature dependent is used the **Background mean flow temperature** T_0 field is also present.

All the background mean flow parameters can be functions of space. They can be either analytical expressions (user defined) or they can be picked up from a flow simulation performed using the CFD Module. In this case, for the pressure and velocity, select, for example, *Absolute pressure (spf)* and *Velocity field (spf)* from the list. By default they are set to the quiescent constant background conditions of air.

FLUID PROPERTIES

In this section, you define the background mean flow density ρ_0 (SI unit: $\mathrm{kg/m^3}$) and the background mean flow speed of sound c_0 (SI unit: m/s). The default is to use the property values from the material (**From material**). Select **User defined** from the list to enter a user-defined value in the text field that appears. The density can also be picked up from a flow simulation, when a CFD physics interface is present in the model. In this case select, for example, *Density (spf/fp1)* from the list.

LAX-FRIEDRICHS FLUX PARAMETER

To display this section, click the **Show** button ( ) and select **Stabilization**. In this section, you specify the value of the *Lax-Friedrichs flux parameter* τ_{LF} (default value: 0.2). This value controls the numerical flux between the elements (nodal discontinuous Lagrange elements) used with the discontinuous Galerkin (DG) method. The numerical flux defines how adjacent elements are connected and how continuous p and $\mathbf{u}$ are. Different definitions of the numerical flux lead to different variants of the DG method. The flux implemented here is the so-called global Lax-Friedrichs numerical flux. The value of the parameter τ_{LF} should be between 0 and 0.5. For $\tau_{LF} = 0$ a so-called central flux is obtained. Setting $\tau_{LF} = 0.5$ gives a maximally dissipative global Lax-Friedrich flux.

For general information about the numerical flux see the Numerical Flux section under Wave Form PDE in the *COMSOL Multiphysics Reference Guide*.

FILTER PARAMETERS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. By default, the filter parameters α, η_c, and s are not active. Select the **Activate** check box to activate the filter. The filter provides higher-order smoothing for the DG formulation and can be used to stabilize the solution, for example, when a large background flow is present or large gradients are present. Inside absorbing layers the settings given here are overridden by the Filter Parameters for Absorbing Layers.

Enter values for the filter parameters in the corresponding text fields (default values: 36, 0.6, and 3). α must be positive and lie between 0 and 36. $\alpha = 0$ means no dissipation and $\alpha = 36$ means maximal dissipation. η_c should be between 0 and 1, where 0 means no filtering and 1 means maximal filtering. The s parameter should be larger than 0 and controls the order of the filtering (a dissipation operator of order $2s$). For $s = 1$, you get a filter that is related to the classical 2nd-order Laplacian. A larger s gives a more pronounced low-pass filter.

For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Guide*.

Domain Sources

Use a **Domain Sources** node to add various sources to a domain. The feature adds right-hand side sources to either the continuity or the momentum equation.

The **Domain mass source** q_{source} and the **Domain pressure source** p_{source} both contribute to the source term f_{p} of the continuity equation. This is a monopole like source that acts uniformly in all directions. The two represent different formulations of the same source type.

$$f_{\text{p}} = q_{\text{source}} + \frac{1}{c_0^2}\frac{\partial p_{\text{source}}}{\partial t}$$

The **Domain velocity source** $\mathbf{u}_{\text{source}}$ and **Domain force source** $\mathbf{f}_{\text{source}}$ both contribute to the source term $\mathbf{f}_{\text{v}}$ of the momentum equation. This is a dipole like source that acts in the direction of its vector. The two represent different formulations of the same source type.

$$\mathbf{f}_v = \rho_0 \frac{\partial \mathbf{u}_{source}}{\partial t} + \mathbf{f}_{source}$$

All source terms can be any analytical or interpolation function of the time. The variable `t` is reserved to represent time in expressions.

DOMAIN SOURCES

In this section, add one or more of the following sources:

- Add a **Domain mass source** q_{source} (SI unit: kg/(m^3·s))
- Add a **Domain pressure source** p_{source} (SI unit: Pa)
- Add a **Domain velocity source** $\mathbf{u}_{source}$ (SI unit: m/s).
- Add a **Domain force source** $\mathbf{f}_{source}$ (SI unit: N/m^3).

Sound Hard Wall

The **Sound Hard Wall** adds a boundary condition for a sound hard boundary or wall, which is a boundary at which the normal component of the velocity is zero (a slip condition). This is assumed true for both the acoustic velocity and the background velocity

$$-\mathbf{n} \cdot \mathbf{u} = 0 \qquad -\mathbf{n} \cdot \mathbf{u}_0 = 0$$

Note that this condition is not identical to the Symmetry condition where only the normal component of the acoustic velocity is assumed to be zero.

Initial Values

The **Initial Values** node adds initial values for the acoustic pressure and the acoustic velocity that can serve as an initial guess/state for the solver. If more than one initial value is needed, from the **Physics** toolbar click to add more **Initial Values** nodes.

INITIAL VALUES

Enter a value or expression for the initial values for the **Pressure** p (SI unit: Pa) and the **Acoustic velocity**, $\mathbf{u}$ (SI unit: m/s).

Normal Velocity

The **Normal Velocity** adds an inward normal velocity either given as a scalar v_n or as a velocity vector $\mathbf{v}_b$, the condition given is

$$-\mathbf{n} \cdot \mathbf{u} = v_{\mathrm{n}}(t) = -\mathbf{n} \cdot \mathbf{v}_{\mathrm{b}}(t)$$

where **n** is the surface normal. Both expressions should be functions of time. This feature represents an external source term. It can also be used to model a vibrating transducer surface or other vibrating surfaces.

NORMAL VELOCITY

Select a **Type** — **Inward Velocity** (the default) or **Velocity**.

- For **Inward Velocity** enter an expression for the **Inward velocity** $v_{\mathrm{n}}(t)$ (SI unit: m/s). Use a positive value for inward velocity or a negative value for outward velocity.
- For **Velocity** enter expressions for the components of the **Velocity** $\mathbf{v}_{\mathrm{b}}(t)$ (SI unit: m/s).

Pressure

The **Pressure** node creates a boundary condition that acts as a pressure source at the boundary. The acoustic pressure is given at the boundary $p = p_b(t)$.

PRESSURE

Enter the an expression for the **Boundary pressure** p_{b} (SI unit: Pa).

Symmetry

The **Symmetry** node adds a boundary condition imposing symmetry in the pressure and velocity field. Use this condition to reduce the size of a model by cutting it in half where there are known symmetries. The condition implies symmetry in the acoustic fields

$$-\mathbf{n} \cdot \mathbf{u} = 0$$

Acoustic Impedance

The **Acoustic Impedance** node adds a boundary condition defining a relation between the local acoustic pressure p and the normal acoustic velocity $\mathbf{n}\cdot\mathbf{u}$, that is, the acoustic impedance Z. The condition specifies the inward normal velocity

$$-\mathbf{n} \cdot \mathbf{u} = -\frac{p}{Z}$$

This condition can be used to model the properties of artificial boundaries. When the impedance is set equal to the characteristic specific impedance of a propagating plane wave $\rho_0 c_0$, the condition represent the simplest nonreflecting boundary condition. This is also the default value of the impedance when the impedance condition is added.

Note that this condition is not equivalent to the general impedance condition, as given by Myers (see Ref. 4), when a flow is present. This means that, for example, it cannot be used at a reacting surface which has a tangential flow. For this kind of detailed conditions a frequency domain representation is needed.

The acoustic impedance condition (with the default value $\rho_0 c_0$) should be used at the outer boundary when Absorbing Layers are used in a model.

ACOUSTIC IMPEDANCE

Enter a value or expression for the **Acoustic Impedance** Z (SI unit: Pa·s/m). The default expression is `cwe.c0*cwe.rho0` which is the characteristic specific impedance $\rho_0 c_0$ of a propagating plane wave.

General Flux/Source

The **General Flux/Source** node, found under the **More** submenu, adds the most general boundary condition for the convected wave equation, as it is formulated (in the conservative form) for the discontinuous Galerkin method (DG). The condition defines the normal flux $\mathbf{g}$ at an exterior boundary, given by

$$-\mathbf{n}\cdot\Gamma = \begin{bmatrix} \rho_0(\mathbf{n}\cdot\mathbf{u}) + \frac{p}{c_0^2}(\mathbf{n}\cdot\mathbf{u}_0) \\ \frac{p}{c_0^2}u_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(u(\mathbf{n}\cdot\mathbf{u}_0) + u_0(\mathbf{n}\cdot\mathbf{u})) + n_1 p \\ \frac{p}{c_0^2}v_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(v(\mathbf{n}\cdot\mathbf{u}_0) + v_0(\mathbf{n}\cdot\mathbf{u})) + n_2 p \\ \frac{p}{c_0^2}w_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(w(\mathbf{n}\cdot\mathbf{u}_0) + w_0(\mathbf{n}\cdot\mathbf{u})) + n_3 p \end{bmatrix} = \mathbf{g}$$

where $\mathbf{u} = (u, v, w)$, $\mathbf{u}_0 = (u_0, v_0, w_0)$, and $\mathbf{n} = (n_x, n_y, n_z)$ ($\mathbf{n} = (n_r, n_\varphi, n_z)$ in 2D axisymmetric) is the surface normal. Care should be taken when using this advanced condition as the method is sensitive to proper formulation of boundary conditions. Only one characteristic can enter the domain at any boundary at the time, meaning that it is easy to over-specify the problem. Use the mesh normals (`nxmesh`, `nymesh`,

and nzmeshor, respectively, nrmesh, nphimesh, and nzmesh) in the expression you define.

GENERAL FLUX/SOURCE

Enter the expression for the components of the **Flux vector g**.

General Interior Flux

The **General Interior Flux** node, found under the **More** submenu, adds a the most general interior boundary condition for the convected wave equation, as it is formulated (in the conservative form) for the discontinuous Galerkin method (DG). The condition defines the normal flux **g** on an interior boundary by

$$-\mathbf{n} \cdot \Gamma^* = \mathbf{g}$$

here Γ^* represents the total numerical flux. This means that if this condition is added on an interior boundary it overrides the existing Lax-Friedrich numerical flux. So care should be taken when specifying this condition as it needs to include the necessary numerical flux contribution to ensure stability of the method.

On an interior boundary you can use the up() and down() operators to access values from both sides of the boundary. If a dependent variable is used in this expression (without up() or down() operators) an implicit mean() operation is invoked taking the average of the up and down side values. Use the mesh normals (nxmesh, nymesh, and nzmesh or, respectively, nrmesh, nphimesh, and nzmesh) in the expression you define.

GENERAL INTERIOR FLUX

Enter the expression for the components of the **Flux vector g**.

Modeling with the Convected Wave Equation Interface

This section contains information about:

- Meshing, Discretization, and Solvers
- Postprocessing: Variables and Quality
- Absorbing Layers
- Stabilizing Physical Instabilities (Filtering)
- Storing Solution on Selections for Large Models

Meshing, Discretization, and Solvers

The Convected Wave Equation, Time Explicit Interface (CWE) is based on the discontinuous Galerkin method also known as DG-FEM or simply DG. The method is very memory efficient and is based on a time explicit formulation. This means that it is not necessary to invert a full system matrix when stepping forward in time. Inversion of this matrix is necessary in time implicit methods and is very memory consuming for large problems. Because the CWE interface is not based on the classical FEM formulation, used in most of the other acoustics interfaces, other strategies apply for meshing and discretization.

The internal time stepping size of a time explicit method is strictly controlled by the CFL condition and thus the mesh size. Meaning that the smallest mesh elements will restrict the time steps (see Optimizing the Mesh for DG below). It turns out that the DG formulation has a sweet spot for speed and efficiency for wave problems. This is achieved by using fourth order (quartic) shape functions (the default in the interface) and a mesh with the element size of about half the wavelength of the highest frequency component that needs to be resolved. In practice a mesh with size set to $\lambda_{min}/2$ to $\lambda_{min}/1.5$ can usually be used.

The DG method can only use triangular mesh elements in 2D and 2D axisymmetric and tetrahedral elements in 3D. It is recommended to avoid mesh elements with short edges, since these are bad for the DG method.

When solving, the internal time step, used by the solver, is automatically calculated based on mesh, the wave speed and background mean flow speed, and the specifics of

the solver selected. By default the time explicit solver uses the Runge-Kutta 4th order the method (RK4). This method is good as long as the mesh has a uniform mesh element size. If a mesh includes a large distribution of mesh element sized it can be advantageous to switch to the Adam-Bashforth 3 (local) method (AB3loc). This method uses intermediate local time steps in the region where there are small mesh elements.

For general information about the two methods see the The Time Explicit Solver Algorithms in the *COMSOL Multiphysics Reference Guide*.

OPTIMIZING THE MESH FOR DG

As mentioned above, the internal time step, used by the solver, is controlled by the smallest mesh element in the model. So when meshing care should be taken to avoid small elements. Some important strategies apply:

1. Always use the **Avoid too small elements** option in the **Element Quality Optimization** settings on the **Free Tetrahedral** nodes in the mesh (in 3D only). The **Optimization Level** can be set to different degrees **Basic**, **Medium**, or **High**. Using this option can greatly improve the mesh for DG applications and thus speed up the computation significantly.
2. Avoid small edges and surfaces in the geometry as these control the mesh. Several tools exist to remedy this by using the **Virtual Operations** in the geometry.
3. Curved surfaces and boundaries need to be resolved adequately to ensure numerical stability. At the same time too small elements should of course be avoided.

For general information about optimizing quality see Element Quality Optimization in the *COMSOL Multiphysics Reference Guide*.

For information about virtual operations see Virtual Geometry and Mesh Control Operations in the *COMSOL Multiphysics Reference Guide*.

Postprocessing: Variables and Quality

POSTPROCESSING VARIABLES

Several variables are defined to be used when postprocessing. They include the pressure, velocity components, and the norm of the velocity, as well as material

parameters. The instantaneous intensity also exists as a variable (`cwe.Ii`). It is defined as

$$\mathbf{i} = (\rho_0\mathbf{u} + \rho\mathbf{u}_0)\left(\frac{p}{\rho_0} + \mathbf{u}\cdot\mathbf{u}_0\right)$$

Notice that the instantaneous intensity differs from the usual intensity which is an averaged value.

QUALITY

When analyzing the results from a simulation with the CWE interface, be aware of the fact that fourth order elements are used to discretize the dependent variables (as discussed in the section above). This, in some sense, means that within a mesh element, the shape function has a lot of freedom and can contain a lot of details. These details are revealed by setting a high **Resolution** in the **Quality** section in the plots. The default plots generated already have that option set. If you add more user defined plots, remember to set the resolution.

Absorbing Layers

The Convected Wave Equation, Time Explicit Interface includes so-called *absorbing layers,* also often referred to as *sponge layers*. The layers work by combining three techniques: a scaling system, filtering, and simple nonreflecting conditions. For a review of the method see, for example, Ref. 5.

The layers are set up by adding the **Absorbing Layer** under the **Definitions** node. This adds a special scaled system. The scaling effectively slows down the propagating waves and ensures that they "align up" (normal) towards the outer boundary. This means that the waves will hit the outer boundary in a closer to normal direction. Filtering attenuates and filters out high frequency components of the wave. The filter parameters are controlled under the Filter Parameters for Absorbing Layers section at the main physics level. Finally, at the outer boundary of the layer add a simple Acoustic Impedance condition, which will work well to remove all remaining waves, as normal incidence has been ensured.

For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual.*

The physical thickness of the layer should be such that it contains 2 to 3 mesh elements. This in practice means that it should roughly have a thickness equal to the largest wavelength resolved in the model. This is due to the meshing requirements described in the Meshing, Discretization, and Solvers section. You want to avoid to create small mesh elements in the absorbing layers. These can potentially slow down the solution procedure since the internal time stepping is controlled by the mesh size.

For the **Absorbing Layers** select the **Type** (Cartesian, cylindrical, spherical, or user defined) under the **Geometry** section. Enter values for the **Physical Width** and **Pole Distance** under the **Scaling** section.

For more details on the **Geometry** and **Scaling** see the Infinite Elements, Perfectly Matched Layers, and Absorbing Layers section in the *COMSOL Multiphysics Reference Manual*.

For the layers to work optimally the filter should not be too aggressive. Moreover, the scaled coordinates in the layer domain should also vary smoothly. To inspect the scaled system you can, for example, plot the coordinate variables `x_absorb_ab1`, `y_absorb_ab1`, and `z_absorb_ab1`. In 2D axisymmetric they are `r_absorb_ab1` and `z_absorb_ab1`. Note that the variables are not in the plot menu under the **Replace Expression**.

Using the absorbing layers with the three combined techniques will enable the reduction of spurious reflections by a factor between 100 and 1000 compared to the incident amplitude.

See the *Gaussian Pulse in 2D Uniform Flow: Convected Wave Equation and Absorbing Layers* tutorial model for an example. The Application Library path **Acoustics_Module/Tutorials/gaussian_pulse_absorbing_layers**

Stabilizing Physical Instabilities (Filtering)

In the Convected Wave Equation Model (at the domain level), it is possible to activate a filter similar to the one used in the absorbing layer. This filter can be used to stabilize and suppress physical instabilities that can occur in the solution. These are well known in linearize-Euler-like equations, see Ref. 6. In the presence of a background flow, vorticity waves can occur; these are propagated with the background flow and are not acoustic waves.

Storing Solution on Selections for Large Models

Since the CWE interface is suited for solving large acoustic problems (measured in the number of wavelengths per geometry unit it can handle), the model solved can easily contain many million degrees of freedom (DOFs). Storing the solution with a desired time resolution can thus results in huge data files. To circumvent this, a good practice is to use the **Store fields in output** option found under the **Values of Dependent Variables** section in the main **Time Dependent** solver (for example, in the **Study 1>Step 1: Time Dependent** node). Using the **For selections** option, it is possible to only store data on predefined selections. This can, for example, be on a symmetry plane, along an edge, or at the location of a receiver in a model. When postprocessing the data, best results are obtained by adding the same selections to the data set used.

Theory for the Convected Wave Equation Interface

The Convected Wave Equation, Time Explicit Interface theory is described in this section:

- Governing Equations of the Convected Wave Equation
- Boundary Conditions
- The Lax-Friedrichs Flux
- References for the Convected Wave Equation Interface

Governing Equations of the Convected Wave Equation

The governing equations solved in the Convected Wave Equation (CWE) interface describe the propagation of linear acoustic waves (assuming an adiabatic equation of state) in the presence of a background flow. These equation are derived by Pierce in Ref. 1 (section 8.6) and Ref. 2 and are the equations solved by this interface,

$$\begin{aligned}\frac{\partial \rho}{\partial t} + (\mathbf{u}_0 \cdot \nabla)\rho + (\mathbf{u} \cdot \nabla)\rho_0 + \rho(\nabla \cdot \mathbf{u}_0) + \rho_0(\nabla \cdot \mathbf{u}) &= f_\mathrm{p} \\ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}_0 \cdot \nabla)\mathbf{u} + (\mathbf{u} \cdot \nabla)\mathbf{u}_0 + \frac{1}{\rho_0}\nabla p - \frac{\rho}{\rho_0^2}\nabla p_0 &= \mathbf{f}_\mathrm{v} \\ \rho &= \frac{p}{c_0^2}\end{aligned} \tag{6-1}$$

Pierce argues for the use of the adiabatic assumption for the acoustic processes (perturbations in the entropy, $s = 0$) but also for not retaining the 0th order entropy variable s_0 (background mean flow entropy). The argument is that the entropy s only varies because of variations in the background fields (it is zero in a homogeneous medium). This leads to a term in the momentum equation that is second order in gradients of the background field, for example, $(\partial \mathbf{u}_0/\partial x)^2$ and so forth. These terms are disregarded. This also means that the equations are not valid when these terms are large, meaning when gradients in the background fields are large.

In order to fit into the discontinuous Galerkin (DG) formulation, the governing equations need to be put on a general conservative form of the type

$$\mathbf{d}_a \frac{\partial}{\partial t}\mathbf{U} + \nabla \cdot \Gamma(\mathbf{U}) = \mathbf{S} \tag{6-2}$$

where $\mathbf{U}$ is the vector containing the dependent variables $(p, \mathbf{u})$, $\mathbf{d}_a$ is the mass matrix of the system, Γ is the flux matrix, and $\mathbf{S}$ is the right-hand-side (RHS) source vector. The conservative form of Equation 6-1 is derived as follows. Start with Euler's equations on a conservative form (omitting the RHS). For now, the dependent variables represent the full fields (not the acoustic perturbations). The continuity, momentum, and equation of state can be written

$$\begin{gathered} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \\ \frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (p\mathbf{I} + \rho \mathbf{u}\mathbf{u}^{\mathrm{T}}) = \mathbf{0} \\ \rho = \rho(p) \end{gathered} \tag{6-3}$$

The equations describe the conservation of mass ρ and momentum flux $\rho\mathbf{u}$. Linearize these equations according to the usual scheme, using

$$p = p_0(\mathbf{x}) + p_1(\mathbf{x}, t) \qquad \mathbf{u} = \mathbf{u}_0(\mathbf{x}) + \mathbf{u}_1(\mathbf{x}, t) \qquad \rho = \rho_0(\mathbf{x}) + \rho_1(\mathbf{x}, t)$$

Now, insert these into Equation 6-3 and retain only 1st order terms (the acoustic perturbations)

$$\begin{gathered} \frac{\partial \rho_1}{\partial t} + \nabla \cdot (\rho_1 \mathbf{u}_0 + \rho_0 \mathbf{u}_1) = 0 \\ \rho_0 \frac{\partial \mathbf{u}_1}{\partial t} + \mathbf{u}_0 \frac{\partial \rho_1}{\partial t} + \nabla \cdot (p_1 \mathbf{I} + \rho_1 \mathbf{u}_0 \mathbf{u}_0{}^{\mathrm{T}} + \rho_0 \mathbf{u}_1 \mathbf{u}_0{}^{\mathrm{T}} + \rho_0 \mathbf{u}_0 \mathbf{u}_1{}^{\mathrm{T}}) = \mathbf{0} \\ \rho_1 = \frac{p_1}{c_0^2} \end{gathered} \tag{6-4}$$

This is the conservative form of the equations implemented in the CWE interface. In the remaining of this section, the subscript 1 will be omitted from the acoustic fields. The subscript 0 is kept on the variables that represent the background mean properties.

Equation 6-4 can now be put on the form given in Equation 6-2 yielding the following components

$$\mathbf{U} = \begin{bmatrix} p \\ u \\ v \\ w \end{bmatrix} \qquad \mathbf{d}_a = \begin{bmatrix} 1/c_0^2 & 0 & 0 & 0 \\ u_0/c_0^2 & \rho_0 & 0 & 0 \\ v_0/c_0^2 & 0 & \rho_0 & 0 \\ w_0/c_0^2 & 0 & 0 & \rho_0 \end{bmatrix} \qquad \Gamma(\mathbf{U}) = \begin{bmatrix} \Gamma(\mathbf{U})_p \\ \Gamma(\mathbf{U})_u \\ \Gamma(\mathbf{U})_v \\ \Gamma(\mathbf{U})_w \end{bmatrix} \tag{6-5}$$

with $\mathbf{u} = [u, v, w]^T$ and $\mathbf{u}_0 = [u_0, v_0, w_0]^T$, and the flux components are

$$\begin{aligned} \Gamma(\mathbf{U})_p &= \rho_0 \mathbf{u} + \frac{p}{c_0^2}\mathbf{u}_0 \\ \Gamma(\mathbf{U})_u &= \frac{p}{c_0^2}u_0\mathbf{u}_0 + \rho_0(u\mathbf{u}_0 + u_0\mathbf{u}) + \begin{bmatrix} p \\ 0 \\ 0 \end{bmatrix} \\ \Gamma(\mathbf{U})_v &= \frac{p}{c_0^2}v_0\mathbf{u}_0 + \rho_0(v\mathbf{u}_0 + v_0\mathbf{u}) + \begin{bmatrix} 0 \\ p \\ 0 \end{bmatrix} \\ \Gamma(\mathbf{U})_w &= \frac{p}{c_0^2}w_0\mathbf{u}_0 + \rho_0(w\mathbf{u}_0 + w_0\mathbf{u}) + \begin{bmatrix} 0 \\ 0 \\ p \end{bmatrix} \end{aligned} \tag{6-6}$$

Boundary Conditions

Within the DG formulation all boundary conditions are given in terms of the normal flux $-\mathbf{n}\cdot\Gamma$ at a given (exterior) boundary. Thus prescribing a boundary condition means specifying the vector $\mathbf{g}$ in the expression

$$-\mathbf{n}\cdot\Gamma = \begin{bmatrix} \rho_0(\mathbf{n}\cdot\mathbf{u}) + \frac{p}{c_0^2}(\mathbf{n}\cdot\mathbf{u}_0) \\ \frac{p}{c_0^2}u_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(u(\mathbf{n}\cdot\mathbf{u}_0) + u_0(\mathbf{n}\cdot\mathbf{u})) + n_1 p \\ \frac{p}{c_0^2}v_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(v(\mathbf{n}\cdot\mathbf{u}_0) + v_0(\mathbf{n}\cdot\mathbf{u})) + n_2 p \\ \frac{p}{c_0^2}w_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(w(\mathbf{n}\cdot\mathbf{u}_0) + w_0(\mathbf{n}\cdot\mathbf{u})) + n_3 p \end{bmatrix} = \mathbf{g} \tag{6-7}$$

This very general condition is the one available in the General Flux/Source condition. Details about the proper formulation of the flux terms entering the vector $\mathbf{g}$ can be found in Ref. 3.

The Lax-Friedrichs Flux

The Lax-Friedrichs (LF) flux parameter used in the definition of the Lax-Friedrichs numerical flux is given, element wise for each dependent variable, by the expression

$$\boldsymbol{\tau}^* = \tau_{\mathrm{LF}} \begin{bmatrix} \dfrac{c_0 + u_0}{c_0^2} \\ (c_0 + u_0)\rho_0 \\ (c_0 + u_0)\rho_0 \\ (c_0 + u_0)\rho_0 \end{bmatrix} \tag{6-8}$$

where $u_0 = |\mathbf{u}_0|$ is the norm of the background mean flow velocity. In the CWE interface we use the local value of the velocity. See Ref. 3 for further details on the numerical fluxes in general.

The value of the parameter τ_{LF} (the parameter that can be edited in the user interface) controls the numerical flux between the elements (nodal discontinuous Lagrange elements) used with the DG method. The numerical flux defines how adjacent elements are connected and how continuous p and $\mathbf{u}$ are. Different definitions of the numerical flux lead to different variants of the DG method. The flux implemented here is the so-called global Lax-Friedrichs numerical flux. The value of the parameter τ_{LF} should be between 0 and 0.5. For $\tau_{\mathrm{LF}} = 0$ a so-called central flux is obtained. Setting $\tau_{\mathrm{LF}} = 0.5$ gives a maximally dissipative global Lax-Friedrich flux.

For general information about the numerical flux see the Numerical Flux section under Wave Form PDE in the *COMSOL Multiphysics Reference Guide*.

References for the Convected Wave Equation Interface

1. A. D. Pierce, “Acoustics and Introduction to its Physical Principles and Applications”, Acoustical society of America (1991).

2. A. D. Pierce, “Wave equation for sound in fluids with unsteady inhomogeneous flow”, J. Acoust. Soc. Am. **87**, 2293 (1990).

3. J. S. Hesthaven and T. Warburton, “Nodal Discontinuous Galerkin Methods: Algorithms, Analysis, and Applications”, Springer (2008).

4. M. K. Myers, “On the Acoustic Boundary Condition in the Presence of Flow”, J. Sound Vibration **73**, 429-434 (1980).

5. P. G. Petropoulos, L. Zhao, and A. C. Cangellaris, “A Reflectionless Sponge Layer Absorbing Boundary Condition for the Solution of Maxwell’s Equations with High-Order Staggered Finite Difference Schemes”, J. Comp. Phys. **139**, 184-208 (1998).

6. C. W. Tam, “Computational Aeroacoustics: Issues and Methods”, AIAA Journal, vol. 33, 1995.

7

Geometrical Acoustics Interfaces

This chapter describes the physics interfaces found under the **Acoustics>Geometrical Acoustics** branch ( ).

The Ray Acoustics Interface

The **Ray Acoustics (rac)** interface ( ), found under the **Acoustics>Geometrical Acoustics** subbranch ( ) when adding a physics interface, is used to compute the trajectories, phase, and intensity of acoustic rays. Ray acoustics is valid in the high-frequency limit where the acoustic wavelength is much smaller than the characteristic geometric features. The interface can be used to model acoustics in rooms, concert halls, and many outdoor environments.

The properties of the media in which the rays propagate can change continuously within domains or discontinuously at boundaries. At exterior boundaries it is possible to assign a variety of wall conditions, including combinations of specular and diffuse reflection. Impedance and absorption can depend on the frequency, intensity, and direction of incident rays. Transmission and reflection are also modeled at material discontinuities. A background velocity may also be assigned to any medium.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Medium Properties**, **Wall**, **Material Discontinuity**, and **Ray Properties**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Ray Acoustics** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `rac`.

RAY RELEASE AND PROPAGATION

By default, the **Allow frequency distributions at release features** check box is cleared. Select this check box to model propagation of rays of different frequencies simultaneously. The ray frequency can be specified at release features by entering a value or expression, sampling the frequency from a distribution, or entering a list of values. If this check box is cleared, the frequency is the same for all rays and is specified

in the settings for the Ray Properties node.

Regardless of whether the **Allow frequency distributions at release features** check box is selected, properties at walls can depend on the frequency (variable `rac.f`) of each ray released. Domain material properties, like the speed of sound, can also depend on the frequency. Entering a frequency distribution of the released rays also enables emitting more complex source signals (decomposed in their Fourier components).

To define the speed of sound or another medium property as a function of ray properties such as the ray frequency or intensity, the ray variable must be enclosed in the `noenv()` operator. For example, to use the ray frequency `rac.f` in an expression for the speed of sound, it must be included as part of the expression `noenv(rac.f)`.

See also Medium Properties.

Enter a value or expression for the **Speed of sound in exterior domains** (SI unit: m/s). The default value is `343[m/s]`. Then enter a value or expression for the **Density of exterior domains** (SI unit: kg/m^3). The default value is `1.2[kg/m^3]`. These material properties are used when tracing rays outside the geometry or when tracing rays through domains that are not included in the selection of the Ray Acoustics physics interface. The medium properties in the deselected and exterior void domains must be constant and cannot be functions of any field variables, such as temperature.

The **Maximum number of secondary rays** prevents an inordinate number of rays from being generated by capping them at the number supplied in the text field. The default is 500. Rather than being produced directly by release features such as the Release from Grid node, secondary rays are released when an existing ray is subjected to certain boundary conditions. For example, when a ray undergoes refraction at a Material Discontinuity between different media, the incident ray is refracted and a reflected ray is created; the degrees of freedom for this reflected ray are taken from one of the available secondary rays, which are preallocated when the study begins.

If an insufficient number of secondary rays are preallocated, a reflected ray may not be released when an existing ray undergoes refraction. Thus, the **Maximum number of secondary rays** should be large enough that all reflected rays which significantly affect the solution can be released.

INTENSITY COMPUTATION

Select an option from the **Intensity computation** list — **None** (the default), **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media**. For **None** the ray intensity is not computed.

- For **Compute intensity** auxiliary dependent variables are used to compute the intensity of each ray. For a complete list of the auxiliary dependent variables that are defined, see Intensity and Wavefront Curvature in Theory for the Ray Acoustics Interface. This option is more accurate and is usually less computationally demanding than **Compute intensity in graded media** but is only valid for computing intensity in homogeneous (constant speed of sound) media.
- For **Compute intensity and power** the total power transmitted by each ray is defined as an auxiliary dependent variable, in addition to the auxiliary dependent variables that are declared when **Compute intensity** is selected. The Sound Pressure Level Calculation subnode is available for the Wall feature.
- For **Compute intensity in graded media** auxiliary dependent variables are used to compute the intensity of each ray. The intensity is affected by gradients in the speed of sound. Graded media may be present, for example, when modeling the effect of the saline climb in underwater acoustics. For a complete list of the auxiliary dependent variables that are defined, see Intensity and Wavefront Curvature in Theory for the Ray Acoustics Interface. This intensity computation method is valid for both homogeneous and graded media. If all media are homogeneous, meaning that the gradient of the speed of sound is zero everywhere except at material discontinuities, then it is recommended to select **Compute intensity** instead, since it is the more accurate intensity computation method for such cases.
- For **Compute intensity and power in graded media** the total power transmitted by each ray is defined as an auxiliary dependent variable, in addition to the auxiliary dependent variables that are declared when **Compute intensity in graded media** is selected. The Sound Pressure Level Calculation subnode is available for the Wall feature. If all media are homogeneous then it is recommended to select **Compute intensity and power** instead.

When the **Intensity computation** is enabled (one of the options **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** is selected), a list for the **Reference pressure for the sound pressure level** appears. Select **Use reference pressure for air** (the default) to set the reference pressure to 20 μPa, **Use reference pressure for water** to set the reference pressure to 1 μPa, or **User-defined reference pressure** to enter a value or expression in

the text field. The reference pressure is used to calculate the sound pressure level (variable `rac.Lp`) and sound intensity level (variable `rac.LI`) along rays.

If the **Intensity computation** is enabled (one of the options **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** is selected), the **Compute phase** check box is available. By default, the **Compute phase** check box is cleared. Select this check box to compute the phase along each ray. An auxiliary dependent variable for the phase `rac.Psi` is created. The phase changes along the ray trajectory based on its frequency and is also subjected to phase shifts that happen at walls with specular reflection and absorption.

When **Compute intensity in graded media** or **Compute intensity and power in graded media** is selected from the **Intensity computation** list, enter a **Tolerance for curvature tensor computation** (dimensionless). This tolerance is used internally when computing the principal radii of curvature in a graded medium, and is used to define a criterion by which both principal radii of curvature are declared approximately equal. A larger tolerance makes the solution less accurate but more stable.

ADDITIONAL VARIABLES

By default, the **Store ray status data** check box is cleared. Select this check box to add new variables for quantities that cannot necessarily be recovered from the ray trajectory data alone. This is especially true if automatic remeshing has been used in a model. The following variables are created:

- The release time of a given ray (variable name `rac.rti`).
- The time at which a ray stopped at a boundary (variable name `rac.st`).
- The final status of the ray (variable name `rac.fs`). This indicates the status of a ray at the final time step. The value is an integer which has one of the following values:
  - 0 for unreleased rays
  - 1 for rays that are still in the modeling domain
  - 2 for frozen rays
  - 3 for stuck rays
  - 4 for rays that have disappeared.

To summarize the total number of rays having each final status, the following global variables are also defined.

TABLE 7-1: GLOBAL VARIABLES BASED ON RAY STATUS

NAME	DESCRIPTION
`fac`	Fraction of active rays at final time step
`fds`	Fraction of disappeared rays at final time step
`ffr`	Fraction of frozen rays at final time step
`fse`	Fraction of secondary rays released
`fst`	Fraction of stuck rays at final time step

The global variable names in Table 7-1 all take the unreleased secondary rays into account. For example, suppose an instance of the Ray Acoustics interface includes 100 primary rays and 100 allocated secondary rays. At the last time step, suppose that 80 of the primary rays have disappeared at boundaries and that 40 secondary rays have been emitted, all of which are still active. Then the variable `rac.fac`, the fraction of active rays at the final time step, would have the value $(20 + 40)/(100 + 100)$ or 0.3.

When creating impulse response plots, energy decay curves, or reflectograms in postprocessing it is necessary to select the **Store ray status data** check box. Typically some of the derived variables are necessary. Of special importance is the stop time that can give the arrival time of a ray at a surface used as receiver.

Calculating an impulse response also requires the **Compute phase** option to be selected. A pressure variable `rac.p` is created that includes phase information along the rays.

See also Impulse Response, Results Plots, Data Sets, and Derived Values.

ADVANCED SETTINGS

This section is only shown when **Advanced Physics Options** are enabled (click the **Show** button on the **Model Builder**).

The **Wall accuracy order** sets the accuracy order of the time stepping used for time steps during which a ray-wall interaction happens. Select an order of **1** to use a forward Euler step and compute the motion both before and after the wall collision. Select an order of **2** (the default) to use a second-order Taylor method and compute the motion before the wall collision. After the collision a second-order Runge-Kutta method is used.

Select an option from the **Arguments for random number generation** list: **Generate unique arguments**, **Generate random arguments**, or **User defined**. This setting determines how the additional argument to random functions is defined in features such as the Wall boundary condition with the **Diffuse scattering** wall condition. Typically the random numbers are functions of the ray index, position, time, and another argument i, which is defined as follows:

- For **Generate unique arguments** the additional argument is based on the position of each node in the Model Builder. As a result, random numbers generated in different nodes are created independently of each other, but the same result can be reproduced by running the same study several times.
- For **Generate random arguments** the additional argument is randomly created, causing the random functions to return different results each time the study is run.
- For **User defined** the additional argument is defined by a user input in the **Settings** window each feature. Independent solutions can be obtained by running a parametric sweep for different values of i.

By default the **Allow propagation outside selected domains** check box is selected. When this check box is selected, rays can propagate in domains that are not included in the selection for the physics interface. These exterior domains do not need to be meshed. Rays can even propagate through the void region outside the geometry. However, all boundaries that the rays interact with must be meshed.

If a boundary condition is applied to a surface that is not adjacent to any domains in the selection for the physics interface, the default meshing algorithm will automatically create a boundary mesh as needed.

There are a variety of ways to make rays propagate outside the geometry. The most straightforward ways are to use the Release from Grid feature and specify initial coordinates that are not within any domain. Alternatively, rays can escape from a domain into the exterior of the physics interface selection by applying a Material Discontinuity feature on exterior boundaries.

The **Allow multiple release times** check box, which is cleared by default, allows an array of release times for the rays to be specified in any of the ray release features. If the check box is cleared, all rays are released at time $t = 0$.

Enter a value for the **Maximum number of wall interactions per time step**. The default value is 1000. If a ray undergoes more than the specified number of boundary

interactions in a single time step taken by the solver, the ray will disappear. This is included as a safeguard to prevent rays from getting stuck in infinite loops if the time between successive ray-wall interactions becomes infinitesimally small.

DEPENDENT VARIABLES

The dependent variables (field variables) are the **Ray position components** and **Wave number components**. The name can be changed but the names of fields and dependent variables must be unique within a model.

Theory for the Ray Acoustics Interface

Domain, Boundary, and Global Nodes for the Ray Acoustics Interface

The Ray Acoustics Interface has these domain, boundary, and global nodes available (listed in alphabetical order).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Accumulator (Boundary)
- Accumulator (Domain)
- Auxiliary Dependent Variable
- Axial Symmetry
- Background Velocity
- Inlet
- Inlet on Axis
- Material Discontinuity
- Medium Properties
- Nonlocal Accumulator
- Ray Continuity
- Ray Detector
- Ray Properties
- Ray Termination
- Release
- Release from Data File
- Release from Edge
- Release from Grid
- Release from Grid on Axis
- Release from Point
- Release from Point on Axis
- Sound Pressure Level Calculation
- Wall

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Medium Properties

Use the **Medium Properties** node to specify the speed of sound and density of the medium.

PRESSURE ACOUSTICS MODEL

This section is available if **Compute intensity** or **Compute intensity in graded media** is selected from the **Intensity computation** list in the physics interface **Ray Properties** section. Select an option from the **Fluid model** list — **Linear elastic** (the default), **Linear elastic with attenuation**, or **Thermally conducting and viscous**. This section determines which additional fluid properties can be specified in the **Medium Properties** section below.

MEDIUM PROPERTIES

The default **Speed of sound** c (SI unit: m/s) uses values **From material**. For **User defined** enter a value or expression. The default is 343 m/s.

If **Compute intensity** or **Compute intensity in graded media** is selected from the **Intensity computation** list in the physics interface **Ray Properties** section, the following edit fields are available, depending on the option selected from the **Fluid model** list in the **Pressure Acoustics Model** section:

- If **Linear elastic** is selected, specify the **Density** ρ (SI unit: kg/m^3). The default density uses values **From material**. For **User defined** enter a value or expression. The default is 1.2 kg/m^3.
- If **Linear elastic with attenuation** is selected, specify the **Density** ρ (SI unit: kg/m^3). The default density uses values **From material**. For **User defined** enter a value or expression. The default is 1.2 kg/m^3. Enter a value or expression for the Attenuation coefficient α (SI unit: 1/m). The default is 0.
- If **Thermally conducting and viscous** is selected, specify the following material properties. By default, all fluid properties take default values **From material**. Choose **User defined** to enter a value or expression for the:
  - **Density** ρ (SI unit: kg/m^3). The default is 1.2 kg/m^3.
  - **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)). The default value is 1005.4 J/(kg·K).
  - **Ratio of specific heats** γ (dimensionless). The default value is 1.4.
  - **Thermal conductivity** k (SI unit: W/(m·K)). The default value is 0.0257 W/(m·K).
  - **Dynamic viscosity** μ (SI unit: Pa·s). The default value is $1.81 \cdot 10^{-5}$ Pa·s.
  - **Bulk viscosity** μ_B (SI unit: Pa·s). The default value is 0.

To define the speed of sound or another medium property as a function of ray properties such as the ray frequency or intensity, the ray variable must be enclosed in the `noenv()` operator. For example, to use the ray frequency `rac.f` in an expression for the speed of sound, it must be included as part of the expression `noenv(rac.f)`.

Attenuation Within Domains

Wall

Use the **Wall** node to determine what happens to the rays when contact with a boundary is made. The **Wall** node is the default boundary condition on all exterior boundaries.

The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

WALL CONDITION

Select a **Wall condition** — **Freeze** (the default), **Specular Reflection**, **Stick**, **Disappear**, **Pass through**, **Diffuse scattering**, **Mixed diffuse and specular reflection**, or **General reflection**.

TABLE 7-2: WALL CONDITION OPTIONS

OPTIONS	DESCRIPTION
Freeze	Select to fix the ray position and wave vector at the instant a wall is struck. So, the ray position no longer changes after contact with the wall and the wave vector of the ray remains at the same value as when the ray struck the wall. This boundary condition is typically used to recover the ray intensity or phase at the instant contact was made with the wall.
Specular Reflection	Select to specularly reflect from the wall. Reinitialization of the principal radii of curvature of the reflected ray is consistent with the algorithms used for reflected rays at material discontinuities. This means, for example, that the ray intensity may become very large or very small in magnitude after being specularly reflected by a curved surface, depending on the concavity.
Stick	Select to fix the ray position at the instant the wall is struck. The wave vector components are set to zero.
Disappear	This option means that the ray is not displayed once it has made contact with the wall. Use it if the ray location after contact with the wall is not of interest.
Pass through	This option allows rays to cross the boundary unimpeded. The Pass Through condition does not cause rays to be refracted at the boundary between different materials. If used at the boundary between two different media, it is possible for the resulting rays to undergo a non-physical change in angular frequency and wave vector magnitude. To model refraction, instead use the Material Discontinuity node.
Diffuse scattering	Select to reflect rays at a wall according to Lambert's cosine law. That is, the probability of a reflected ray propagating in a given direction within a solid angle $d\omega$ is given by $\cos(\theta)d\omega$ where θ is the angle between the direction of the ray and the wall normal.

TABLE 7-2: WALL CONDITION OPTIONS

OPTIONS	DESCRIPTION
Mixed diffuse and specular reflection	Select to reflect rays at a wall either specularly or according to Lambert's cosine law, based on a user-defined probability. By combining this wall condition with a Primary ray condition, it is possible to include up to three different types of ray-wall interactions at a single boundary.
General reflection	Select to allow an arbitrary direction vector to be specified after a ray makes contact with the wall. The ray direction vector components can be functions of the incident ray direction, phase, or any other quantity.

GENERAL REFLECTION SETTINGS

This section is available when **General reflection** (see Table 7-2) is selected as the **Wall condition**.

Enter values for the **Reflected ray direction vector** $\mathbf{L}_p$ (dimensionless) either in Cartesian coordinates (x, y, z) (the default) or select the **Specify tangential and normal direction vector components** check box to enter coordinates in the tangent-normal coordinate system (t_1, t_2, n). In this case the normal direction is selected so that an incident ray is reflected back into the domain it previously occupied if the specified normal direction vector component is positive. The tangential directions are oriented so that they form a right-handed coordinate system, together with the normal direction.

PRIMARY RAY CONDITION

Select a **Primary ray condition** — **None** (the default), **Probability**, or **Expression**. When the default, **None**, is kept, it means that the **Wall condition** is always respected by the incident rays.

Probability

If **Probability** is selected, the **Wall condition** is applied with a certain probability. Enter a value for the **Probability**, γ (dimensionless). If the **Wall condition** is not used, the ray instead behaves according to the Otherwise setting.

The value of γ should always be between 0 and 1.

For example, if the **Wall condition** is set to:

- **Freeze** and γ is set to 0.1, then for every 10 rays that strike the wall, on average one freezes and the remaining 9 rays behave according to the Otherwise setting.
- **Stick** and γ is set to 0.5 then on average half of the rays stick to the wall and the other half behave according to the Otherwise setting.

Expression

If **Expression** is selected, the **Evaluation expression** e (dimensionless) is evaluated whenever a ray strikes the wall. The default expression is `1`. If the Evaluation expression is nonzero, the ray behaves according to the **Wall condition**, otherwise the ray behaves according to the Otherwise setting.

Otherwise

The options available for the **Otherwise** setting are the same as for the Wall Condition, except that **General reflection** and **Mixed diffuse and specular reflection** are not available. The **Otherwise** setting can be used to make rays interact with a wall differently with a certain probability or when a certain condition is satisfied. For example, to model reflection at a partially specular surface in which 50% of the ray intensity is absorbed and 30% is reflected specularly:

- Select **Mixed diffuse and specular reflection** as the **Wall condition**,
- set the **Probability of specular reflection** to `0.3/(1-0.5)`,
- set the **Primary ray condition** to **Probability**,
- set the **Probability** to `1-0.5`, and
- select **Freeze** as the **Otherwise** option.

PHASE SHIFT

This section is available when

- the **Intensity computation** is set to **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** under the physics interface **Ray Properties** section,
- The **Compute phase** check box is selected under the physics interface **Advanced settings** section, and
- the **Wall condition** is set to **Specular Reflection**, **Diffuse scattering**, **General reflection**, or **Mixed diffuse and specular reflection**.

Select the **Apply manual phase shift** check box to specify the phase shift applied to reflected rays. By default this check box is cleared, causing the phase shift be derived from the reflection coefficient.

If the **Apply manual phase shift** check box is selected, enter a Phase shift $\Delta\Psi$ (SI unit: rad). The default is 0.

If the **Wall condition** is set to **Mixed diffuse and specular reflection**, this section is instead called **Phase shifts** and the phase shifts for the diffusely and specularly reflected rays are specified separately. All of the text fields are given subscripts s or d for specularly and diffusely reflected rays, respectively.

ABSORPTION COEFFICIENT

This section is available when

- the **Intensity computation** is set to **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** under the physics interface **Intensity Computation** section, and
- the **Wall condition** is set to **Specular Reflection**, **Diffuse scattering**, **General reflection**, or **Mixed diffuse and specular reflection**.

Select one of the following from the **Compute reflected intensity using** list: **Absorption coefficients**, **Absorber, specified impedance**, **Reflection coefficients**, **Fluid-fluid interface**, **Fluid-solid interface**, or **Layered fluid half space**.

For **Absorption coefficients** enter the **Absorption coefficient** α (dimensionless). The default is 0. The intensity of the reflected ray will be proportional to $1\text{-}\alpha$.

For **Absorber, specified impedance** enter the **Characteristic impedance of absorber** Z_1 (SI unit: Pa·s/m). The default value is 1 GPa·s/m, corresponding to a hard wall.

For **Reflection coefficients** enter the **Reflection coefficient** R (dimensionless). The default value is 1. The intensity of the reflected ray will be proportional to R^2.

For **Fluid-fluid interface** enter the following:

- **Speed of sound, adjacent fluid** c_1 (SI unit: m/s). The default is 1500 m/s.
- **Density, adjacent fluid** ρ_1 (SI unit: kg/m^3). The default is 1000 kg/m^3.
- **Attenutation coefficient, adjacent fluid** α_1 (SI unit: 1/m). The default is 0.

For **Fluid-solid interface** enter the following:

- **Compressional speed of sound, adjacent solid** $c_{p,1}$ (SI unit: m/s). The default is 1500 m/s.

- **Shear speed of sound, adjacent solid** $c_{s,1}$ (SI unit: m/s). The default is 1500 m/s.
- **Density, adjacent solid** ρ_1 (SI unit: kg/m^3). The default is 1000 kg/m^3.
- **Compressional attenutation coefficient, adjacent solid** $\alpha_{p,1}$ (SI unit: 1/m). The default is 0.
- **Shear attenutation coefficient, adjacent solid** $\alpha_{s,1}$ (SI unit: 1/m). The default is 0.

For **Layered fluid half-space** enter the following:

- **Thickness, adjacent fluid layer** h (SI unit: m). The default is 1 μm.
- **Speed of sound, adjacent fluid layer** c_1 (SI unit: m/s). The default is 1500 m/s.
- **Density, adjacent fluid layer** ρ_1 (SI unit: kg/m^3). The default is 1000 kg/m^3.
- **Attenutation coefficient, adjacent fluid layer** α_1 (SI unit: 1/m). The default is 0.
- **Speed of sound, adjacent fluid domain** c_2 (SI unit: m/s). The default is 1500 m/s.
- **Density, adjacent fluid domain** ρ_2 (SI unit: kg/m^3). The default is 1000 kg/m^3.
- **Attenutation coefficient, adjacent fluid domain** α_2 (SI unit: 1/m). The default is 0.

If the **Wall condition** is set to **Mixed diffuse and specular reflection**, this section is instead called **Absorption Coefficients** and the absorption coefficients for the diffusely and specularly reflected rays are specified separately. All of the text fields are given subscripts *s* and *d* for specularly and diffusely reflected rays, respectively.

To create detailed models the wall properties, for example, the surface normal impedance or the absorption coefficient, must depend on the properties of the incident ray. Doing this is possible as all wall properties may depend on:

- The frequency of the ray `rac.f`
- The intensity of the ray `rac.I`
- The direction of the ray, that is, the acute angle of incidence `rac.phii`

SURFACE ROUGHNESS

This section is available when

- the **Intensity computation** is set to **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** under the physics interface **Intensity Computation** section, and
- the **Wall condition** is set to **Specular Reflection**, **Diffuse scattering**, **General reflection**, or **Mixed diffuse and specular reflection**.

Select an option from the **Surface roughness** model list: **None** (the default) or **Rayleigh roughness**. If **Rayleigh roughness** is selected, enter the **RMS roughness** σ (SI unit: m). The default is 0. Use this setting to reduce the intensity of rays that are reflected by a rough wall.

NEW VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

When a ray crosses or touches a boundary, the value of any user-defined auxiliary dependent variable can be changed. The value can be a function of any combinations of ray variables and variables defined on the boundary. A simple application is to use this to count the number of times a ray strikes the wall.

Select the **Assign new value to auxiliary variable** check box or boxes based on the number of auxiliary variables in the model. Then enter the new value or expression in the field. For example, if there is an auxiliary variable, psi, then enter a value for psi_{new} in the field. So, to increment the value of psi by 1 when a ray touches or crosses a boundary, enter psi+1 in the text field for psi_{new}.

ADVANCED SETTINGS

If the **Primary ray condition** is set to **Probability**, or if the **Diffuse scattering** or **Mixed diffuse and specular reflection** wall condition is used, then the **Wall** feature generates random numbers.

If, in addition, the **Arguments for random number generation** setting is set to **User defined** in the physics interface **Advanced Settings** section, the **Advanced Settings** section is available.

Enter the **Additional input argument to random number generator** i (dimensionless). The default value is 1.

Axial Symmetry

The **Axial Symmetry** node is automatically added to 2D axisymmetric models. The options available for the feature are the same as for Wall, except as follows. The **Boundary Selection** is locked and is only applicable to the symmetry axis. The only wall

conditions available are **Specular Reflection** (the default), **Freeze**, **Stick**, and **Disappear**. In addition, the **Primary Ray Condition** section is not included.

The **Axial Symmetry** condition only applies to straight boundaries that are both on the axis of symmetry and adjacent to a domain in the selection of the physics interface. If rays are propagating in the void region outside of the selected domains, it is possible for the rays to enter the region where $r < 0$, even though no geometry is defined there.

To ensure that rays cannot propagate into the region where $r < 0$, draw line segments on the parts of the symmetry axis where no geometric entities exist, then add the **Wall** node to these edges. Keep in mind that rays can only be subjected to boundary conditions if the corresponding boundaries are meshed.

Accumulator (Boundary)

The **Accumulator** subnode is available from the context menu (right-click the Wall or Material Discontinuity parent node) or from the **Physics** toolbar, **Attributes** menu. Each **Accumulator** subnode defines a variable, called the accumulated variable, on each boundary element in the selection of the parent node. The accumulated variables are incremented when rays interact with the boundaries where they are defined.

ACCUMULATOR SETTINGS

Select an option from the **Accumulator type** list — **Density** (default) or **Count**.

- For **Density** the accumulated variable is divided by the surface area of the boundary element where it is defined.
- For **Count** the accumulated variable is the sum of the source terms of all rays that hit the boundary element, and is unaffected by the boundary element size.

Select an option from the **Accumulate over** list — **Ray-wall interactions** (the default) or **Rays in boundary elements**.

- For **Ray-wall interactions** the accumulated variable is affected by all rays that hit the boundary element.
- For **Rays in boundary elements** the accumulated variable is only affected by rays that freeze or stick to the boundary element.

Enter the **Accumulated variable name**. The default is `rpb`. The accumulated variable is defined as `<scope>.<name>`, where `<scope>` includes the name of the physics interface node, parent boundary condition, and the **Accumulator** node, and `<name>` is the accumulated variable name.

For example, if the **Accumulator** subnode is added to a Wall node in an instance of the Ray Acoustics interface using the default variable name `rpb`, the accumulated variable name might be `rac.wall1.bacc1.rpb`.

Enter a **Source** R. The unit of the source term depends on the settings in the **Units** section. Whenever a ray collides with a boundary element in the selection of the parent node, the accumulated variable in that element is incremented by the source term. If the **Accumulator type** is set to **Density**, the source term is divided by the area of the boundary element (in 3D) or the length of the boundary element (in 2D).

For example, if the source is `1` and **Count** is selected from the **Accumulator type** list, a variable is created to store the number of times each boundary element is hit by a ray.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

SMOOTHING

The accumulated variables are computed using discontinuous shape functions. Select the **Compute smoothed accumulated variable** check box to compute a smoothed accumulated variable by computing the average value of the variable within a sphere of a user-defined radius. Then enter a **Smoothing radius** r (SI unit: m). The default is `0.1[m]`.

Accumulator Theory: Boundaries

Material Discontinuity

The **Material Discontinuity** node is the default boundary feature on interior boundaries. It computes the initial direction of the refracted ray using Snell's law. If extra degrees of freedom have been allocated for secondary rays, a reflected ray is also produced. If

the incident ray undergoes total internal reflection, no refracted ray is produced and no secondary rays are needed to release the reflected ray.

If **Intensity computation** is set to **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** in the physics interface **Intensity Computation** section, the **Material Discontinuity** feature computes the radii of curvature and the reinitialized intensity of the reflected and refracted rays.

The maximum number of reflected rays can be controlled via the **Maximum number of secondary rays** text field, which is found under the **Ray Release and Propagation** section for the physics interface.

The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

THRESHOLD INTENSITY

This section is available if **Intensity computation** is set to **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** under the **Intensity Computation** section for the physics interface.

Enter a **Threshold intensity** I_{th} (SI unit: W/m^2). The default is $1\cdot10^{-3}$ W/m^2. If the interaction of a ray with a material discontinuity would create a reflected ray of intensity less than the threshold intensity, the release of this reflected ray is suppressed. This prevents an arbitrarily large number of degrees of freedom from being used to model the propagation of rays of exponentially decreasing intensity. When a nonzero threshold intensity is specified, some small decreases in the total energy of the system may be observed if the release of low-intensity secondary rays is suppressed.

AUXILIARY DEPENDENT VARIABLES, REFRACTED RAY

If an Auxiliary Dependent Variable has been added to the model then there is an option to reinitialize the values of auxiliary dependent variables for the refracted ray. These setting are the same as in the New Value of Auxiliary Dependent Variables section for the Wall feature.

AUXILIARY DEPENDENT VARIABLES, REFLECTED RAY

If an Auxiliary Dependent Variable has been added to the model then there is an option to reinitialize the values of auxiliary dependent variables for the reflected ray. These setting are the same as in the New Value of Auxiliary Dependent Variables

section for the Wall feature.

ADVANCED SETTINGS

Select the **Show boundary normal** check box to view the boundary normal in the **Graphics** window.

Material Discontinuity Theory

Ray Properties

The behavior of the **Ray Properties** node is based on whether the **Allow frequency distributions at release features** check box is selected under the physics interface **Ray Release and Propagation** section.

By default, the check box is not selected. Then the **Ray Properties** node is used to specify the frequency of the rays. When the check box is selected, the ray frequency is specified in the individual release features.

RAY PROPERTIES

This section is shown when the **Allow frequency distributions at release features** check box is cleared in the physics interface **Ray Release and Propagation** section.

Enter a **Ray frequency** f (SI unit: Hz) here. The default is 1000 Hz.

Release

Use the **Release** node to release rays within domains based on arbitrary expressions or based on the positions of the mesh elements.

RELEASE TIMES

Enter **Release times** (SI unit: s) or click the **Range** button ( ) to select and define a range of specific times. At each release time, rays are released with initial position and ray direction vector as defined next.

This section is only available when the **Allow multiple release times** property is active. This can be found in the **Advanced** section of the physics interface settings when **Advanced Physics Options** are shown in the Model Builder.

INITIAL POSITION

Select an **Initial position** — **Density** (the default) or **Mesh based**.

Density

For **Density** enter a value for the **Number of rays per release** N and the **Density proportional to** ρ. Both are dimensionless numbers.

The **Density proportional to** ρ is an expression — the resulting ray distribution approximately has a density that is proportional to this expression. The resulting distribution looks a bit random, and it depends on the order in which the mesh elements are numbered. The distribution is probably not exactly the same in different COMSOL Multiphysics versions, but the total number of rays released is always N.

The **Density proportional to** expression must be strictly positive.

Select a **Release distribution accuracy order** between **1** and **5** (the default is **5**), which determines the integration order that is used when computing the number of rays to release within each mesh element. The higher the accuracy order, the more accurately rays will be distributed among the mesh elements.

The **Position refinement factor** (default `0`) must be a nonnegative integer. When the refinement factor is `0`, each ray is always assigned a unique position, but the density is taken as a uniform value over each mesh element. If the refinement factor is a positive integer, the distribution of rays within each mesh element is weighted according to the density, but it is possible for some rays to occupy the same initial position. Further increasing the **Position refinement factor** increases the number of evaluation points within each mesh element to reduce the probability of multiple rays occupying the same initial position.

Mesh Based

For **Mesh based** the rays are released from a set of positions determined by a selection of geometric entities (of arbitrary dimension) in the mesh. Given a **Refinement factor** between 1 and 5, the centers of the refined mesh elements are used. Thus, the number of positions per mesh element is `refine^dim`, except for pyramids, where it is `(4*refine^2-1)*refine/3`.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list — **Expression** (the default), **Spherical**, **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **Expression** a single ray is released in the specified direction. Enter coordinates for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless) based on space dimension.
- For **Spherical** a number of rays are released at each point, sampled from a spherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50.
- For **Hemispherical** a number of rays are released at each point, sampled from a hemispherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis** $\mathbf{r}$ based on space dimension.
- For **Conical** a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Cone axis** $\mathbf{r}$ based on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is $\pi/3$ radians.
- The **Lambertian** option is only available in 3D. Anumber of rays are released at each point, sampled from a hemisphere in wave vector space with probability density based on the cosine law. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis** $\mathbf{r}$ based on space dimension.

If **Conical** is selected, select an option from the **Conical distribution** list — **Uniform density** (the default), **Specify polar and azimuthal distributions**, **Marginal rays only**, or **Marginal and axial rays only**.

- For Uniform density rays are released with polar angles from 0 to the specified cone angle. The rays are distributed in wave vector space so that each ray subtends approximately the same solid angle.
- For **Specify polar and azimuthal distributions** specify the **Number of polar angles** N_ϕ (dimensionless) and the **Number of azimuthal** angles N_θ (dimensionless). Rays are released at uniformly distributed polar angles from 0 to the specified cone angle. A single axial ray ($\phi = 0$) is also released. For each value of the polar angle, rays are released at uniformly distributed azimuthal angles from 0 to 2π. Unlike other options for specifying the conical distribution, it is not necessary to directly specify the **Number of rays in wave vector space** N_w (dimensionless), which is instead derived from the relation $N_w = N_\phi \times N_\theta + 1$.

- For **Marginal rays only** the rays are all released at an angle α with respect to the cone axis. The rays are released at uniformly distributed azimuthal angles from 0 to 2π.
- For **Marginal and axial rays only** the rays are all released at an angle α with respect to the cone axis, except for one ray which is released along the cone axis. The marginal rays are released at uniformly distributed azimuthal angles from 0 to 2π.

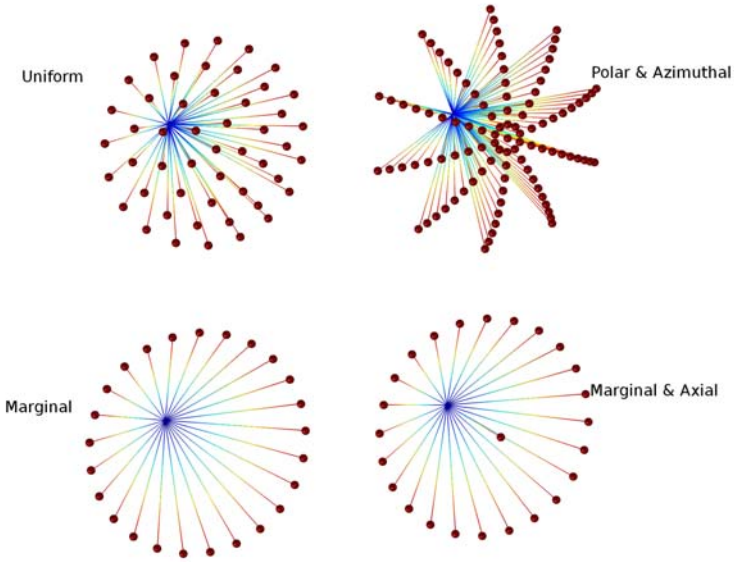

Figure 7-1: Comparison of the four available cone-based release distributions.

For **Spherical**, **Hemispherical**, **Conical**, and **Lambertian**, select an option from the **Sampling from Distribution** list — **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray direction vectors are computed using an algorithm that seeks to distribute the rays as evenly as possible in wave vector space. This algorithm will give the same initial ray directions whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudorandom numbers. The result may be the same when rerunning the study multiple times on the same computer, but the solution is likely to be different on different architectures.

For **Expression** it is also possible to initialize the ray direction vector either in the global coordinate system or in a coordinate system that moves with the same velocity as the

background medium. Select an option from the **Initial wave vector specification** list: **With respect to fluid** (the default) or **With respect to coordinate system**.

- For **With respect to fluid** the initial wave vector is computed with respect to a coordinate system that moves at the background velocity, so the initial ray direction might not be parallel to the vector entered in the **Ray direction vector** text field if the medium is moving.
- For **With respect to coordinate system** the initial ray direction is parallel to the vector entered in the **Ray direction vector** text field as long as a ray could reasonably propagate in that direction. For example, rays cannot be released in certain directions if the background fluid is moving with a supersonic velocity.

INITIAL RAY FREQUENCY

This section is available when the **Allow frequency distributions at release features** check box is selected under the physics interface **Ray Release and Propagation** section.

Select a **Distribution function** — **None** (the default), **Normal**, **Lognormal**, **Uniform**, or **List of values**.

When **None** is selected, enter an initial value f_0. The default value is 1000 Hz.

Select **Normal** to create a normal distribution function, **Lognormal** to create a log-normal distribution function, or **Uniform** to create a uniform distribution function. For any of these selections, the **Number of values** sets the number of points in the distribution function. Enter a user-defined **Mean** (default 1000 Hz) and **Standard deviation** (default 100 Hz). Select **List of values** to enter a list of distinct frequency values.

INITIAL PHASE

This section is available when the **Compute phase** check box is selected under the physics interface **Intensity Computation** section. Enter an **Initial phase** Ψ_0 (SI unit: rad). The default value is `0`.

INITIAL INTENSITY

This section is available when:

- the **Intensity computation** is set to **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** in the physics interface **Intensity Computation** section, and
- **Expression** is selected as the **Ray direction vector**.

Enter a value for the **Initial intensity** I_0 (SI unit: $\mathrm{W/m^2}$). The default is 1000 $\mathrm{W/m^2}$.

INITIAL RADII OF CURVATURE

This section is available when:

- the **Intensity computation** is set to **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** in the physics interface **Intensity Computation** section, and
- **Expression** is selected as the **Ray direction vector**.

Select a **Wavefront shape**. In 3D the available options are **Plane wave** (the default), **Spherical wave**, and **Ellipsoid**. In 2D the available options are **Plane wave** (the default) and **Cylindrical wave**.

- For an idealized plane wave the radii of curvature would be infinite. However, because the algorithm used to compute intensity requires finite values, when **Plane wave** is selected the initial radii of curvature are instead given an initial value that is 10^8 times greater than the characteristic size of the geometry.
- For a **Spherical wave** or **Cylindrical wave**, enter the **Initial radius of curvature** r_0 (SI unit: m).
- For an **Ellipsoid**, enter the **Initial radius of curvature, I** $r_{1,0}$ (SI unit: m) and the **Initial radius of curvature, 2** $r_{2,0}$ (SI unit: m). Also enter the **Initial principal curvature direction, I** $\mathbf{e}_{1,0}$ (dimensionless).

Principal Radii of Curvature

TOTAL SOURCE POWER

This section is available:

- when the **Intensity computation** is set to **Compute intensity** or **Compute intensity in graded media** under the physics interface **Intensity Computation** section, and
- when **Spherical**, **Hemispherical**, or **Conical** is selected as the **Ray direction vector**.
- It is also available when the **Intensity computation** is **Compute intensity and power** or **Compute intensity and power in graded media** under the physics interface **Intensity Computation** section, and any choice of **Ray direction vector** displays this section.

Enter a **Total source power** P_{src} (SI unit: W). The default is 1 W. In 2D, instead enter the **Total source power per unit thickness** P_{src} (SI unit: W/m). The default is 1 W/m.

INITIAL VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

For each of the **Auxiliary Dependent Variable** nodes added to the model, select a **Distribution function** for the initial value of the auxiliary dependent variables and whether the initial value of the auxiliary dependent variables should be a scalar value or sampled from a distribution function.

The number of rays simulated can increase substantially and the following options are available for each **Auxiliary Dependent Variable** added to the model.

When **None** is selected, enter an initial value. The symbol for the initial value is the auxiliary variable name followed by a subscript 0, so for the default name rp the initial value has symbol rp_0.

For the initial value of the auxiliary dependent variables, select **Normal** to create a normal distribution function, **Lognormal** to create a log-normal distribution function, or **Uniform** to create a uniform distribution function. For any selection, the **Number of values** sets the number of points in the distribution function. Enter a user-defined **Mean** (default 0) and **Standard deviation** (default 1). Select **List of values** to enter a set of numerical values directly.

By default auxiliary dependent variables are initialized after all other degrees of freedom. Select the **Initialize before wave vector** check box to compute the initial value of the auxiliary dependent variable immediately after computing the initial wave vectors of the rays. By selecting this check box it is possible to define the initial ray direction as a function of the auxiliary dependent variables.

Sound Pressure Level Calculation

When the **Intensity computation** is set to **Compute intensity and power** or **Compute intensity and power in graded media** in the physics interface **Intensity Computation** section, the **Sound Pressure Level Calculation** subnode is available from the context menu (right-click the Wall, parent node) or from the **Physics** toolbar, **Attributes** menu. The **Sound Pressure Level Calculation** subnode computes the sound pressure level resulting from the interaction of the incident and reflected acoustic waves at the wall.

SMOOTHING

Select the **Compute smoothed accumulated variable** check box to enter a **Smoothing radius** r (SI unit: m) The smoothing radius functions like the corresponding setting for the Accumulator (Boundary) node.

Sound Pressure Level Calculation Theory

Accumulator (Domain)

Use the **Accumulator** node to define additional degrees of freedom on a domain. Each **Accumulator** defines a variable, called the accumulated variable, on each domain element in the selection list. The values of the accumulated variables are determined by the properties of rays in each domain element.

ACCUMULATOR SETTINGS

Select an **Accumulator type** — **Density** (default) or **Count**.

- For **Density** the accumulated variable is divided by the volume of the mesh element where it is defined.
- For **Count** the accumulated variable is unaffected by the element size.

Select an option from the **Accumulate over** list — **Elements** (the default) or **Elements and time**.

- For **Elements** the value of the accumulated variable in an element is the sum of the source terms of all rays in that element. If the **Accumulator type** is set to **Density**, this sum is divided by the mesh element volume.
- For **Elements and time** the time derivative of the accumulated variable in an element is the sum of the source terms of all rays in that element. If the **Accumulator type** is set to **Density**, this sum is divided by the mesh element volume. As each ray propagates through a series of mesh elements, it leaves behind a contribution to the accumulated variable that remains even after the ray has moved on.

Enter the **Accumulated variable name**. The default is `rpd`. The accumulated variable is defined as `<name>.<varname>`, where `<name>` is the physics interface name and `<varname>` is the accumulated variable name. For example, in an instance of the Ray Acoustics interface with default name `rac`and default accumulated variable name `rpd`, the variable would be named `rac.rpd`.

Enter a **Source** R. The unit of the source depends on the settings in the **Units** section. The source term is used to calculate the accumulated variable in a manner specified by the **Accumulate over** and **Accumulator type** settings.

If **Elements and time** is selected from the **Accumulate over** list, select an option from the **Source interpolation** list: **Constant**, **Linear** (the default), **Quadratic**, or **Exponential**. The **Source interpolation** determines what functional form the **Source** is assumed to follow during each time step taken by the solver. This information is used to compute the accumulated variable in mesh elements that the rays pass through during each time step.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

Nonlocal Accumulator

Use the **Nonlocal Accumulator** subnode to communicate information from a ray's current position to the position from which it was released.

The subnode is available from the context menu (right-click the Inlet parent node) or from the **Physics** toolbar, **Attributes** menu.

Each **Nonlocal accumulator** subnode defines a variable, called the accumulated variable, that is computed using variables defined on rays released by the parent **Inlet** node or on domains and boundaries encountered by such rays.

ACCUMULATOR SETTINGS

Select an **Accumulator type** — **Density** (default) or **Count**.

- For **Density** the accumulated variable is divided by the volume of the mesh element where it is defined.
- For **Count** the accumulated variable is unaffected by the element size.

Select an option from the Accumulate over list: **Elements** (default) or **Elements and time**.

- For **Elements** the accumulated variable is proportional to the instantaneous value of the **Source** term R for all applicable rays.
- For **Elements and time** the time derivative of the accumulated variable is proportional to the instantaneous value of the **Source** term R for all applicable rays, and thus the accumulated variable considers the time history of rays in the modeling domain instead of just their current values.

Enter the **Accumulated variable name**. The default is `rpi`.

Enter a **Source** R. The unit of the source depends on the settings in the **Units** section. The source term is used to calculate the accumulated variable in a manner specified by the **Accumulate over** and **Accumulator type** settings.

Select a **Source geometric entity level** — **Domains**, **Boundaries**, or **Domains and boundaries**.

- If **Domains** is selected, rays only contribute to the accumulated variable on their releasing surface if they are still active; that is, they are still propagating through a domain.
- If **Boundaries** is selected, the rays only contribute to the accumulated variable if they have become stuck or frozen to a boundary somewhere in the model.
- If **Domains and boundaries** is selected, all of the active, stuck, and frozen rays released by a feature can contribute to the accumulated variable.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

SMOOTHING

Select the **Compute smoothed accumulated variable** check box to enter a **Smoothing radius** r (SI unit: m). The default is 0.1 m.

Inlet

Use the **Inlet** node to determine how to release rays on an interior or exterior boundary in a specific direction.

The Nonlocal Accumulator subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

See Release for information on the following sections: **Release Times**, **Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

COORDINATE SYSTEM SELECTION

It is possible to specify the initial ray direction in terms of the global coordinates or in another coordinate system defined for the model Component. Select an option from the **Coordinate system** list. By default **Global coordinate system** is selected. If other coordinate systems are defined, they can also be selected from the list. When specifying the initial ray direction (see the **Ray Direction Vector** section), direction components can be specified using the basis vectors of whichever coordinate system has been selected from the list.

When a coordinate system other than **Global coordinate system** is selected from the **Coordinate system** list, arrows will appear in the Graphics window to indicate the orientation of the basis vectors of the coordinate system on the selected boundaries.

INITIAL POSITION

Select an **Initial position** — **Uniform distribution** (the default for 2D components) **Projected plane grid** (the default for 3D components), **Mesh based**, or **Density**. Mesh based and Density have the same settings as described for the Release node.

	For 2D components, if **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The union of the selected boundaries is divided into N segments of approximately equal length, and a ray is placed in the middle of each segment.

	For 3D components, if **Projected plane grid** is selected, enter the **Number of rays per release** N (dimensionless). The rays are distributed on a plane grid in planes that are approximately tangential to the selected boundaries (for a plane boundary, you get a uniform distribution).

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list — **Expression** (the default), **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **Expression** a single ray is released in the specified direction. Enter coordinates for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless) based on space dimension. Select the **Specify tangential and normal vector components** check box to specify the initial direction using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).
- For **Hemispherical** a number of rays are released at each point, sampled from a hemispherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension. Select the **Specify tangential and normal vector components** check box to specify the hemisphere axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).
- For **Conical** a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Cone axis r** based on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is $\pi/3$ radians. Select the **Specify tangential and normal vector components** check box to specify the cone axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).
- The **Lambertian** option is only available in 3D. A number of rays are released at each point, sampled from a hemisphere in wave vector space with probability density based on the cosine law. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension. Select the **Specify tangential and normal vector**

components check box to specify the hemisphere axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n)

When the **Specify tangential and normal vector components** check box is selected, arrows indicating the normal direction on the selected boundaries will appear in the Graphics window.

Note that the normal direction may be opposite the built-in variable for the boundary normal (e.g.`nx`, `ny`, `nz`) to ensure that a positive value causes rays to be released into the simulation domain. This often occurs when the Inlet is applied to exterior boundaries.

When the normal direction used by the Inlet feature is opposite the normal vector defined by the geometry, the tangential directions are similarly inverted to ensure that the boundary coordinate system is right-handed.

If **Conical** is selected, select an option from the **Conical distribution** list — **Uniform density** (the default), **Specify polar and azimuthal distributions**, **Marginal rays only**, or **Marginal and axial rays only**.

- For **Uniform density** rays are released with polar angles from 0 to the specified cone angle. The rays are distributed in wave vector space so that each ray subtends approximately the same solid angle.
- For **Specify polar and azimuthal distributions** specify the **Number of polar angles** N_ϕ (dimensionless) and the **Number of azimuthal** angles N_θ (dimensionless). Rays are released at uniformly distributed polar angles from 0 to the specified cone angle. A single axial ray (ϕ=0) is also released. For each value of the polar angle, rays are released at uniformly distributed azimuthal angles from 0 to 2π. Unlike other options for specifying the conical distribution, it is not necessary to directly specify the **Number of rays in wave vector space** N_w (dimensionless), which is instead derived from the relation $N_w = N_\phi \times N_\theta + 1$.
- For **Marginal rays only** the rays are all released at an angle α with respect to the cone axis. The rays are released at uniformly distributed azimuthal angles from 0 to 2π.
- For **Marginal and axial rays only** the rays are all released at an angle α with respect to the cone axis, except for one ray which is released along the cone axis. The marginal rays are released at uniformly distributed azimuthal angles from 0 to 2π.

The four **Conical distribution** options are illustrated in Figure 7-1.

For **Hemispherical**, **Conical**, and **Lambertian**, select an option from the **Sampling from Distribution** list — **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray direction vectors are computed using the same algorithm, which seeks to distribute the rays as evenly as possible in wave vector space, whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudo-random numbers.

INITIAL RADII OF CURVATURE

This section is available when:

- the **Intensity computation** is set to **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** in the physics interface **Intensity Computation** section, and
- when **Expression** is selected as the **Ray direction vector**.

Select a **Wavefront shape**. In 3D the available options are **From surface curvature** (the default), **Plane wave**, **Spherical wave**, and **Ellipsoid**. In 2D the available options are **Plane wave** (the default) and **Cylindrical wave**.

- For a **Spherical wave** or **Cylindrical wave**, enter the **Initial radius of curvature** r_0 (SI unit: m).
- For an **Ellipsoid**, enter the **Initial radius of curvature, I** $r_{1,0}$ (SI unit: m) and the **Initial radius of curvature, 2** $r_{2,0}$ (SI unit: m). Also enter the **Initial principal curvature direction, I** $\mathbf{e}_{1,0}$ (dimensionless).

Principal Radii of Curvature

Inlet on Axis

Use the **Inlet on Axis** node to release rays from specified edges along the axis of symmetry in a 2D axisymmetric geometry. It functions like the Inlet node, except that it can only be applied to the axis of symmetry and that the options in the **Ray Direction Vector** section are slightly different.

See Release for information on the following sections: **Release Times, Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL POSITION

Select an **Initial position** — **Uniform distribution** (the default), **Mesh based**, or **Density**. **Mesh based** and **Density** have the same settings as described for the Release node.

If **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The default is 100. The union of the selected boundaries is divided into N segments of approximately equal length, and a ray is placed in the middle of each segment.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list — **Expression** (the default), **Spherical**, or **Conical**.

- For **Expression** a single ray is released in the specified direction. Enter the radial and axial components of the **Ray direction vector** $\mathbf{L}_0$ (dimensionless).
- For **Spherical** a number of rays are released at each point, from the negative z-direction to the positive z-direction in the half-plane with $r > 0$. These rays are sampled from an anisotropic hemispherical distribution in wave vector space, so that rays are more heavily weighted toward the r-direction rather than the $\pm z$-directions. In 3D this would correspond to a spherical distribution of rays such that each ray subtends approximately the same solid angle in wave vector space.

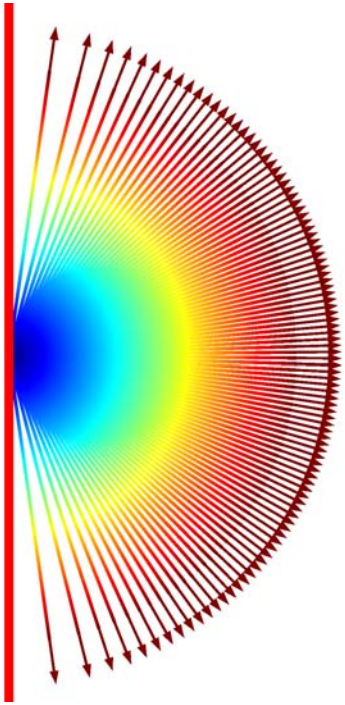

Figure 7-2: The Spherical release of rays causes rays to be weighted more heavily at greater angles from the axis of symmetry.

Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension. Select the **Specify tangential and normal vector components** check box to specify the hemisphere axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).

- For **Conical** a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Cone axis r** based on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is $\pi/3$ radians. Unlike the **Spherical** distribution, the rays in the **Conical** distribution each subtend the same angle in 2D, not the same solid angle in 3D.

For **Hemispherical** and **Conical**, select an option from the **Sampling from Distribution** list — **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray

direction vectors are computed using the same algorithm, which seeks to distribute the rays as evenly as possible in wave vector space, whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudo-random numbers.

Background Velocity

The **Background Velocity** node is used to assign a velocity to the medium in which the rays propagate. When no **Background velocity** node is present, the medium is assumed to be stationary.

BACKGROUND VELOCITY

The default **Velocity field** $\mathbf{u}$ (SI unit: m/s) is **User defined**. Enter values or expressions for the vector components.

Auxiliary Dependent Variable

Use the **Auxiliary Dependent Variable** node to solve additional first-order ordinary differential equations for each ray released.

AUXILIARY DEPENDENT VARIABLE

Enter a **Field variable name**. The default is `rp` and can be changed to anything provided it does not conflict with the name of the variables for the position or momentum degrees of freedom. The name should not conflict with other auxiliary dependent variables.

Enter a **Source** R. The unit of the source depends on the unit of the auxiliary dependent variable and the option selected from the **Integrate** list described below.

Select a way to **Integrate** the equation you have defined — **With respect to time** or **Along ray trajectory**. If you are interested in the residence time of a group of rays in a given system then you can set the **Source** to 1 and set **Integrate** to **With respect to time**. To compute the length of the ray trajectory you can set the **Source** to 1 and set **Integrate** to **Along ray trajectories**.

UNITS

These settings are the same as for Accumulator (Domain).

Release from Edge

Use the **Release from Edge** node to release rays from positions along a set of edges in a 3D geometry.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL POSITION

Select an **Initial position** — **Mesh based** (the default), **Uniform distribution**, or **Density**. Mesh based and Density have the same settings as described for the Release node. If **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The union of the selected edges is divided into N segments of approximately equal length, and a ray is released from the middle of each segment.

Release from Point

Use the **Release from Point** node to release rays from a set of points in a geometry. By default one ray is released at every selected point, although it is possible to release multiple rays at each point if the initial direction or an auxiliary dependent variable is sampled from a distribution.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

Release from Point on Axis

The **Release from Point on Axis** feature is similar to the Release from Point feature, except that it can only release rays from points along the axis of symmetry in a 2D axisymmetric geometry.

See Inlet for information on the following sections: **Ray Direction Vector**.

For all other sections, see Release from Point.

Release from Grid

Use the **Release from Grid** node to release rays from a user-defined grid of points.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL COORDINATES

Enter **Initial coordinates** based on space dimension ($q_{x,0}$, $q_{y,0}$, and $q_{z,0}$ for 3D components) for the ray positions or click the **Range** button () to select and define a range of specific coordinates.

Select an option from the **Grid type** list: **All combinations** (the default) or **Specified combinations**. If **Specified combinations** is selected, the number of initial coordinates entered for each space dimension must be equal, and the total number of rays released is equal to the length of one of the lists of initial coordinates. If **All combinations** is selected, the total number of rays released is equal to the product of the lengths of each list of initial coordinates.

For example, suppose a 2D component includes a **Release from Grid** node with the following initial coordinates:

- $q_{x,0}$ = range(0,1,3)
- $q_{y,0}$ = range(2,2,8)

If **All combinations** is selected, a total of 16 rays will be released, including every possible combination of the initial x- and y-coordinates. If **Specified combinations** is selected, 4 rays will be released with initial positions (0,2), (1,4), (2,6), and (3,8).

Release from Grid on Axis

The **Release from Grid on Axis** feature is similar to the Release from Grid feature, except that it can only release rays from a grid of points along the axis of symmetry in a 2D axisymmetric geometry; that is, $q_{r,0} = 0$ for all released rays.

See Inlet for information on the following sections: **Ray Direction Vector**.

For all other sections, see Release from Grid.

Release from Data File

Use a **Release from Data File** node to specify the initial ray position, direction, and values of auxiliary dependent variables using data from a text file.

See Release for information on the following sections: **Release Times, Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, and **Total Source Power**.

Formatting Guidelines for Files Containing Ray Data

The imported data file should be a text file (`*.txt`) arranged in a spreadsheet format; that is, each row corresponds to a distinct ray and should have the same number of columns as all other rows. Columns can be separated by spaces, tabs, or a combination of the two. Begin a line with the percent ("`%`") character to include comments or empty lines in the data file.

For example, a data file containing the following text would release rays at the positions (0.1, 0.2, 0.6) and (0.2, 0.4, 0.8) in a three-dimensional geometry:

```
% Ray release positions
% qx0 qy0 qz0
0.1 0.2 0.6
0.2 0.4 0.8
```

INITIAL POSITION

Browse your computer to select a text file, then click **Import** to import the data. To remove the imported data, click **Discard**. Enter the **Index of first column containing position data** i to indicate which column represents the first coordinate of the ray position vectors. The default value, 0, indicates the first column.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list — **Expression** (the default), **From file**, **Spherical**, **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **From file**, enter the **Index of first column containing velocity data** i. The default is 3. The columns are zero-indexed; that is, an index of 0 corresponds to the first column.
- For all other settings, the settings are the same as for the Release node.

INITIAL VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

For each of the active **Auxiliary Dependent Variable** nodes in the model, choose an option from the **Distribution function** list: **From file**, **None** (the default), **Normal**, **Lognormal**, **Uniform**, or **List of Values**.

- When **From file** is selected, enter the **Index of column containing data**. The default value is 3. The columns are zero-indexed; that is, an index of 0 corresponds to the first column.

For all other options, the settings are the same as for the Release node.

Ray Continuity

Use the **Ray Continuity** node to specify that rays should cross a pair boundary as if it were invisible. Pair boundaries appear when the geometry sequence ends in **Form Assembly** instead of **Form Union**. Such boundaries require special handling because the mesh elements on either side of the pair boundary aren't required to match up exactly.

PAIR SELECTION

Select one or more identity pairs to allow rays to cross between the source and destination boundaries of these pairs. Such identity pairs are typically created automatically on interior boundaries when the geometry sequence ends in a **Form Assembly** node instead of a **Form Union** node.

The **Ray Continuity** node does not cause the ray position components to change discontinuously; the source and destination boundaries for the identity pairs must be overlapping.

If the geometry sequence ends in a **Form Union** node, the **Ray Continuity** node usually isn't necessary because rays can freely cross interior boundaries where no boundary condition has been applied.

Ray Termination

Use the **Ray Termination** feature to remove rays from the model as they exit the geometry, or to annihilate rays of sufficiently low intensity or power.

The **Ray Termination** feature can terminate rays at the exact time at which their intensity or power reaches the specified threshold, if there exists a closed-form analytic expression for this time. Such a closed-form analytic solution exists if:

- The rays represent planar wavefronts in an absorbing medium, or
- The rays represent spherical or ellipsoidal wavefronts in a non-absorbing medium.

Otherwise, the rays are stopped at the first time step for which the intensity or power is less than the threshold value.

TERMINATION CRITERIA

If **Compute intensity** or **Compute intensity in graded media** is selected from the **Intensity computation** list in the physics interface **Intensity Computation** section, enter the **Threshold intensity** I_{th} (SI unit: W/m^2). The default is `1e-3[W/m^2]`. Rays with intensity lower than the threshold will be annihilated.

If **Compute intensity and power** or **Compute intensity and power in graded media** is selected from the **Intensity computation** list, it is possible to terminate rays based on the ray intensity or power, which are stored as separate degrees of freedom for each ray. Selection an option from the **Termination Criteria** list: **Intensity** (the default), **Power**, or **Intensity and power**. Based on this selection, enter the following:

- For **Intensity** or **Intensity and power**, enter the **Threshold intensity** I_{th} (SI unit: W/m^2). The default is `1e-3[W/m^2]`.
- For **Power** or **Intensity and power**, enter the **Threshold power** Q_{th} (SI unit: W). The default is `1e-3[W]`.

Regardless of whether the ray intensity is computed, select an option from the **Spatial extents of ray propagation** list: **Bounding box, from geometry** (the default), **Bounding box, user defined**, or **None**. This list controls whether rays are terminated based on their positions.

- For **Bounding box, from geometry**, a ray is immediately terminated if it reaches the surface of the bounding box of the geometry, defined as the smallest rectangle or

rectangular prism with edges parallel to the axes of the global coordinate system that contains all of the geometric entities. This bounding box is extended by 5% in each direction so that rays can still interact with boundaries that coincide with the perimeter of the bounding box. Therefore rays can propagate a short distance away from the geometry before disappearing.

- For **Bounding box, user defined**, a ray is immediately terminated if it reaches the surface of a box of user-specified dimensions. To define the box, enter maximum and minimum values of each coordinate.
- For **None**, the rays that propagate outward from the geometry will continue to propagate until the study is complete. If the maximum path length for the **Ray Tracing** study step is too large, this may result in trajectories that are extremely long relative to the characteristic geometry size.

Ray Termination Theory

Ray Detector

Use the **Ray Detector** feature to compute information about rays that are located in a set of selected domains or on a set of selected boundaries. The detector may detect all rays or only the rays released by a specified release feature. Computed variables are the number of rays transmitted, the transmission probability, and a logical expression which can be used to filter the rendered rays during results processing.

The **Ray Detector** feature is also capable of computing the total transmitted ray power. If **Compute intensity and power** or **Compute intensity and power in graded** media is selected from the **Intensity computation** list in the physics interface **Intensity Computation** section, then a global variable for the total power transmitted to the detector is available.

RAY DETECTOR

Select an option from the **Release feature** list—**All** (the default), or any of the features which are capable of releasing rays. When **All** is selected, the variables computed are summed over all release features in the model.

Modeling with the Ray Acoustics Interface

In this section:

Mixed Diffuse and Specular Wall Conditions

At a wall where both specular reflections and diffuse reflections occur it is common to define the scattering coefficient s. This coefficient relates the amount of energy that is specularly E_s and diffusely E_d reflected to the (total) absorption coefficient α. The normalized total reflected energy is E_t. The relations between these are

$$E_t = 1-\alpha = E_s + E_d$$
$$E_s = (1-\alpha)(1-s) = 1-\alpha_s$$
$$E_d = (1-\alpha)s = 1-\alpha_d$$

where α_s is the specular absorption coefficient and α_d is the diffuse absorption coefficient.

In the **Ray Acoustics** interface the mixed reflection condition is treated using a Monte Carlo like approach. When a ray hits a mixed wall condition the ray is either diffusely or specularly reflected according to the **Probability of specular reflection** γ_s. This means that the probability plays the role of the scattering coefficient, by setting γ_s = 1-s. The **Specular absorption coefficient** and the **Diffuse absorption coefficient** should both be set equal to the total absorption α.

Assigning Directivity to a Source

Acoustic sources often have a certain directivity, for example, the directivity of a loudspeaker or even the directivity of a human speaker. When setting up sources in a ray acoustics model this directivity can be specified. In order to do that you need to modify the source power expression in, for example, an **Inlet** feature (with tag `rac.inl1`) where the **Ray direction vector** has been set to **Hemispherical**.

If you have a directivity and sensitivity function `f(alpha, beta, freq)` with `alpha` the angle in *xy*-plane, `beta` the angle in *xz*-plane, and `freq` the ray frequency, then the function to modify the ray power/intensity is as follows:

```
Q0*f(alpha,beta,freq)*rac.inl1.Ntf/rac.racop1(f(alpha,beta,freq))
```

where `Q0` is the base intensity, `rac.inl1.Ntf` is the total number of rays released, and `rac.racop1(f)` weighs the function `f`, for the normalization of the task.

Stopping Rays for a Given Conditions

For some ray acoustics applications it can be advantageous to stop rays once their intensity falls below a certain threshold. This can be simply achieved at walls using the **Primary ray condition** with an **Expression**. Here you can enter a logical expression that must be satisfied for a ray to undergo a certain type of ray-wall interaction. For the intensity specify a condition of the type `rac.I<threashold`. Then the ray is either reflected according the selected condition or it can **Freeze** or **Disappear**, depending on your selection.

Mesh Guidelines

When setting up a mesh for a Ray Acoustics simulation the following guidelines should be followed:

- For modeling ray propagation within domains and interaction with flat surfaces, the mesh can be very coarse.
- When rays may interact with curved boundaries (either reflection or refraction), the accuracy of the reinitialized ray trajectory (that is, the reinitialized wave vector $\mathbf{k}$) depends on the mesh. To create a very fine mesh on curved surfaces without making the mesh unnecessarily fine elsewhere, use the **Curvature factor** in the **Size** settings window. The closer this number is to zero, the finer the mesh will be on a curved surface. A value of 0.1 seems adequate for most applications. If there are small,

tightly curved surfaces in the model, it might also be necessary to reduce the Minimum element size to allow the small curvature factor to do its job.

- When computing ray intensity, the accuracy of the reinitialized intensity at a curved wall or material discontinuity usually doesn't depend on the level of mesh refinement because it uses surface curvature variables that are based directly on the geometry, not on the mesh. However, if ALE is included in the model, the accuracy of the surface curvature does depend on the mesh.
- Domains that are not included in the selection for the Ray Acoustics interface do not need to be meshed. However, a boundary mesh must always be defined wherever any boundary condition is applied. In addition, it is assumed that the speed of sound and density in all deselected domains are the **Speed of sound in exterior domains** and **Density of exterior domains**, which are specified in the physics interface **Ray Release and Propagation** section. Thus, media with graded or temperature-dependent material properties must always be included in the selection of the Ray Acoustics interface, and they must always be meshed.
- **Accumulator** features (both on surfaces and on domains) work by defining additional dependent variables using constant shape functions. When using an **Accumulator** on a **Wall**, for example, the contribution from each ray is distributed uniformly over whatever boundary element it hits. If the mesh on the boundary is very fine, this will make the value of the accumulated variable look noisy unless the number of rays hitting the boundary is much larger than the number of boundary elements.

REPRESENTATION OF CURVED SURFACES

When rays reach the boundaries of geometric entities in a model, they do not interact with an exact parameterized representation of the geometry. Rather, they propagate through the mesh elements that discretize the modeling domain and interact with the boundary elements that cover the surfaces of the geometric entities.

When the surfaces of the geometry are flat, the shape of the surface mesh is indistinguishable from the shape of the geometric entities themselves. Therefore, the fact that rays interact with the mesh instead of the geometry does not introduce any discretization error, and it is possible to accurately compute ray trajectories even when the mesh is extremely coarse.

Curved surfaces in the geometry, however, usually incur a significant amount of discretization error when predicting how rays will interact with them. The time and location at which the ray interacts with the boundary mesh element might be slightly different from the time at which it would have interacted with an exact representation of the surface. In addition, the tangential and normal directions on the boundary mesh

element may differ from the tangential and normal directions on the surface, affecting the accuracy of boundary conditions that involve the tangential and normal directions, such as the **Specular reflection** condition.

The order of the curved mesh elements used to determine the geometry shape is controlled by the **Geometry shape order** list in the **Model Settings** section of the **Settings** window for the main **Component** node. If **Automatic**, the default, is selected, the curved mesh elements are usually represented by quadratic curves; in some cases, linear functions are used to prevent inverted mesh elements from being created.

The effect of the geometry shape order is most notable on a coarse mesh, as shown in Figure 7-3. The mesh elements are shown as pale gray lines in the background and the ray trajectories are represented as thick red arrows. The rays initially propagate downward and are specularly reflected by a parabolic surface. If **Linear** is selected from the **Geometry shape order** list, all rays that hit the same boundary element are specularly reflected in the same direction, as shown on the left. Even though the bottom surface is parabolic, the rays don't all intersect at a single focus due to the discretization error. If **Quadratic** or **Automatic** is selected, rays that hit the same boundary element can still be reflected in different directions because the tangential and normal directions can vary along the surface of the curved element. As a result, the rays reflected by the parabolic surface all intersect at a well-defined focal point as expected.

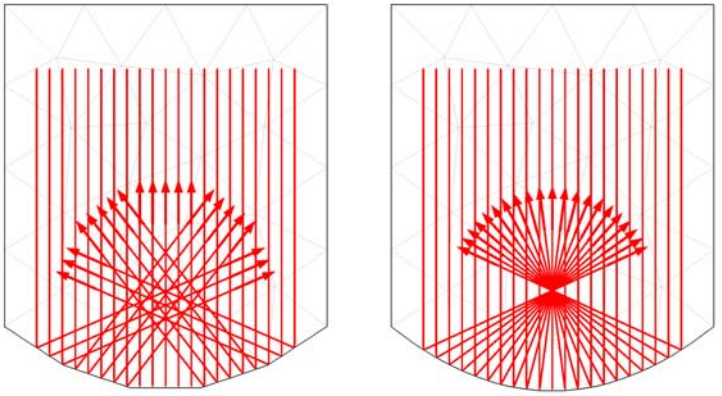

Figure 7-3: Comparison of rays being specularly reflected at a curved boundary represented using linear elements (left) and quadratic elements (right).

RAY TRACING IN AN IMPORTED MESH

It is also possible to compute ray trajectories in an imported mesh. The mesh can be imported from a COMSOL Multiphysics file (`.mphbin` for a binary file format or `.mphtxt` for a text file format) or from a NASTRAN file (`.nas`, `.bdf`, `.nastran`, or `.dat`).

If the mesh is imported from a COMSOL Multiphysics file, the imported mesh always uses linear geometry shape order for the purpose of modeling ray-boundary interactions, even if the model used to generate the mesh had a higher geometry shape order.

If the mesh is imported from a NASTRAN file, the ray-boundary interactions may be modeled using either linear or higher geometry shape order. If **Export as linear elements** is selected when generating the NASTRAN file, or if **Import as linear elements** is selected when importing the file, then linear geometry shape order will be used.

Component Couplings

The purpose of a model is often to compute the sum, average, maximum value, or minimum value of a quantity over a group of rays, such as the average intensity or the maximum path length. An instance of the Ray Acoustics interface with Name `<name>` creates the following four component couplings:

- `<name>.<name>op1(expr)` evaluates the sum of the expression `expr` over the rays. The sum includes all rays that are active, frozen, or stuck to boundaries. It excludes rays that have not yet been released and those that have disappeared.
- `<name>.<name>op_all1(expr)` evaluates the sum of the expression `expr` over all rays, including rays those that are not yet released or have disappeared. Since the coordinates of unreleased and disappeared rays are not-a-number (`NaN`), the sum may return `NaN` if the model includes unreleased or disappeared rays. An expression such as `rac.racop1(isnan(qx))` can be used to compute the total number of unreleased and disappeared rays.
- `<name>.<name>aveop1(expr)` evaluates the average of the expression `expr` over the active, frozen, and stuck rays. Unreleased and disappeared rays contribute to neither the numerator nor the denominator of the arithmetic mean.
- `<name>.<name>aveop_all1(expr)` evaluates the average of the expression `expr` over all rays. It is likely to return `NaN` if the model includes unreleased or disappeared rays.

- `<name>.<name>maxop1(expr)` evaluates the maximum value of the expression `expr` over all active, frozen, and stuck rays.
- `<name>.<name>maxop_all1(expr)` evaluates the maximum value of the expression `expr` over all rays.
- `<name>.<name>maxop1(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the maximum value of the expression `expr` out of all active, frozen, and stuck rays. For example, the expression `rac.racmaxop1(rac.I, qx)` would evaluate the x-coordinate `qx` of the ray with the greatest intensity `rac.I`.
- `<name>.<name>maxop_all1(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the maximum value of the expression `expr` for all rays, including disappeared and unreleased rays.
- `<name>.<name>minop1(expr)` evaluates the minimum value of the expression `expr` over the active, frozen, and stuck rays.
- `<name>.<name>minop_all1(expr)` evaluates the minimum value of the expression `expr` over all rays.
- `<name>.<name>minop1(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the minimum value of the expression `expr` out of all active, frozen, and stuck rays. For example, the expression `rac.racminop1(rac.I, qx)` would evaluate the x-coordinate `qx` of the ray with the minimum intensity `rac.I`.
- `<name>.<name>minop_all1(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the minimum value of the expression `expr` for all rays, including disappeared and unreleased rays.

An instance of the Ray Acoustics interface with the default name `rac` defines the built-in component couplings shown in Table 7-3.

TABLE 7-3: BUILT-IN COMPONENT COUPLINGS FOR THE RAY ACOUSTICS INTERFACE

NAME	DESCRIPTION
`rac.racop1(expr)`	Sum over rays
`rac.racop_all1(expr)`	Sum over all rays
`rac.racaveop1(expr)`	Average over rays
`rac.racaveop_all1(expr)`	Average over all rays
`rac.racmaxop1(expr)`	Maximum over rays
`rac.racmaxop_all1(expr)`	Maximum over all rays
`rac.racminop1(expr)`	Minimum over rays
`rac.racminop_all1(expr)`	Minimum over all rays

TABLE 7-3: BUILT-IN COMPONENT COUPLINGS FOR THE RAY ACOUSTICS INTERFACE

NAME	DESCRIPTION
rac.racmaxop1(expr, evalExpr)	Evaluate at maximum over rays
rac.racmaxop_all1(expr, evalExpr)	Evaluate at maximum over all rays
rac.racmaxop1(expr, evalExpr)	Evaluate at minimum over rays
rac.racmaxop_all1(expr, evalExpr)	Evaluate at minimum over all rays

Using Ray Detectors

A **Ray Detector** feature is a domain or boundary feature that provides information about rays arriving on a set of selected domains or surfaces from a release feature. Such quantities include the number of rays transmitted, the transmission probability, and a logical expression for ray inclusion. The feature provides convenient expressions that can be used in the Filter node of the Ray Trajectories plot, which allows only the rays which reach the ray detector selection to be visualized. The following variables are provided by the **Ray Detector** feature, with the feature tag `<tag>`:

- `<tag>.Ntf` is number of transmitted rays from the release feature to the ray detector at the end of the simulation.
- `<tag>.alpha` is the transmission probability from the release feature to the ray detector.
- `<tag>.rL` is a logical expression for ray inclusion. This can be set in the **Filter** node of the **Ray Trajectories** plot in order to visualize the rays which connect the release feature to the detector.

The **Ray Detector** feature only creates variables, which do not affect the solution. Therefore, they can be added to a model without the need to re-compute the solution, it just needs to be updated. To do this, right click on the **Study** node and select **Update Solution**. The new variables described above will be immediately available for results processing.

Impulse Response, Results Plots, Data Sets, and Derived Values

A variety of dedicated postprocessing tools are available with this physics interface. They are listed below and referred to in the *COMSOL Multiphysics Reference Manual*.

IMPULSE RESPONSE

In ray acoustics applications some of the most important postprocessing results count the impulse response and the energy impulse response. Constructing such a plot requires a few steps.

1. Set the **Intensity computation** to **Compute intensity**. If the media are graded (that is, the speed of sound can change continuously as a function of position), instead choose **Compute intensity in graded media**.
2. Select the **Compute phase** and **Store ray status data** check boxes.
3. Define a receiver surface, typically a small sphere, and apply a **Wall** boundary condition with the **Wall condition** set to **Freeze**.
4. Define a **Ray** data-set with selections restricted to the boundaries of the receiver.
5. In a **1D Plot Group** set up a **Ray** plot that refers to the above data set. Plot the desired quantity, this can be the pressure `rac.p`, the ray intensity `rac.I`, or the sound pressure level `rac.Lp`. In the **x-Axis Data** enter the stop time of the rays `rac.st`.

PLOTS

In the *COMSOL Multiphysics Reference Manual*:

- Ray (Plot)
- Ray Trajectories
- Filter for Ray and Ray Trajectories
- Plot Groups and Plots

RAY DATA SET

In the *COMSOL Multiphysics Reference Manual*:

- Ray (Data Set)
- Data Sets

RAY EVALUATION DERIVED VALUES

In the *COMSOL Multiphysics Reference Manual*:

- Ray Evaluation
- Derived Values and Tables

Theory for the Ray Acoustics Interface

The Ray Acoustics Interface theory is described in this section:

- Introduction to Ray Acoustics
- Material Discontinuity Theory
- Intensity and Wavefront Curvature
- Intensity and Phase Reinitialization
- Wavefront Curvature Calculation in Graded Media
- Attenuation Within Domains
- Ray Termination Theory
- Accumulator Theory: Domains
- Accumulator Theory: Boundaries
- Sound Pressure Level Calculation Theory
- References for the Ray Acoustics Interface

Introduction to Ray Acoustics

It is often convenient to express an acoustic wave in a stationary fluid as a plane wave of the form

$$\phi = ae^{i\Psi}$$

where ϕ (SI unit: m^2/s) is the velocity potential of the fluid, a is its amplitude of the velocity potential, and Ψ (SI unit: rad) is the phase. When the distance from any source is many orders of magnitude larger than the wavelength of the acoustic wave, the wave may be assumed to be locally plane; that is,

$$\Psi = \mathbf{k} \cdot \mathbf{q} - kct$$

where $\mathbf{k}$ (SI unit: rad/m) is the wave vector, $\mathbf{q}$ (SI unit: m) is the position vector, c (SI unit: m/s) is the speed of sound in the medium, and t (SI unit: s) is the time.

While the velocity potential is defined with a stationary fluid in mind, it is possible to apply the same treatment to a homogeneous fluid moving at velocity $\mathbf{u}$ (SI unit: m/s)

by first formulating the equation of the acoustic wave in a coordinate system in which the fluid is stationary. This results in a more general form of the acoustic wave equation,

$$\Psi = \mathbf{k} \cdot \mathbf{q} - (kc + \mathbf{k} \cdot \mathbf{u})t$$

By defining the angular frequency of the wave as

$$\omega = ck + \mathbf{u} \cdot \mathbf{k}$$

it follows that the equation of the acoustic wave may be expressed in the Hamiltonian form

$$\frac{d\mathbf{k}}{dt} = -\frac{\partial \omega}{\partial \mathbf{q}}$$
$$\frac{d\mathbf{q}}{dt} = \frac{\partial \omega}{\partial \mathbf{k}}$$

This is a mixed time-frequency formulation that has many advantages. See, for example, Ref. 1 and Ref. 5 for further details. It allows a ray acoustic problem to be broken down into its Fourier components, solving for the propagation of one frequency component per ray. Another benefit is the simple use of frequency dependent boundary conditions.

Initial Conditions — Direction

There are several options available for specifying the initial direction of rays. These options work by assigning values to the degrees of freedom corresponding to the wave vector $\mathbf{k}$ (SI unit: rad/m) of each ray.

EXPRESSION

The default is to enter an expression for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless). This vector need not have a magnitude of unity because it is always normalized automatically. The initial wave vector is

$$\mathbf{k} = \frac{\omega \mathbf{L}_0}{c|\mathbf{L}_0|}$$

where ω (SI unit: 1/s) is the angular frequency and c (SI unit: m/s) is the speed of sound in the medium.

SPHERICAL

When **Spherical** is selected the initial wave vectors are sampled from a distribution in wave vector space at each release point. The number of rays released from each point is usually equal to the specified value N_w (dimensionless), although it may be larger if the initial values of any auxiliary dependent variables are also sampled from a distribution.

In 2D the initial wave vector components are

$$k_x = \frac{\omega}{c}\cos\theta$$

$$k_y = \frac{\omega}{c}\sin\theta$$

where θ goes from 0 to 2π in N_w steps. In 3D the initial wave vector components are sampled according to the expressions

$$k_x = \frac{\omega}{c}\cos\theta\sin\varphi$$

$$k_y = \frac{\omega}{c}\sin\theta\sin\varphi$$

$$k_z = \frac{\omega}{c}\cos\varphi$$

The azimuthal angle θ is uniformly distributed from 0 to 2π. The polar angle φ is sampled from the interval $[0, \pi]$ with probability density proportional to $\sin\varphi$. The polar angle is arbitrarily chosen as the angle that the initial wave vector makes with the positive z-axis, but any direction could be chosen because the sphere is isotropic. Therefore each ray subtends approximately the same solid angle in wave vector space.

HEMISPHERICAL

The **Hemispherical** option is the same as the **Spherical** option, except that in 2D θ goes from 0 to π and in 3D φ goes from 0 to $\pi/2$. The angle (θ in 2D or φ in 3D) is measured from the direction given by the **Hemisphere axis** setting.

CONICAL

The **Conical** option is the same as the **Spherical** option, except that in 2D θ goes from 0 to α and in 3D φ goes from 0 to α. The angle (θ in 2D or φ in 3D) is measured from the direction given by the **Cone axis** setting.

LAMBERTIAN

The **Lambertian** option releases rays within a hemisphere in 3D, but the probability distribution function is different from that of the **Hemispherical** option. Recall that for an isotropic hemispherical distribution the polar angle φ has a probability density proportional to $\sin\varphi$; for the Lambertian distribution the probability density is instead proportional to $\sin\varphi\cos\varphi$. Because of this extra cosine term, distributions following this probability density are said to follow Lambert's cosine law.

Material Discontinuity Theory

At a Material Discontinuity between two different media, the wave vector is reinitialized using Snell's law. First, the angle of incidence θ_i is computed:

$$\theta_i = \operatorname{acos}\left(\frac{\mathbf{n}_i \cdot \mathbf{n}_s}{|\mathbf{n}_i||\mathbf{n}_s|}\right)$$

where $\mathbf{n}_i$ is a unit vector in the direction of the incident ray and $\mathbf{n}_s$ is a unit vector normal to the material discontinuity.

At a boundary between two isotropic, non-absorbing media, the refracted ray propagates in the direction $\mathbf{n}_t$ given by the following relations:

$$\mathbf{n}_t = \eta\mathbf{n}_i + \gamma\mathbf{n}_s$$

$$\gamma = -\eta\cos\theta_i + \cos\theta_t$$

$$\eta = \frac{c_2}{c_1}$$

$$\theta_t = \operatorname{asin}(\eta\sin\theta_i)$$

where the ray propagates from the medium with speed of sound c_1 into the medium with speed of sound c_2.

If the maximum number of secondary rays has not yet been released, a reflected ray is also released at the material discontinuity with the initial direction

$$\mathbf{n}_r = \mathbf{n}_i - 2\mathbf{n}_s\cos\theta_i$$

If the ray intensity is computed, the release of secondary rays may be suppressed when the intensity of the reflected ray is less than the user-defined Threshold intensity I_{th} (SI unit: W/m^2).

Intensity and Wavefront Curvature

When the **Intensity computation** is set to **Compute intensity** or **Compute intensity and power** in the physics interface **Intensity Computation** section, the intensity and polarization of each ray is computed along its trajectory. The method used to compute the intensity treats each ray as a propagating wavefront. The wavefronts are assumed to subtend constant solid angles within each domain, which is valid only when the domains are homogeneous. The following auxiliary dependent variables are defined for each ray:

In 3D:

- The initial ray intensity I_0, which is reinitialized at material discontinuities and walls.
- Two initial principal radii of curvature, $r_{1,i}$ and $r_{2,i}$.
- Two principal radii of curvature, r_1 and r_2, which represent the maximum and minimum radii of curvature of the intersection of the wavefront with an arbitrary plane.
- Three components of a unit vector $\mathbf{e}_1$ in the direction corresponding to the first principal radius of curvature. This information is used to reinitialize the principal radii of curvature at curved boundaries.

In 2D:

- The initial ray intensity I_0.
- One initial principal radius of curvature, $r_{1,i}$.
- One radius of curvature r_1 of the wavefront. All wavefronts are assumed to be converging or diverging cylindrical waves, so it is not necessary to define a second radius of curvature.

If the **Intensity computation** is set to **Compute intensity in graded media** or **Compute intensity and power in graded media**, the intensity of each ray is computed along its trajectory. The method used to compute the intensity treats each ray as a propagating wavefront. Additional variables are defined to enable the calculation of wavefront curvature in graded media. The following auxiliary dependent variables are defined for each ray:

In 3D:

- The intensity help variable Γ.
- Two principal curvature calculation help variables α_1 and α_2 and the rotation angle ϕ which indicates the orientation of the principal curvature directions. An additional

help variable is used internally to detect poles in the local coordinate system definition and to redefine the local coordinate system accordingly.

In 2D:

- The intensity help variable Γ.
- One principal curvature calculation help variable α_1.

PRINCIPAL RADII OF CURVATURE

In 3D models, each ray is treated as a wavefront for which two principal radii of curvature, r_1 and r_2, are defined. In addition, the values of the two principal radii of curvature are stored as $r_{1,i}$ and $r_{2,i}$ whenever the ray reaches a boundary.

Within domains, the equations

$$\frac{dr_1}{ds} = -1 \qquad \frac{dr_2}{ds} = -1$$

are solved. Negative radii of curvature indicate that the wavefront is expanding as the ray propagates, while positive radii of curvature indicate that the wavefront is converging. A continuous locus of points at which either of the principal radii of curvature equals zero is called a caustic surface. The unit vector $\mathbf{e}_1$ is defined so that r_1 is the radius of curvature of the intersection of the wavefront with the plane tangent to $\mathbf{e}_1$ and the wave vector $\mathbf{k}$. Similarly, if a vector $\mathbf{e}_2$ is defined so that

$$\mathbf{e}_2 = \frac{\mathbf{k} \times \mathbf{e}_1}{|\mathbf{k} \times \mathbf{e}_1|}$$

Then r_2 is the radius of curvature of the intersection of the wavefront with a plane tangent to $\mathbf{e}_2$ and $\mathbf{k}$. The components of $\mathbf{e}_1$ are stored as auxiliary dependent variables for each ray. The components of $\mathbf{e}_2$ can then be derived from $\mathbf{e}_1$ and $\mathbf{k}$ at any time.

The principal radii of curvature are reinitialized at material discontinuities, and the orientation of the unit vectors $\mathbf{e}_1$ and $\mathbf{e}_2$ may change. If the unit vector in the direction of the incident ray $\mathbf{n}_i$ is not parallel to the surface normal $\mathbf{n}_s$, then it is possible to define a unique tangent plane, called the plane of incidence, that contains the incident, reflected, and refracted rays. The unit vector normal to this plane, $\mathbf{u}_0$, is defined as

$$\mathbf{u}_0 = \frac{\mathbf{n}_i \times \mathbf{n}_s}{|\mathbf{n}_i \times \mathbf{n}_s|}$$

If the ray is normal to the surface, then the incident ray is parallel to the refracted ray and antiparallel to the reflected ray, and $\mathbf{u}_0$ can be any arbitrary unit vector orthogonal

to $\mathbf{n}_i$. In addition to the unit vector normal to the plane of incidence, the following unit vectors tangent to the incident wavefront, refracted wavefront, reflected wavefront, and surface normal are defined:

$$\begin{aligned}\mathbf{u}_i &= \mathbf{n}_i \times \mathbf{u}_0\\ \mathbf{u}_t &= \mathbf{n}_t \times \mathbf{u}_0\\ \mathbf{u}_r &= \mathbf{n}_r \times \mathbf{u}_0\\ \mathbf{u}_s &= \mathbf{n}_s \times \mathbf{u}_0\end{aligned}$$

where the subscripts i, t, and r denote the incident, refracted, and reflected wavefronts, respectively. For a wavefront propagating in a direction $\mathbf{n}$, with principal curvatures $k_1 = 1/r_1$ and $k_2 = 1/r_2$ defined for directions $\mathbf{e}_1$ and $\mathbf{e}_2$, respectively, the principal curvatures in two other orthogonal directions $\mathbf{e}_1'$ and $\mathbf{e}_2'$ (both orthogonal to $\mathbf{n}$) are

$$\begin{aligned}k_1' &= k_1\cos^2\theta + k_2\sin^2\theta\\ k_2' &= k_1\sin^2\theta + k_2\cos^2\theta\\ k_{12}' &= (k_1 - k_2)\cos\theta\sin\theta\end{aligned} \tag{7-1}$$

where θ is the angle of rotation about $\mathbf{n}$ which transforms the vectors $\mathbf{e}_1$ and $\mathbf{e}_2$ to $\mathbf{e}_1'$ and $\mathbf{e}_2'$, respectively. Because k_1' and k_2' are not principal curvatures, it is necessary to include off-diagonal elements of the wavefront curvature tensor equal to k_{12}'.

The following algorithm is used to reinitialize the principal radii of curvature of the wavefront and their orientations. The reinitialization of the curvature variables follows the method of Stavroudis in Ref. 2.

1. Given $\mathbf{n}_i$ and $\mathbf{n}_s$, compute unit vectors in the directions of the reflected and refracted rays, $\mathbf{n}_r$ and $\mathbf{n}_t$.
2. Compute the vectors $\mathbf{u}_0$, $\mathbf{u}_i$, $\mathbf{u}_t$, $\mathbf{u}_r$, and $\mathbf{u}_s$.
3. Compute the angle of rotation $\theta^{(i)}$ needed to transform the local coordinate system with axes parallel to $\mathbf{e}_1$, $\mathbf{e}_2$, and $\mathbf{n}_i$ to a local coordinate system with axes parallel to $\mathbf{u}_0$, $\mathbf{u}_i$, and $\mathbf{n}_i$.
4. Transform the principal curvatures to the local coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_i$, and $\mathbf{n}_i$ by using Equation 7-1, substituting $\theta^{(i)}$ for θ.
5. Given the two principal curvatures of the surface, $k_{1,s}$ and $k_{2,s}$ with directions $\mathbf{e}_{1,s}$ and $\mathbf{e}_{2,s}$, transform the curvature variables to a local coordinate system with axes parallel to $\mathbf{u}_0$, $\mathbf{u}_s$, and $\mathbf{n}_s$. Let the new curvature variables be denoted by $k_{1,s}'$, $k_{2,s}'$, and $k_{12,s}'$.

6 The curvatures of the wavefront and the surface have now been defined in coordinate systems that share the axis $\mathbf{u}_0$ and only differ by a rotation by the angle of incidence θ_i about $\mathbf{u}_0$. Defining the variables η and γ as in Material Discontinuity Theory, compute the curvature variables of the refracted ray in a coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_t$, and $\mathbf{n}_t$ using the equations

$$k_{1,t}' = \eta k_1' + \gamma k_{1,s}'$$

$$k_{12,t}' = \frac{\eta \cos\theta_i}{\cos\theta_t} k_{12}' + \frac{\gamma}{\cos\theta_t} k_{12,s}'$$

$$k_{2,t}' = \frac{\eta \cos^2\theta_i}{\cos^2\theta_t} k_2' + \frac{\gamma}{\cos^2\theta_t} k_{2,s}'$$

7 Obtain the principal curvatures of the refracted ray by rotating the coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_t$, and $\mathbf{n}_t$ by an angle $\theta^{(t)}$ about $\mathbf{n}_t$. The angle $\theta^{(t)}$ is defined as

$$\theta^{(t)} = \frac{1}{2}\operatorname{atan}\left(\frac{2k_{12,t}'}{k_{1,t}' - k_{2,t}'}\right)$$

8 Reinitialize the principal curvatures of the refracted ray:

$$k_{1,t} = k_{1,t}'\cos^2\theta^{(t)} + 2k_{12,t}'\cos\theta^{(t)}\sin\theta^{(t)} + k_{2,t}'\sin^2\theta^{(t)}$$
$$k_{2,t} = k_{1,t}'\sin^2\theta^{(t)} - 2k_{12,t}'\cos\theta^{(t)}\sin\theta^{(t)} + k_{2,t}'\cos^2\theta^{(t)}$$

9 Invert the principal curvatures to obtain the principal radii of curvature of the refracted ray. Rotate $\mathbf{u}_0$ about $\mathbf{n}_t$ by the angle $\theta^{(t)}$ to obtain the reinitialized principal curvature direction $\mathbf{e}_{1,t}$.

10 If a reflected ray is released, compute the curvature variables of the reflected ray in a coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_r$, and $\mathbf{n}_r$ using the equations

$$k_{1,r}' = k_1' - 2k_{1,s}'\cos\theta_i$$

$$k_{12,r}' = -k_{12}' + 2k_{12,s}'$$

$$k_{2,r}' = k_2' - \frac{2}{\cos\theta_i} k_{2,s}'$$

11 Obtain the principal curvatures of the reflected ray by rotating the coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_r$, and $\mathbf{n}_r$ by an angle $\theta^{(r)}$ about $\mathbf{n}_r$. The angle $\theta^{(r)}$ is defined as

$$\theta^{(r)} = \frac{1}{2}\text{atan}\left(\frac{2k_{12,r}'}{k_{2,r}' - k_{1,r}'}\right)$$

I2 Initialize the principal curvatures of the reflected ray:

$$k_{1,r} = k_{1,r}'\cos^2\theta^{(r)} + 2k_{12,r}'\cos\theta^{(r)}\sin\theta^{(r)} + k_{2,r}'\sin^2\theta^{(r)}$$
$$k_{2,r} = k_{1,r}'\sin^2\theta^{(r)} - 2k_{12,r}'\cos\theta^{(r)}\sin\theta^{(r)} + k_{2,r}'\cos^2\theta^{(r)}$$

I3 Invert the principal curvatures to obtain the principal radii of curvature of the reflected ray. Rotate $\mathbf{u}_0$ about $\mathbf{n}_r$ by the angle $\theta^{(r)}$ to obtain the initialized principal curvature direction $\mathbf{e}_{1,r}$.

In 2D components, only one principal radius of curvature is computed; each ray is treated as a cylindrical wave. In addition, no auxiliary degrees of freedom for the principal curvature direction are required because the out-of-plane direction can always be treated as one of the axes of the local coordinate system that defines the orientation of the wavefront.

In 2D axisymmetric model components, radii of curvature are computed for the in-plane direction, i.e. the *rz*-plane, as well as the out-of-plane or azimuthal directions. During ray-boundary interactions, these radii of curvature are reinitialized as if the ray interacted with a 3D surface of revolution, which may have finite radii of curvature in both the in-plane and out-of-plane directions.

INTENSITY CALCULATION

The value of the ray intensity is stored as the auxiliary dependent variable I_0 when a ray is released. At any point along the ray's trajectory, the intensity is equal to

$$I = I_0\frac{r_{1,i}r_{2,i}}{r_1 r_2}$$

In 2D, the second principal radius of curvature r_2 is treated as an arbitrarily large distance that remains constant for each ray.

Intensity and Phase Reinitialization

At a material discontinuity, the intensity of the reflected and refracted rays can be computed using the material properties of the media in the adjacent domains:

$$I_r = I|R|^2$$

$$I_t = I(1 - |R|^2)$$

$$R = \frac{Z_2 \cos\theta_i - Z_1 \cos\theta_t}{Z_2 \cos\theta_i + Z_1 \cos\theta_t} \tag{7-2}$$

$$Z = \rho c_c$$

where ρ is the density of the medium and c_c is the complex speed of sound,

$$c_c = \frac{c}{1 - i\frac{c}{\omega}\alpha}$$

where

- c (SI unit: m/s) is the real speed of sound in the medium,
- ω (SI unit: rad/s) is the angular frequency of the acoustic ray, and
- α (SI unit: 1/m) is the attenuation coefficient.

The subscripts 1 and 2 denote the sides of the boundary into which the reflected and refracted ray propagate, respectively; see Ref. 4.

At a Wall, the reflection coefficient R can be defined explicitly or in terms of the absorption coefficient α or characteristic impedance of absorber Z_1. In terms of the absorption coefficient,

$$R = \sqrt{1 - \alpha}$$

Where the principal square root is used for purposes of phase reinitialization. In terms of the characteristic impedance of absorber,

$$R = \frac{Z_1 \cos\theta / Z_0 - 1}{Z_1 \cos\theta / Z_0 + 1}$$

$$Z_0 = \rho c_c$$

The **Wall** feature also includes built-in options to compute the reflection coefficient for fluid-fluid interfaces, fluid-solid interfaces, and a fluid layer adjacent to a semi-infinite fluid domain. When treating the **Wall** boundary as a **Fluid-fluid interface**, the reflection coefficient is computed using Equation 7-2 as if the wall were a material discontinuity where the fluid properties of the adjacent medium are specified, except that no refracted ray is produced.

If the **Wall** boundary is modeled as a **Fluid-solid interface**, the reflection coefficient is instead defined as

$$R = \frac{Z_{\text{tot}} - \dfrac{Z_0}{\cos\theta_i}}{Z_{\text{tot}} + \dfrac{Z_0}{\cos\theta_i}}$$

where

$$Z_{\text{tot}} = \frac{Z_{p,1}}{\cos\theta_{p,t}}\cos^2(2\theta_{s,t}) + \frac{Z_{s,1}}{\cos\theta_{s,t}}\sin^2(2\theta_{s,t})$$

where the subscripts p and s refer to the propagation of compressional and shear waves in the adjacent solid domain, respectively. For example, $\theta_{p,t}$ is the refraction angle computed using the compressional complex speed of sound in the adjacent solid domain.

If the **Wall** boundary is modeled as a **Layered fluid half-space**, the boundary is treated as a thin layer of one fluid backed by a semi-infinite domain of a second fluid. The reflection coefficient is

$$R = \frac{\zeta_1(\zeta_2 - \zeta_0) - i(\zeta_1^2 - \zeta_0\zeta_2)\tan\phi}{\zeta_1(\zeta_2 + \zeta_0) - i(\zeta_1^2 - \zeta_0\zeta_2)\tan\phi}$$

$$\zeta_0 = \frac{Z_0}{\cos\theta_i}$$
$$\zeta_1 = \frac{Z_1}{\cos\theta_{t,1}}$$
$$\zeta_2 = \frac{Z_2}{\cos\theta_{t,2}}$$

where the subscripts 1 and 2 refer to the properties of the thin layer and the semi-infinite fluid domain, respectively. Foe example, $\theta_{t,1}$ is the angle of refraction in the thin layer. The angle ϕ is the phase delay in the thin layer,

$$\phi = \frac{\omega}{c_1} h_1 \cos\theta_{t,1}$$

where h_1 (SI unit: m) is the layer thickness.

If the **Rayleigh roughness** model is used, the reflected intensity is multiplied by an additional factor to account for surface roughness:

$$I_r = I|R\exp(-0{,}5\Gamma^2)|^2$$

$$\Gamma = 2k\sigma\cos\theta_i$$

where k (SI unit: rad/m) is the wave vector magnitude of the ray and σ (SI unit: m) is the RMS roughness of the surface.

If the phase of the ray is also computed, the phase shift of the reflected ray at a boundary is

$$\Delta\Psi_r = \arg(R)$$

At material discontinuities, the phase of the refracted ray is unchanged.

Wavefront Curvature Calculation in Graded Media

It is possible to compute the ray intensity by changing **Intensity computation** to **Compute intensity** or **Compute intensity in graded media** in the settings window for the Ray Acoustics interface. The options **Compute intensity and power** and **Compute intensity and power in graded media** can also be used to compute intensity, with the only difference being that these options define an additional auxiliary dependent variable for the total power transferred by the ray. The setting **Compute intensity** is more robust and accurate than **Compute intensity in graded media**, but is only applicable to homogeneous media. The setting **Compute intensity in graded media** can be used for both homogeneous and graded media, but it introduces more numerical error than **Compute intensity**.

ASSUMPTIONS FOR COMPUTING INTENSITY IN HOMOGENEOUS MEDIA

When **Compute intensity** is selected, the ray intensity is computed along each ray path using the following assumptions:

1. The wavefront represented by each ray subtends a constant plane angle (2D) or solid angle (3D), which can change only when rays are reflected or refracted at boundaries.
2. Given the radii of curvature at two positions within the same domain and the intensity at one of these two positions, the intensity at the other position can be computed using the relationship

$$I = I_0 \frac{r_{1,0} r_{2,0}}{r_1 r_2}$$

3 Except when rays are reflected or refracted at boundaries, the principal radii of curvature change at a constant rate:

$$\frac{dr_1}{ds} = \frac{dr_2}{ds} = -1$$

4 The principal curvature directions do not change, except when rays are reflected or refracted at boundaries.

These assumptions are not valid in graded media, in which the speed of sound changes continuously as a function of position. It is possible for the angle between two different rays to change as they propagate through the medium, so the solid angle subtended by the wavefront is no longer constant. As a result, the ray intensity cannot be expressed using the ratio of initial and final principal radii of curvature.

CURVATURE TENSOR DEFINITION IN GRADED MEDIA

The calculation of ray intensity in graded media is based on the concept of a curvature tensor $\mathbf{K}$, defined in terms of the principal curvatures κ_1 and κ_2 and the corresponding principal curvature directions $\mathbf{e}_1$ and $\mathbf{e}_2$:

$$\mathbf{K} = \kappa_1 \mathbf{e}_1 \mathbf{e}_1^T + \kappa_2 \mathbf{e}_2 \mathbf{e}_2^T$$

Because $\mathbf{e}_1$ and $\mathbf{e}_2$ are orthogonal, it follows that κ_1 and κ_2 are eigenvalues of $\mathbf{K}$. It also follows that $\mathbf{K}$ is singular because there is no contribution that is orthogonal to both $\mathbf{e}_1$ and $\mathbf{e}_2$.

The signs of the principal curvatures are chosen so that positive curvature indicates that the wavefront is converging, whereas negative curvature indicates that the wavefront is diverging.

The selection of the coordinate system in which the curvature tensor is defined is crucial. It is convenient to describe the coordinate system so that two of the coordinate axes lie in the plane containing $\mathbf{e}_1$ and $\mathbf{e}_2$, since this reduces the number of nonzero terms in $\mathbf{K}$. Further reduction in the number of nonzero terms can be achieved if $\mathbf{K}$ can be defined in a coordinate system in which $\mathbf{e}_1$ and $\mathbf{e}_2$ are basis vectors. This is possible in 2D because one of the principal curvature directions is always parallel to the out-of-plane direction, but in 3D it is not feasible because the principal curvature directions can change as rays propagate through a graded medium.

In the most general 3D case, the curvature tensor is described using the following symbols, each of which corresponds to a different orthonormal basis:

- $\mathbf{K}_X$: curvature tensor defined in the Cartesian coordinate system with basis vectors $\mathbf{x}$, $\mathbf{y}$, and $\mathbf{z}$.
- $\mathbf{K}_W$: curvature tensor defined in the coordinate system consisting of the two principal curvature directions $\mathbf{e}_1$ and $\mathbf{e}_2$ and the direction of propagation $\mathbf{t}$.
- $\mathbf{K}_S$: curvature tensor defined in a coordinate system in which one basis vector is the direction of propagation $\mathbf{t}$. The other two basis vectors $\mathbf{s}_1$ and $\mathbf{s}_2$ can be determined arbitrarily, as long as they are orthogonal to each other and to $\mathbf{t}$, such that $\mathbf{s}_1 \times \mathbf{s}_2 = \mathbf{t}$. It is more convenient to operate in this coordinate system if the basis vectors can be expressed strictly in terms of $\mathbf{x}$, $\mathbf{y}$, $\mathbf{z}$, and $\mathbf{t}$.

The basis vectors $\mathbf{s}_1$ and $\mathbf{s}_2$ are defined as follows:

$$\mathbf{s}_1 = \frac{\mathbf{t} \times \mathbf{z}}{|\mathbf{t} \times \mathbf{z}|}$$

$$\mathbf{s}_2 = \mathbf{t} \times \mathbf{s}_1$$

To avoid poles in the definition of the basis vectors, the following alternative definitions are used when the rays propagate nearly parallel to the z-axis:

$$\mathbf{s}_1 = \frac{\mathbf{t} \times \mathbf{x}}{|\mathbf{t} \times \mathbf{x}|}$$

$$\mathbf{s}_2 = \mathbf{t} \times \mathbf{s}_1$$

The relationship between $\mathbf{s}_1$, $\mathbf{s}_2$, $\mathbf{e}_1$, and $\mathbf{e}_2$ is given by

$$\mathbf{e}_1 = \mathbf{s}_1 \cos(\phi) - \mathbf{s}_2 \sin(\phi)$$

$$\mathbf{e}_2 = \mathbf{s}_1 \sin(\phi) + \mathbf{s}_2 \cos(\phi)$$

where the rotation angle ϕ is an auxiliary dependent variable that is stored for each ray.

The relationship between $\mathbf{K}_S$ and $\mathbf{K}_W$ is given by

$$\mathbf{K}_S = \mathbf{Q}_0 \mathbf{K}_W \mathbf{Q}_0^T$$

where $\mathbf{Q}_0$ is the rotation matrix:

$$\mathbf{Q}_0 = \begin{bmatrix} \mathbf{s}_1 \cdot \mathbf{e}_1 & \mathbf{s}_1 \cdot \mathbf{e}_2 & \mathbf{s}_1 \cdot \mathbf{t} \\ \mathbf{s}_2 \cdot \mathbf{e}_1 & \mathbf{s}_2 \cdot \mathbf{e}_2 & \mathbf{s}_2 \cdot \mathbf{t} \\ \mathbf{t} \cdot \mathbf{e}_1 & \mathbf{t} \cdot \mathbf{e}_2 & \mathbf{t} \cdot \mathbf{t} \end{bmatrix} = \begin{bmatrix} \cos(\phi) & \sin(\phi) & 0 \\ -\sin(\phi) & \cos(\phi) & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

The relationship between $\mathbf{K}_X$ and $\mathbf{K}_S$ is given by

$$\mathbf{K}_X = \mathbf{Q}\mathbf{K}_S\mathbf{Q}^T$$

where $\mathbf{Q}$ is the rotation matrix:

$$\mathbf{Q} = \begin{bmatrix} \mathbf{x} \cdot \mathbf{s}_1 & \mathbf{x} \cdot \mathbf{s}_2 & \mathbf{x} \cdot \mathbf{t} \\ \mathbf{y} \cdot \mathbf{s}_1 & \mathbf{y} \cdot \mathbf{s}_2 & \mathbf{y} \cdot \mathbf{t} \\ \mathbf{z} \cdot \mathbf{s}_1 & \mathbf{z} \cdot \mathbf{s}_2 & \mathbf{z} \cdot \mathbf{t} \end{bmatrix}$$

In 2D, one of the principal curvature directions is always known, so it is possible to define $\mathbf{s}_1$ and $\mathbf{s}_2$ so that they coincide with $\mathbf{e}_1$ and $\mathbf{e}_2$. The rotation matrix $\mathbf{Q}_0$ is the identity, and the auxiliary dependent variable for the angle ϕ can be omitted.

DERIVATIVES OF THE CURVATURE TENSOR

Following Ref. 3, the derivative of the curvature tensor along the ray trajectory in a graded medium is given by the expression

$$\frac{d\mathbf{K}_S}{ds} = \sum_{i=1}^{5} \mathbf{M}_i$$

where the terms $\mathbf{M}_i$ correspond to the different ways in which the curvature can depend on the medium properties. They are defined using the following expressions:

$$\mathbf{M}_1 = \mathbf{K}_S \cdot \mathbf{K}_S$$

$$\mathbf{M}_2 = -c\left(\mathbf{t} \cdot \nabla_S\left(\frac{1}{c}\right)\right)\mathbf{K}_S$$

$$\mathbf{M}_3 = -c\mathbf{\Pi}\left(\nabla_S \otimes \nabla_S\left(\frac{1}{c}\right)\right)\mathbf{\Pi}$$

$$\mathbf{M}_4 = 2c^2\mathbf{\Pi}\left(\nabla_S\left(\frac{1}{c}\right) \otimes \nabla_S\left(\frac{1}{c}\right)\right)\mathbf{\Pi}$$

$$\mathbf{M}_5 = -c\left[\left(\mathbf{K}\nabla_S\left(\frac{1}{c}\right)\right)\otimes\mathbf{t} + \mathbf{t}\otimes\nabla_S\left(\frac{1}{c}\right)\mathbf{K}\right]$$

where $\mathbf{\Pi}$ is the projection matrix:

$$\mathbf{\Pi} = \mathbf{I} - (\mathbf{t}\otimes\mathbf{t})$$

The gradient operator ∇_S consists of derivatives that are taken with respect to the local coordinates with basis vectors $\mathbf{s}_1$, $\mathbf{s}_2$, and $\mathbf{t}$.

DERIVATIVES OF THE PRINCIPAL CURVATURES

By application of the chain rule, the derivatives of the nonzero elements of $\mathbf{K}_S$ can be expressed in terms of the principal curvatures that occupy the diagonal elements of $\mathbf{K}_W$ and the rotation angle ϕ:

$$\frac{\partial\kappa_1}{\partial s} = \frac{\partial K_{S,11}}{\partial s}\cos(\phi)^2 - 2\frac{\partial K_{S,12}}{\partial s}\sin(\phi)\cos(\phi) + \frac{\partial K_{S,22}}{\partial s}\sin(\phi)^2$$

$$\frac{\partial\kappa_2}{\partial s} = \frac{\partial K_{S,11}}{\partial s}\sin(\phi)^2 + 2\frac{\partial K_{S,12}}{\partial s}\sin(\phi)\cos(\phi) + \frac{\partial K_{S,22}}{\partial s}\cos(\phi)^2$$

$$\frac{\partial\phi}{\partial s} = \frac{1}{\kappa_2-\kappa_1}\left[\frac{\partial K_{S,12}}{\partial s}\cos(2\phi) + \frac{1}{2}\left(\frac{\partial K_{S,11}}{\partial s} - \frac{\partial K_{S,22}}{\partial s}\right)\sin(2\phi)\right]$$

NUMERICAL STABILIZATION

The principal curvatures κ_1 and κ_2 are not ideal choices for the auxiliary dependent variables stored by each ray because their values can become arbitrarily large as rays approach caustics. Similarly, the principal radii of curvature r_1 and r_2 are not ideal choices because their values can become arbitrarily large when a diverging wavefront begins to converge while propagating through a graded medium.

Instead, the auxiliary dependent variables stored by each ray are the help variables α_1 and α_2. In 2D, only one help variable is allocated because the out-of-plane principal radius of curvature is assumed to be infinite. The principal curvatures are expressed in terms of the help variables using the expression

$$\kappa_i = \kappa_0\tan\alpha$$

where κ_0=1 1/m and $i \in [1, 2]$. The derivatives are then related by the expression

$$\frac{d\alpha_i}{ds} = \frac{1}{\kappa_0}\frac{d\kappa_i}{ds}\cos^2\alpha_i$$

Similarly, the intensity may become infinitely large at caustics, and its reciprocal becomes infinitely large as rays undergo attenuation, so a help variable Γ is used to represent the intensity of the ray:

$$I = I_0 \tan\Gamma$$

where I_0=1 W/m^2. The relationship between the ray intensity help variable and the principal curvatures is

$$\frac{d\Gamma}{ds} = \frac{1}{2}\sin(2\Gamma)[\kappa_1 + \kappa_2 - 2\alpha]$$

where α is the attenuation coefficient of the medium.

Attenuation Within Domains

Rays can gradually lose energy as they propagate through absorbing media. For a plane wave propagating through an absorbing medium with attenuation coefficient α (SI unit: 1/m), the intensity decreases exponentially:

$$\frac{dI}{ds} = -2\alpha I$$

The attenuation coefficient is controlled by the option selected from the **Fluid model** list in the **Pressure Acoustics Model** section of the Medium Properties.

If **Linear elastic** is selected, no attenuation occurs.

If **Linear elastic with attenuation** is selected, the attenuation factor is user-defined.

If **Thermally conducting and viscous** is selected, the attenuation factor is defined in terms of the medium properties:

$$\alpha = \frac{\omega^2 b}{2\rho c^3}$$

$$b \equiv \frac{4}{3}\mu + \mu_B + \frac{(\gamma - 1)k}{C_p}$$

where

- c (SI unit: m/s) is the speed of sound in the medium,
- ρ (SI unit: $\mathrm{kg/m^3}$) is the density.
- C_p (SI unit: J/(kg·K)) is the heat capacity at constant pressure.
- γ (dimensionless) is the ratio of specific heats.
- k (SI unit: W/(m·K)) is the thermal conductivity.
- μ (SI unit: Pa·s) is the dynamic viscosity.
- μ_B (SI unit: Pa·s) is the bulk viscosity.

Ray Termination Theory

The Ray Termination feature can annihilate rays when their intensity or power decreases below a specified threshold.

If the ray intensity is computed, then the threshold ray intensity I_{th} (SI unit: $\mathrm{W/m^2}$) can be specified. Then the termination criterion is

$$I < I_{\mathrm{th}}$$

If the ray power is computed, then the threshold ray power Q_{th} (SI unit: W) can be specified. Then the termination criterion is

$$Q < Q_{\mathrm{th}}$$

It is also possible to specify termination criteria based on both intensity and power; in this case, the rays terminate if either criterion is satisfied.

Whenever possible, the **Ray Termination** feature attempts to stop rays at the exact instant when the ray intensity or power reaches a specified threshold value. A closed-form analytic solution for the exact stop time is available under the following circumstances:

- The rays represent planar wavefronts in an absorbing medium, or
- The rays represent spherical or ellipsoidal wavefronts in a non-absorbing medium.

If neither of these special conditions is met, the ray can only terminate if the termination criterion is met at the beginning of a time step taken by the solver; the ray intensity or power at the instant the ray is stopped may be less than the threshold.

PLANAR WAVEFRONTS IN AN ABSORBING MEDIUM

In this special case, the principal wavefront radii of curvature are so large that the ray intensity can only change significantly in an attenuating medium. The exact time t_s (SI unit: s) at which the ray intensity reaches the threshold value is

$$t_s = t_0 + \tau \log\left(\frac{I}{I_{\text{th}}}\right)$$

where

- t_0 (SI unit: s) is the previous time step, reflection time, or refraction time,
- I (SI unit: W/m^2) is the intensity at t_0, and
- I_{th} (SI unit: W/m^2) is the threshold intensity.

If the termination criterion is based on the ray power Q (SI unit: W) instead of ray intensity, the analogous expression for the stop time is

$$t_s = t_0 + \tau \log\left(\frac{Q}{Q_{\text{th}}}\right)$$

The characteristic time for ray attenuation τ (SI unit: s) is

$$\tau = \frac{1}{2\alpha V_g}$$

where

- α (SI unit: 1/m) is the attenuation coefficient of the medium and
- V_g (SI unit: m/s) is the group velocity magnitude.

DIVERGING WAVEFRONTS IN A NON-ABSORBING MEDIUM

In a non-absorbing medium in which the wavefront is non-planar, the ray can only terminate due to the intensity decreasing below the threshold, since the power does not change. The time t_s (SI unit: s) at which the ray intensity reaches the threshold in a non-attenuating, homogeneous medium is

$$t_s = t_0 + \frac{1}{2V_g}\left(r_1 + r_2 + \sqrt{(r_1 - r_2)^2 + 4\frac{I}{I_{\text{th}}}|r_1 r_2|}\right)$$

where

- t_0 (SI unit: s) is the previous time step, reflection time, or refraction time,
- V_g (SI unit: m/s) is the group velocity magnitude,

- r_1 (SI unit: m) is the first principal radius of curvature of the wavefront at t_0,
- r_2 (SI unit: m) is the second principal radius of curvature of the wavefront at t_0,
- I (SI unit: $\mathrm{W/m^2}$) is the intensity at t_0, and
- I_{th} (SI unit: $\mathrm{W/m^2}$) is the threshold intensity.

In 2D models, or in 3D models where the wavefronts are cylindrical (one extremely large radius of curvature and one finite radius of curvature), the stop time is instead

$$t_s = t_0 + \frac{1}{V_g}\left(r_1 + \frac{I}{I_{\mathrm{th}}}|r_1|\right)$$

where r_1 (SI unit: m) is the finite principal radius of curvature of the cylindrical wavefront.

Accumulator Theory: Domains

The Accumulator (Domain) node is used to transfer information from rays to the domains they occupy or pass through. Each **Accumulator** defines a variable, called the accumulated variable, in the selected domains. The accumulated variable is discretized using constant shape functions, so its value is uniform over every mesh element and may be discontinuous between adjacent mesh elements.

The name of the accumulated variable is specified in the **Accumulated variable name** edit field in the **Accumulator Settings** section of the settings window. The default variable name, rpd, will be used in the remainder of this section when referring to the accumulated variable.

ACCUMULATOR TYPE

The options in the **Accumulator type** list are **Density** and **Count**. If **Density** is selected, the source term is divided by the area or volume of the mesh element when calculating each ray's contribution to the accumulated variable. If **Count** is selected, no division by the area or volume of the mesh element occurs.

The equations in the following section are valid for the **Density** type. The corresponding value of the accumulated variable for the **Count** type is

$$\mathrm{rpd}_{\mathrm{count}} = \mathrm{rpd}_{\mathrm{density}} \times V$$

where V is the mesh element volume (in 3D) or area (in 2D).

ACCUMULATION OVER ELEMENTS

When **Elements** is selected from the **Accumulate over** list, the value of the accumulated variable in a mesh element is the sum of the source terms R_i evaluated for all rays in that mesh element:

$$\mathrm{rpd} = \frac{1}{V}\sum_{i=1}^{N} R_i$$

where N is the total number of rays in the element and V is the area or volume of the mesh element. In other words, the contribution of each ray to the accumulated variable is distributed uniformly over the mesh element the ray is in, regardless of the ray's exact position within the element.

If **Elements and time** is selected from the **Accumulate over** list, then the sum of the source terms for rays in the mesh element is used to define the time derivative of the accumulated variable, rather than its instantaneous value:

$$\frac{d(\mathrm{rpd})}{dt} = \frac{1}{V}\sum_{i=1}^{N} R_i$$

Thus the value of the accumulated variable depends on the time history of rays in the mesh element, instead of the instantaneous positions of the rays. As each ray propagates, it will leave behind a trail based on its contributions to the accumulated variables in mesh elements it has traversed. The algorithm for accumulating over time takes into account the fraction of a time step taken by the solver that the ray spends in each mesh element, even if it crosses between elements during the time step.

Accumulator Theory: Boundaries

The Accumulator (Boundary) feature transfers information from rays to the boundaries they hit or pass through. Each **Accumulator** defines a variable, called the accumulated variable, on the selected boundaries. The accumulated variable is discretized using constant shape functions, so its value if uniform over every mesh element and may be discontinuous between adjacent mesh elements.

The name of the accumulated variable is specified in the **Accumulated variable name** edit field in the **Accumulator Settings** section of the settings window. The default variable name, rpb, will be used in the remainder of this section when referring to the accumulated variable.

The options in the **Accumulator type** list are **Density** and **Count**. If **Density** is selected, the source term is divided by the surface area or length of the boundary mesh element when calculating each ray's contribution to the accumulated variable. If **Count** is selected, no division by the surface area or length of the boundary element occurs.

The equations in the following section are valid for the **Density** type. The corresponding value of the accumulated variable for the **Count** type is

$$\mathrm{rpb}_{\mathrm{count}} = \mathrm{rpb}_{\mathrm{density}} \times V$$

where V is the boundary element surface area (in 3D) or length (in 2D).

When **Rays in boundary elements** is selected from the **Accumulate over** list, the accumulated variable in a boundary element gets incremented by the source term R whenever a ray freezes or sticks to the boundary:

$$\mathrm{rpb}_{\mathrm{new}} = \mathrm{rpb} + \frac{R}{V}$$

where division by the mesh element area or length occurs because the accumulator is assumed to be of type **Density**. Thus the source term evaluated for an incident ray is uniformly distributed over the boundary element it freezes or sticks to.

If instead **Ray-wall interactions** is selected from the **Accumulate over** list, then the accumulated variable gets incremented regardless of what type of ray-wall interaction occurs. Thus, it is possible for the same ray to increment the accumulated variable in many different boundary elements, or even in the same element multiple times.

BUILT-IN GLOBAL VARIABLES

By default, the boundary **Accumulator** defines the following global variables:

TABLE 7-4: BUILT-IN GLOBAL VARIABLES FOR BOUNDARY ACCUMULATORS

NAME	EXPRESSION
`<scope>.<name>_ave`	Average of accumulated variable
`<scope>.<name>_int`	Integral of accumulated variable
`<scope>.<name>_max`	Maximum of accumulated variable
`<scope>.<name>_min`	Minimum of accumulated variable
`<scope>.<name>_sum`	Sum of accumulated variable over elements

Here, `<scope>` includes the physics interface name and the names the Accumulator and parent feature. For example, the average of the accumulated variable over a boundary may be called rac`.wall1.bacc1.rpb_ave`, where `rac` is the name of the

Ray Acoustics interface, `wall1` is the name of the parent Wall node, `bacc1` is the name of the Accumulator node, and `rpb` is the accumulated variable name. These variables are all available in the **Add/Replace Expression** menus during results evaluation.

These global variables are computed by defining a set of component couplings on the selection of the parent physics feature, such as the Wall feature to which the Accumulators are added. The following expressions for the global variables are used.

TABLE 7-5: BUILT-IN GLOBAL VARIABLE DEFINITIONS FOR BOUNDARY ACCUMULATORS

NAME	EXPRESSION
`<scope>.<name>_ave`	`<wscope>.aveop(<scope>.<name>)`
`<scope>.<name>_int`	`<wscope>.intop(<scope>.<name>)`
`<scope>.<name>_max`	`<wscope>.maxop(<scope>.<name>)`
`<scope>.<name>_min`	`<wscope>.minop(<scope>.<name>)`
`<scope>.<name>_sum`	`<wscope>.intop(<scope>.<name>/<scope>.meshVol)`

Here, `<wscope>` is the scope of the parent boundary feature, e.g. `goprac.wall1`.

Sound Pressure Level Calculation Theory

It is possible to compute the sound pressure level due to incident and reflected rays at a surface. To do so, **Compute intensity and power** or **Compute intensity and power in graded media** must be selected from the **Intensity computation** list in the settings window for the Ray Acoustics interface. Then the Sound Pressure Level Calculation node can be added as a subnode to the Wall node.

The sound pressure level is computed by first computing the Wall intensity I_w (SI unit: $\mathrm{W/m^2}$). The wall intensity on a given boundary element is

$$I_{w,i} = \frac{1}{A_i}\sum_j Q_j(|R|^2 + 2|R|\cos(\arg(R)) + 1)$$

where the subscript i denotes the ith boundary element, the subscript j indicates the jth ray, Q_j (SI unit: W) is the power transferred by the jth ray, and R (dimensionless) is the reflection coefficient. The sum is taken over all rays that hit the ith boundary element.

The sound pressure level $L_{p,i}$ in the ith element is

$$L_{p,i} = 10\log\left(\frac{\rho c I_{w,i}}{p_{\mathrm{ref,SPL}}^2}\right)$$

where ρ is the density of the fluid, c is the speed of sound, and $p_{\text{ref,SPL}}$ is the reference pressure corresponding to 0 dB. Note that the base-10 logarithm is used.

References for the Ray Acoustics Interface

1. L.D. Landau and E.M. Lifshitz, *Fluid Mechanics, Course of Theoretical Physics, Volume 6*, Butterworth-Heinemann, 2003.

2. O.N. Stavroudis, *The Optics of Rays, Wavefronts, and Caustics*, Academic, 1972.

3. R.M. More and K. Kosaka, "Wave-front curvature in geometrical optics", Phys. Rev. E, vol. 57, no. 5, pp. 6127–6134, 1998.

4. A. D. Pierce, *Acoustics: An Introduction to Its Physical Principles and Applications*, Acoustical Society of America, 1991.

5. F. B. Jensen, W. A. Kuperman, M. B. Porter, and H. Schmidt, *Computational Ocean Acoustics*, Second Edition, Springer, 2011.

The Acoustic Diffusion Equation Interface

The **Acoustic Diffusion Equation** interface ( ), found under the **Acoustics>Geometrical Acoustics** subbranch ( ), solves a diffusion equation for the acoustic energy density. This type of analysis also known as energy finite elements or EFEM. It is applicable for high-frequency acoustics inside coupled rooms when the acoustic fields are diffuse. The interface only exists in 3D. The diffusion of the acoustic energy density depends on the mean free acoustic path and thus on the individual room geometry. Absorption may be applied at walls and a transmission loss may be applied when coupling rooms together. Increased diffusion due to room fitting can be added. Material properties and sources may be specified in frequency bands.

The interface is well suited for quick assessment of sound pressure level distribution inside buildings and other large structures. Compared to a ray acoustics simulation this interface does not include any phase information, direct sound, and early reflections. The interface supports stationary studies for modeling a steady state sound energy or sound pressure level distribution. A time dependent study can be used to determine energy decay curves and reverberation times. The reverberation time of coupled and uncoupled rooms can also be determined using the eigenvalue study. The eigenvalue is directly related to the exponential decay time.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Acoustic Diffusion Model**, **Room**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Acoustic Diffusion Equation** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `ade`.

SOUND PRESSURE LEVEL SETTINGS

See Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface. Note that only **Use reference pressure for air** or **User-defined reference pressure** are available selections.

BAND STUDIED

This section cannot be edited when the **Band Input Type** is **Flat (continuous)**. For all other options under **Band Input Type**, the default N_{study} (dimensionless) is 1, meaning that material properties etc. that are defined in the first band are used. To sweep the bands enter a parameter (defined under Definitions>Parameters) and solve the model using a parametric sweep in the study. In this way solving the model for the desired number of bands.

BAND INPUT TYPE

Select a **Type** — **Flat (continuous)** (the default), **Octave bands**, **1/3 octave bands**, or **User defined bands**. The frequency content of a model solved using the acoustic diffusion equation all lies implicitly in material parameters, absorption properties, sources and so forth. These can be given in a broadband (or for a single frequency) by selecting Flat (continuous) or in bands when given in this way.

- For **Octave bands** the table that displays defaults to 11 rows under **Band number** with the associated **Lower band limit (Hz)**, **Center frequency (Hz)**, and **Upper band limit (Hz)** values.
- For **1/3 octave bands** the table that displays defaults to 32 rows under **Band number** with the associated **Lower band limit (Hz)**, **Center frequency (Hz)**, and **Upper band limit (Hz)** values.
- For **User defined bands** the table that displays defaults to 1 row under **Band number** with default associated values in the **Lower band limit (Hz)**, **Center frequency (Hz)**, and **Upper band limit (Hz)** columns.
  - Under the table click the **Add** () or **Delete** ( ) buttons to edit the table contents. Or right-click a table cell and select **Add** or **Delete**.
  - To save the contents of a table, click the **Save to File** button () and enter a **File name** in the **Save to File** dialog box, including the extension `.txt`. Click to **Save** the text file. The information is saved in space-separated columns in the same order as displayed on screen.
  - Use the **Load from File** button () and **Load from File** dialog box to import data in text files, generated by, for example, a spreadsheet program. Data must be

separated by spaces or tabs (or be in a Microsoft Excel Workbook spreadsheet if the license includes LiveLink™ *for* Excel®).

The lower band limit, center frequency, and upper band limit in the studied band are defined as the global variables `ade.fL`, `ade.fC`, and `ade.fU`, respectively. These can, for example, be used when plotting data in postprocessing.

DEPENDENT VARIABLES

This physics interface defines one dependent variable (field), the **Sound energy density** *w*. If required, edit the name, which changes both the field name and the dependent variable name. If the new field name coincides with the name of another pressure field in the model, the interfaces share degrees of freedom and dependent variable name. The new field name must not coincide with the name of a field of another type, or with a component name belonging to some other field.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Domain, Boundary, and Global Nodes for the Acoustic Diffusion Equation Interface
- Theory for the Acoustic Diffusion Equation Interface

Domain, Boundary, and Global Nodes for the Acoustic Diffusion Equation Interface

The Acoustic Diffusion Equation Interface has these domain, boundary, and point nodes available (listed in alphabetical order).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

	In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Acoustic Diffusion Model
- Continuity[1]
- Destination Selection
- Domain Source
- Fitted Domain
- Initial Values
- Inward Energy Flux
- Mapped Room Coupling
- Point Source
- Room
- Room Coupling
- Wall

[1]Described for the Pressure Acoustics, Frequency Domain interface

Acoustic Diffusion Model

The **Acoustic Diffusion Model** is added by default and is used to define the material properties and the governing diffusion equations.

MATERIAL PROPERTIES

For each of the following properties the default takes values **From material**, or select **User defined** to enter a different value or expression in the text field.

- **Density** ρ (SI unit: kg/m^3). The default is 1.2 kg/m^3.

- **Speed of sound** c (SI unit: m/s). The default is 343 m/s.
- **Volume absorption coefficient** m_a (SI unit: 1/m). The default is 0 1/m. This parameter determines the amount of absorption due to losses in the air. The effects are usually of interest in very large rooms/volumes or at very high frequencies. The built in option **Classical viscous and thermal absorption** defines the frequency absorption coefficient due to viscous and thermal losses. For the **Flat (continuous)** band option it is evaluated at the studied frequency f_{eval} (the default is 1000 Hz) while it is averaged over the studied band for the other Band Input Type.

Room

The **Room** node is added by default and is used to define the mean free path model of each room on a model. A Wall subnode is also added by default. Additional **Wall** subnodes and the Inward Energy Flux subnode are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Each room in a model needs a **Room** node with associated definitions of all wall properties. The volume and surface area of the room is calculated and used to determine the mean free path (depending on the model selected).

MEAN FREE PATH MODEL

Select a **Mean free path model** — **Regular cubic room** (the default), **Long room**, or **User defined**. For **User defined** enter a **Mean free path** λ (SI unit: m).

Classical statistical room acoustic variables that are based on the room geometry and absorption properties of the walls are defined and can be used for postprocessing. See Statistical Model of Reverberation Time for details. They are:

- The total room absorption `ade.A`
- The Sabine reverberation time `ade.T60_Sabine`
- The Eyring-Norris reverberation time `ade.T60_EN`
- The Schroeder frequency of the room `ade.f_Schroeder`

Each room feature (say `ade.room1, ade.room2, etc.`) also has several room variables associated that may be plotted in postprocessing. They are:

- The room volume `ade.room1.V`
- The room surface area `ade.room1.S`
- The mean free path of the room `ade.room1.lambda`

Wall

The **Wall** subnode is available from the context menu (right-click the Room parent node) or from the **Physics** toolbar, **Attributes** menu. It is used to define the wall absorption properties.

WALL ABSORPTION PROPERTIES

The absorption coefficient properties to define depend on the choice of Band Input Type for The Acoustic Diffusion Equation Interface.

- If the **Band Input Type** is **Flat (continuous)** enter an **Absorption coefficient** α (dimensionless).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter an **Absorption coefficient** α (dimensionless) in each row corresponding to the **Band Number**.

Select an **Exchange coefficient model** — **Sabine type** (the default), **Eyring type**, **Modified Eyring type**, or **User defined**. For **User defined** enter an **Exchange coefficient** h (SI unit:

m/s). The choice gives different statistical models which are applicable for different ranges of absorption coefficients. The Sabine type is applicable for $\alpha < 0.2$ while the two others are valid up to an absorption of 1.

Inward Energy Flux

The **Inward Energy Flux** subnode is available from the context menu (right-click the Room parent node) or from the **Physics** toolbar, **Attributes** menu. It is used to define the energy flux that stems from wall sources. This could be a vibrating wall of a structure or the energy flux as measured from external sources to a building.

ENERGY FLUX

The inward energy flux properties to define depend on the choice of Band Input Type for The Acoustic Diffusion Equation Interface.

- If the **Band Input Type** is **Flat (continuous)** enter an **Inward energy flux** J_{in} (SI unit: W/m^2).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter an **Inward energy flux** J_{in} (SI unit: W/m^2) in each row corresponding to the **Band Number**.

Initial Values

The **Initial Values** node adds initial values for the sound energy density. Add more **Initial Values** nodes from the **Physics** toolbar. When calculating the energy impulse response of a system using a time dependent study set the initial value inside a small volume (small domain) or define it in terms of a Gaussian shape.

INITIAL VALUES

Enter a value or expression for the **Sound energy density** w (SI unit: J/m^3) initial value. The default is 0 J/m^3.

Fitted Domain

Use the **Fitted Domain** node to define the fitting parameters. These are parameters that define a region with, for example, furniture inside a room. The domain is characterized in terms if its average statistical properties.

FITTED DOMAIN PARAMETERS

Enter values or expressions for the following:

- **Number density of fitting** n_f (SI unit: $1/\mathrm{m}^3$).
- **Average cross-section of fitting** Q_f (SI unit: m^2).

How you define the **Absorption coefficient of fitting** is based on the choice of Band Input Type for The Acoustic Diffusion Equation Interface.

- If the **Band Input Type** is **Flat (continuous)** enter a value in the text field for the **Absorption coefficient of fitting** α_f (dimensionless). For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter an **Absorption coefficient of fitting** α_f (dimensionless) in each row corresponding to the **Band Number**.

Domain Source

Use the **Domain Source** node to define a source, the right-hand side of the governing diffusion equation. The source can be a constant value defined inside a small volume, it can be a function of time, or it can a function of space. The value corresponds to the total emitted power Q of the source it is automatically transformed into a power density q by dividing with the source domain volume (calculated automatically).

DOMAIN SOURCE

How you define the domain source depends on the choice of Band Input Type for The Acoustic Diffusion Equation Interface.

- If the **Band Input Type** is **Flat (continuous)** enter a **Domain source** Q (SI unit: W).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter a **Domain source** Q (SI unit: W) in each row corresponding to the **Band Number**.

Room Coupling

Use the **Room Coupling** node to couple rooms together allowing an acoustic energy flux back and forth. This can be used to model transmission through walls or openings defined in terms of their transmission loss. The feature is also used to couple rooms that are connected through a large open surface. It is also available as the **Pair Room Coupling**.

TRANSMISSION LOSS

How you define the transmission loss depends on the choice of Band Input Type for The Acoustic Diffusion Equation Interface.

- If the **Band Input Type** is **Flat (continuous)** enter a **Transmission loss** TL (SI unit: dB).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter a **Transmission loss** TL (SI unit: dB) in each row corresponding to the **Band Number**.

CONTINUITY

Select the **Force continuity** check box to constrain the dependent variable resulting in a zero transmission loss. Use this option when rooms are coupled through large open surfaces. Note that the absorption properties of these surfaces need to be set correctly independent of this option.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Mapped Room Coupling

Use the **Mapped Room Coupling** node to couple two rooms when they do not share a common inner boundary. This is, for example, necessary if mid-planes have not been added in the CAD drawing. The condition has the same functionality as the Room Coupling feature except that a mapping is used between the source and destination walls. The Destination Selection subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

TRANSMISSION LOSS

How you define the transmission loss depends on the choice of Band Input Type for The Acoustic Diffusion Equation Interface.

- If the **Band Input Type** is **Flat (continuous)** enter a **Transmission loss** TL (SI unit: dB).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter a **Transmission loss** TL (SI unit: dB) in each row corresponding to the **Band Number**.

CONTINUITY

Select the **Force continuity** check box to ensure continuity resulting in a zero transmission loss.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Destination Selection

The **Destination Selection** subnode is available from the context menu (right-click the Mapped Room Coupling parent node) or from the **Physics** toolbar, **Attributes** menu.

Point Source

Use the **Point Source** node to add a point source to model a source of small dimensions. For a time dependent model it is possible to enter an expression that depend on time.

POINT SOURCE

Enter a **Point source power** q_p (SI unit: W). The default is 0 W.

Modeling with the Acoustic Diffusion Equation Interface

In this section:

The Eigenvalue Study Type

The *Acoustic Diffusion Equation* interface and eigenvalue study will produce the modes that decay exponentially for each room including the effect of couplings to adjacent rooms and wall absorption. The modes are of the form

$$w(\mathbf{x}, t) = w_i(\mathbf{x})e^{-\lambda_i t}$$

where each mode w_i has an associated eigenvalue λ_i (this is the exponential time scale). The eigenvalue directly gives the reverberation time T_{60} of that sub-system

$$T_{60} = \frac{60}{10\log_{10}(e)\lambda_i} \approx \frac{55.3}{4\lambda_i}$$

Typically, there will be as many modes as there are rooms in the building.

A list relating the eigenvalues to the reverberation times is automatically shown under **Derived Values** in the **Results** node. Identify the mode associated with a given reverberation time by looking at the default plot **Sound Energy Density Mode**.

Combined Stationary and Time Dependent Study

By combining a Stationary and a Time Dependent study you can readily model the so-called steady state decay behavior of the system. Use the stationary solution as an initial condition for the transient model. Remember to disable all sources in the transient study step by enabling the **Modify physics tree and variables for study step** option. Evaluating the sound pressure level, `ade.Lp`, in different points will result in the energy decay curves.

Theory for the Acoustic Diffusion Equation Interface

This section contains a small review of the classical models for the prediction of reverberation time in room acoustics followed by the theory background of the acoustic diffusion equation.

In this section:

- Statistical Model of Reverberation Time
- The Acoustic Diffusion Equation
- References for the Acoustic Diffusion Equation Interface

Statistical Model of Reverberation Time

In classical statistical acoustics (see Ref. 9 and Ref. 16) the reverberation time T_{60} (RT) is the time taken for the sound pressure level (created by an impulse source) to decay 60 dB. The reverberation time can be calculated using the Sabine formula

$$T_{60} = \frac{55.3V}{cA}$$

where V is the room volume, c is the speed of sound, and A is the total absorption of all surfaces. The factor 55.3 is a round off of $24 \cdot \ln(10)$. The total absorption is calculated by

$$A = \sum S_i \alpha_i = S\bar{\alpha}$$

where S_i and α_i denote the area and absorption of the i'th area. S is the total surface area and $\bar{\alpha}$ is the average absorption of the room. For large rooms this equation is corrected by the absorption of air

$$T_{60} = \frac{55.3V}{cA + 4m_a V}$$

where m_a is the air absorption constant (see Table 3.1 in Ref. 9). Other versions of the T_{60} reverberation time exist one is the Eyring-Norris given by

$$T_{60} = \frac{55.3V}{-cS\ln(1-\bar{\alpha})}$$

The models for the reverberation time given above are statistical models for diffuse sound field (large number of reflections). At low frequencies the modal behavior of the room becomes important. The lower bound for the application of these statistical models is the Schroeder frequency (see Ref. 10 and Ref. 16) given by

$$f > 2000(\text{m/s})^{3/2}\sqrt{T_{60}/V}$$

The classical expressions given above are available for postprocessing in the Acoustic Diffusion Equation interface.

The Acoustic Diffusion Equation

The acoustic diffusion model is based on the assumption that the volumes (rooms) studied contain scatterers that uniformly scatter the sound field and that the sound field is diffuse (large number of reflections). Using the diffusion of light in a scattering environment as an analogy one can express a diffusion equation for the sound-energy density $w = w(\mathbf{x},t)$ (SI unit: J/m^3). The diffusion equation describes the energy flow from high to low energy regions. Further details about the model equations and boundary conditions are found in papers by Xiang *et al.* (see Ref. 3, Ref. 7, and Ref. 8), and papers by Billon, Valeau and more (see Ref. 5, Ref. 6, Ref. 11, and Ref. 15).

DOMAIN EQUATIONS

The domain diffusion equation for the sound-energy density $w = w(\mathbf{x},t)$ is given by

$$\frac{\partial w}{\partial t} + \nabla \cdot \mathbf{J} + cm_a w = q(\mathbf{x}, t)$$

where the local energy flux vector $\mathbf{J}$ (SI unit: $\text{J/m}^2\text{/s} = \text{W/m}^2$) is defined in the usual way, as

$$\mathbf{J} = -D_t \nabla w$$

The total diffusion coefficient is $D_t = D = \lambda c/3$ (SI unit: $\text{m}^2\text{/s}$), λ is the mean free path (SI unit: m), c is the speed of sound (SI unit: m/s), and m_a is the volumetric absorption coefficient of air (SI unit: 1/m). The volumetric absorption coefficient (or attenuation coefficient) should not be confused with the energy absorption coefficient α used in boundary conditions which is dimensionless. The source term q represents

the spatial sound source (SI unit: $J/m^3/s = W/m^3$). The term $cm_a w$ accounts for volume absorption in air (dissipation). Note that in certain models the $cm_a w$ term accounts for the total absorption at boundaries in a "mean" sense (only using no flux boundary conditions), this approach is not used here.

In the interface, m_a can be given as a user input or it can be define as the "classical viscous and thermal (volumetric) absorption" coefficient (as in pressure acoustics) given by

$$m_{a,\,\text{classical}} = \frac{\omega^2 \mu}{2\rho c^3}\left(\frac{4}{3} + \frac{\gamma - 1}{\text{Pr}}\right)$$

The absorption coefficient is either integrated over the band or it is given at a certain frequency. The volumetric absorption is only important for very large domains/rooms. It is furthermore assumed that $m_a \lambda << 1$ such that the diffusion coefficient remains unchanged.

The mean free path λ is the distance a sound particle on average travels in between reflections. It is related to the average reflections frequency π by $\lambda = c/\pi$. The mean free path is a property of each room and for a regular cubic like room it can be calculated by (the usual convention)

$$\lambda = \frac{4V}{S}$$

where V is the room volume and S the total room surface area (see Ref. 18). The mean free path is defined for every room as it depends on the room geometry and shape. For long rooms like corridors Visentin (see Ref. 13 and Ref. 14) proposes to use another definition of the mean free path given by

$$\lambda = \sqrt{\frac{S}{4\pi}}$$

In general, the mean free path varies for different room shapes. Entering a user defined mean free path based on for example measurements is also possible (see Ref. 1, Ref. 2, and Ref. 17).

Fitted rooms/zones with obstacles (chairs, machines etc.):

In zones with fittings, like furniture or other absorbers, the scatterers are modeled statistically by their number density n_f (SI unit: $1/m^3$), their average cross-section Q_f (SI unit: m^2), and their absorption coefficient α_f (dimensionless). For the domain with scatterers the mean free path becomes

$$\lambda_f = \frac{1}{n_f Q_f}$$

defining the fitting diffusion constant

$$D_f = \frac{c\lambda_f}{3}$$

With these definitions in place, the governing diffusion equation can be modified to take the fitting into account (see, for example, Ref. 15)

$$\frac{\partial w}{\partial t} + \nabla \cdot (-D_t \nabla w) + c\left(m_a + \frac{\alpha_f}{\lambda_f}\right) w = q(\mathbf{x}, t) \qquad D_t = \frac{D_f D}{D_f + D}$$

VALIDITY

The time limit after which the acoustic diffusion model leads to correct results has been discussed by many authors see, for example, Ref. 6, Ref. 12, or Ref. 15. They suggest that a limit of one mean free time λ/c could be considered for the diffusion equation to have physical meaning. Before this time, the high probability of the particles of not having hit a scatterer/surface yet leads to non valid results. Typically this type of models only apply at frequencies above the Schroeder frequency. This is a good "rule of thumb" measure, in several publications results have been seen to match measurements also at lower frequencies. Below the Schroeder frequency the room eigenmodes are important and can be modeled using Pressure Acoustics.

SOURCES

The source term q ($\mathbf{x}$,t) (SI unit of power density J/m^3/s = W/m^3) can be defined as a point source or a volume source.

- Point source located at $\mathbf{x}_s$ are of the type q ($\mathbf{x}$,t) = $P(t)\delta(\mathbf{x} - \mathbf{x}_s)$ where P is the power of the source (SI unit: W).
- Volume/domain sources of volume V_s emitting the power Q (SI unit: W) yielding a power density Q/V_s.
- An impulse source in a transient model is typically modeled as an initial value for $w(\mathbf{x},0)$ in a given small domain (constant in space). The source (initial value) could also be Gaussian shaped in space, here a larger domain.

BOUNDARY CONDITIONS

Wall (with different absorption properties):

At walls, a mixed boundary condition accounting for absorption losses is used (the surface normal $\mathbf{n}$ being outwards to the volume Ω)

$$\mathbf{J} \cdot \mathbf{n} = -D_t \frac{\partial w}{\partial n} = J_{out} = hw$$

where h is the exchange coefficient. Different models exist for this coefficient as given by Xiang *et al.* (see Ref. 3 and Ref. 4). The Sabine type exchange coefficient is

$$h = cA_S = c\frac{\alpha}{4} \qquad \text{for} \qquad \alpha < 0.2$$

This expression is valid for surfaces where the absorption coefficient is smaller than 0.2. The Eyring type model is defined by

$$h = cA_E = c + \frac{-\ln(1-\alpha)}{4} \qquad \text{for} \qquad \alpha < 1$$

This model is not suited for absorption coefficients close to 1. Finally, there is the (modified) model by Jing and Xiang (see Ref. 4)

$$h = cA_M = c\frac{\alpha}{2(2-\alpha)} \qquad \text{for} \qquad \alpha \leq 1$$

In this model, the boundary condition is theoretically grounded and can model high absorption for a small portion of surfaces. In addition, the diffusion-equation model inherently assumes that overall absorption in rooms under test must not be “high”.

Interior BCs (coupling between rooms/volumes):

On interior boundaries between rooms (thin walls, doors, grills, thin panels etc.) there can be a transmission loss (TL) associated. The transmission loss is related to the transmission coefficient τ of the boundary as

$$\mathrm{TL} = 10\log\left(\frac{1}{\tau}\right)$$

This results in a condition where the field w is discontinuous across the boundary (a slit). Using the usual up/down conventions the condition, including possible absorption. The TL for an opening is typically given for the boundary in which the opening is located. That is the combined transmission and absorption is smeared on a boundary.

With the surface normal pointing out of the domains, the flux on the down side of the surface/wall (including absorption) is

$$\mathbf{J} \cdot \mathbf{n}_{\text{down}} = \text{down}\left(hw + \frac{\tau c}{4} w\right) - \text{up}\left(\frac{\tau c}{4} w\right)$$

on the up side of the surface/wall (including absorption) the total flux is

$$\mathbf{J} \cdot \mathbf{n}_{\text{up}} = \text{up}\left(hw + \frac{\tau c}{4} w\right) - \text{down}\left(\frac{\tau c}{4} w\right)$$

The transmission coefficient is due to reciprocity the same from both sides. This results in the combined condition

$$\mathbf{J} \cdot \mathbf{n}_{\text{up}} - \mathbf{J} \cdot \mathbf{n}_{\text{down}} = [hw]_{\text{up}} - [hw]_{\text{down}} - \frac{\tau c}{4}(w_{\text{up}} - w_{\text{down}})$$

The "room coupling" condition adds the fluxes related to the TL. Note also that the terms of the type $\tau cw/4$ should probably be modified for small TL because τ then becomes larger than 0.2 (at around 7 dB). An option exists to force continuity by constraining

$$w_{\text{up}} = w_{\text{down}}$$

which basically corresponds to a null transmission loss.

Inflow and Outflow Boundary Conditions:
Some general boundary conditions defining a total inflow or outflow of energy are added as

$$\mathbf{J} \cdot \mathbf{n} = -D_{\text{t}} \frac{\partial w}{\partial n} = J_{\text{out}}$$

$$\mathbf{J} \cdot \mathbf{n} = -D_{\text{t}} \frac{\partial w}{\partial n} = -J_{\text{in}}$$

POSTPROCESSING VARIABLES/EXPRESSIONS

The square pressure (valid for spatially averaged quantities and volumes that are large compared to the wavelength but small compared to the mean free path):

$$p^2(\mathbf{x}, t) = \rho c^2 w(\mathbf{x}, t)$$

Solution of the energy equation yields (with an impulse source) the energy room impulse response (excluding the direct sound, as the sound field is assumed diffuse).

It is also known as the energy-time function (ETF) or energy-time curve (ETC). The SPL response is given by

$$L_{\mathrm{p}}(\mathbf{x}, t) = 10\log\left(\frac{\rho c^2 w(\mathbf{x}, t)}{P_{\mathrm{ref}}^2}\right)$$

this is in general the local SPL for any source. For a steady state model this is the spatial SPL distribution $L_p(\mathbf{x})$.

The steady-state sound-energy decay $d(\mathbf{x},t)$ can be calculated from an energy impulse response using so-called Schroeder-integration. This corresponds to first finding the steady state (using a stationary solver) and then use this solution as initial condition (source turned off) in a time dependent model. The same response can be calculated using Schroeder integration of an impulse response model

$$d(\mathbf{x}, t) = \frac{1}{N(\mathbf{x})}\int_t^\infty w(\mathbf{x}, \tau)d\tau \qquad N(\mathbf{x}) = \int_0^\infty w(\mathbf{x}, \tau)d\tau$$

This integration can be set up in COMSOL Multiphysics postprocessing using the built in `timeint()` operator.

A variable for the a posteriori assessment of whether the diffuse field assumption is met is given by (named "A posteriori diffuse field evaluation norm" in the postprocessing list)

$$L_{\mathrm{diffuse}} = \frac{|\mathbf{J}|}{cw} = \frac{D_{\mathrm{t}}|\nabla w|}{cw}$$

If this variable is small then the acoustic diffusion assumption is met. Another variable that can be assessed is the Schroeder frequency as given above.

DIFFERENT STUDY TYPES

Transient

A transient simulation is used to model the energy impulse response (the response to a source of the type $E_0\lambda(t)$, typically approximated by a step function of a shirt time) of a room or the steady-state energy decay response (decay from a steady state solution). In a single room the response will be the same as measured in all points (except for a small time lag). This will result in identical reverberation times (RT) estimates. The interest of the transient model is more evident when several rooms are

coupled. Here several time scales exist because of the interaction between the rooms and these can be seen in a transient energy impulse response.

Steady State

The steady state model is used to find the spatial SPL distribution inside a room or collection of rooms and volumes when subject to a steady noise source. This could be an engine noise or the noise generated by other machinery.

Eigenvalue

An eigenvalue analysis of the transient equation results in all the "basis functions" of the problem. That is, modes that do not change shape in space but only decay in time with the exponential time scale given by the eigenvalue. They are of the type

$$w(x, t) = w_i(x)e^{-\lambda_i t}$$

where λ_i is the i'th eigenvalue and w_i the associated mode. The eigenvalues are a direct measures of the reverberation time (RT) of the different rooms. T_{60} is for example simply given by

$$T_{60} = \frac{60}{10 \cdot \log(e) \cdot \lambda_i} = \frac{55.3}{4\lambda_i}$$

where the last equation is the classical expression. Evaluating the eigenvalues will gives an easy measures of the slopes of the energy-time functions (ETFs) and can be combined with a transient model to give the full picture of the reverberation in the different volumes/rooms and their coupled behavior. When performing an eigenvalue study the variable `ade.T60` gives the reverberation time associated with the given eigenvalue. Inspect the sound energy density modes plot to determine see to which room the reverberation time is associated.

References for the Acoustic Diffusion Equation Interface

1. J. Picaut, L. Simon, and J. D. Polack, "Sound field in long rooms with diffusely reflecting boundaries," *Appl. Acoust*, vol. 56, pp. 217–240, 1999.

2. J. Pujolle, "Les différentes définitions du libre parcours moyen du son dans une salle," *Revue d'Acoustique*, vol. 36, pp. 44–50, 1976.

3. N. Xiang, Y. Jing, and A. C. Bockman, "Investigation of acoustically coupled enclosures using a diffusion-equation model," *J. Acoust. Soc. Am.,* vol. 126, 2009.

4. Y. Jing and N. Xiang, "On boundary conditions for the diffusion equation in room-acoustic prediction," *J. Acoust. Soc. Am.* vol. 123, pp. 145–153, 2008.

5. A. Billon and others, "Modeling the sound transmission between rooms coupled through partition walls by using a diffusion model," *J. Acoust. Soc. Am.,* vol. 123, 2008.

6. V. Valeau and others, "On the use of a diffusion equation for room acoustic prediction," *J. Acoust. Soc. Am.*, vol. 119, 2006.

7. Y. Jing and N. Xiang, "On The Use of a Diffusion Equation Model for Sound Energy Flow Prediction in Acoustically Coupled Spaces," COMSOL Conference Boston 2008.
www.comsol.com/paper/on-the-use-of-a-diffusion-equation-model-for-sound-energy-flow-prediction-in-aco-5124

8. Y. Jing and N. Xiang, "Boundary Conditions for the Diffusion Equation Model in Room-acoustic Prediction," COMSOL Conference Boston 2007.

9. T. J. Cox and P. D'Antonio, *Acoustic, Absorbers and Diffusers*, Taylor and Francis, 2009.

10. R. Schroeder, "New method of measuring reverberation time," *J. Acoust. Soc. Am.,* vol. 37, pp. 409–412, 1965.

11. C. Foy, V. Valeau, A. Billon, J. Picaut, and M. Hodgson, "An Empirical Diffusion Model for Acoustic Prediction in Rooms with Mixed Diffuse and Specular Reflections," *Acta Acustica united with Acustica*, vol. 95, pp. 97–105, 2009.

12. J. Escolano, J. M. Navarro, and J. L. Lopes, "On the limitation of a diffusion equation model for acoustic predictions of rooms with homogeneous dimensions (L)," *J. Acoust. Soc. Am.*, vol. 128, pp. 1586, 2010.

13. C. Visentin, N. Prodi, V. Valeau, and J. Picaut, "A numerical investigation of the Fick's law of diffusion in room acoustics," *J. Acoust. Soc. Am.*, vol. 132, pp. 3180, 2012.

14. C. Visentin, *A numerical and experimental validation of the diffusion equation applied to room acoustics*, Ph.D. thesis, 2012.

15. V. Valeau, M. Hodgson, and J. Picaut, "A Diffusion-Based Analogy for the Prediction of Sound Fields in Fitted Rooms," *Acta Acustica united with Acustica*, vol. 93, pp. 94–105, 2007.

16. A. D. Pierce, *Acoustics, An Introduction to Its Physical Principles and Applications*, Chapter SIX, Acoustical Society of America, 1991.

17. L. Beranek and N. Nishihara, "Mean-free-paths in concert and chamber music halls and the correct method for calibrating dodecahedral sound sources," *J. Acoust. Soc. Am.*, vol. 135, pp. 223, 2014.

18. H. Kuttruff, *Room Acoustics*, Fifth Edition, Spon Press, 2009.

8

Multiphysics Couplings

This chapter contains a description of all the Multiphysics coupling features of the Acoustics Module. Several of the coupling features are used in predefined multiphysics interfaces but can as well be added manually when modeling multiphysics systems.

Coupling Features

The Acoustics Module contains a number of Multiphysics Coupling features that can couple acoustic domains to structures and porous materials, thermoviscous acoustics to pressure acoustics, or simply a flow to an aeroacoustic interface. The couplings features are described below.

The couplings features that exist are:

- Acoustic-Structure Boundary
- Thermoviscous Acoustic-Structure Boundary
- Aeroacoustic-Structure Boundary
- Acoustic-Thermoviscous Acoustic Boundary
- Acoustic-Porous Boundary
- Porous-Structure Boundary
- Background Potential Flow

Acoustic-Structure Boundary

The **Acoustic-Structure Boundary** coupling ( ) is used to couple a Pressure Acoustics model to any structural component. The feature couples to Solid Mechanics, Poroelastic Waves, Shell, Membrane, and Multibody Dynamics interfaces.

The coupling includes the fluid load on the structure and the structural acceleration as experienced by the fluid. For thin interior structures like shells or membranes with fluid on both sides, a slit is added to the pressure variable and care is taken to couple the up and down sides.

Mathematically the condition on exterior boundaries reads

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - q_d)\right) = -\mathbf{n}\cdot\mathbf{u}_{tt}$$

$$\mathbf{F}_A = p_t\mathbf{n}$$

where $\mathbf{u}_{tt}$ is the structural acceleration, $\mathbf{n}$ is the surface normal, p_t is the total acoustic pressure and $\mathbf{F}_A$ is the load (force per unit area) experienced by the structure. On interior boundaries the condition reads

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - q_d)\right)_{up} = \mathbf{n}\cdot\mathbf{u}_{tt}$$

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - q_d)\right)_{down} = -\mathbf{n}\cdot\mathbf{u}_{tt}$$

$$\mathbf{F}_A = (p_{t,down} - p_{t,up})\mathbf{n}$$

such that the acoustic load is given by the pressure drop across the thin structure. The up and down subscripts refer to the two sides of the interior boundary.

In 2D components it is possible to define the out of plane thickness for the structural interfaces while in acoustics this thickness effectively unity (set to 1 m). A rescaling is automatically performed for the pressure load to account for this difference in interpretation in 2D.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `asb1`.

ACOUSTIC-STRUCTURE BOUNDARY

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces.

The default values depend on how this node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when a multiphysics interface is selected in the **Model Wizard** or **Add Physics** window, then the two participating interfaces are selected.

You can also select **None** from either list to uncouple this node from a physics interface. If the physics interface is removed from the **Model Builder**, for example **Pressure**

Acoustics, Frequency Domain is deleted, then the **Acoustics** list defaults to **None** as there is nothing to couple to.

If a physics interface is deleted and then added to the model again, then in order to re-establish the coupling, you need to choose the physics interface again from the **Acoustics** or **Structure** lists. See Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

Thermoviscous Acoustic-Structure Boundary

The **Thermoviscous Acoustic-Structure Boundary** coupling ( ) is used to couple a Thermoviscous Acoustics model to any structural component. The feature couples to Solid Mechanics, Shell, Membrane, and Multibody Dynamics interfaces.

The coupling prescribes continuity in the displacement field

$$\mathbf{u}_{\mathrm{t,fluid}} = i\omega\mathbf{u}_{\mathrm{solid}} \qquad \text{or} \qquad \mathbf{u}_{\mathrm{t,fluid}} = \frac{\partial \mathbf{u}_{\mathrm{solid}}}{\partial t}$$

where $\mathbf{u}_{\mathrm{t,fluid}}$ is the total fluid velocity (including a background component if applicable) and $\mathbf{u}_{\mathrm{solid}}$ is the solid displacement. The first equation is in the frequency domain and the second in the time domain, respectively. This coupling results in the stress also being continuous across the boundary. The condition for the total temperature T_{t} can be set to either isothermal or adiabatic. In the case where a shell or membrane is interior to the thermoviscous acoustics domain, a slit is automatically applied to the pressure p and temperature T degrees of freedom (DOFs).

The coupling has a special formulation for eigenfrequency studies, which allows the program to find the eigenvalue for coupled systems.

For an example using this multiphysics coupling see the *Vibrating Micromirror with Viscous and Thermal Damping* model: Application Library path **Acoustics_Module/Vibrations_and_FSI/vibrating_micromirror**

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `tsb1`.

THERMAL

Select the **Thermal condition** that applies on the solid surface to be either **Isothermal** (the default selected) or **Adiabatic**.

CONSTRAINT SETTINGS

These settings are shown when the **Advanced Physics Options** is selected. It controls how the constraints (continuity in displacement) are handled. Select either **Study controlled** (the default selected), **Weak constraints**, or **Pointwise constraints**.

When set to **Study controlled**, the weak constraints are automatically selected when an eigenfrequencies study is performed. This type of coupling is necessary as the eigenvalue (the angular frequency ω) enters the coupling expression. This will create extra variables at the boundary, so-called Lagrange multipliers, ensuring the correct behavior and solution. For a normal frequency domain study the pointwise constraint is selected. In the time domain there are no options and the weak formulation is always used.

THERMOVISCOUS ACOUSTIC-STRUCTURE BOUNDARY

This section defines the physics involved in the multiphysics coupling. The **Thermoviscous Acoustics** and **Structure** lists include all applicable physics interfaces. See Acoustic-Structure Boundary for details.

Aeroacoustic-Structure Boundary

The **Aeroacoustic-Structure Boundary** coupling ( ) is used to couple an Aeroacoustic model (it only applies to the Linearized Navier-Stokes interfaces) to any structural component. The feature couples to Solid Mechanics, Shell, Membrane, and Multibody Dynamics interfaces. The coupling can be used to, for example, model fluid structure interaction (FSI) in the frequency domain.

The coupling prescribes continuity in the displacement field

$$\mathbf{u}_{\mathrm{t,fluid}} = i\omega\mathbf{u}_{\mathrm{solid}} \qquad \text{or} \qquad \mathbf{u}_{\mathrm{t,fluid}} = \frac{\partial \mathbf{u}_{\mathrm{solid}}}{\partial t}$$

where $\mathbf{u}_{\mathrm{t,fluid}}$ is the total fluid velocity (including a background component if applicable) and $\mathbf{u}_{\mathrm{solid}}$ is the solid displacement. The first equation is in the frequency domain and the second in the time domain, respectively. This coupling results in the stress also being continuous across the boundary. The condition for the total temperature T_{t} can be set to either isothermal or adiabatic. In the case where a shell or

membrane is interior to the thermoviscous acoustics domain, a slit is automatically applied to the pressure p and temperature T degrees of freedom (DOFs).

The coupling has a special formulation for eigenfrequency studies which allows the user to find the eigenvalue for coupled systems.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `aesb1`.

THERMAL

Select the **Thermal condition** that applies on the solid surface to be either **Isothermal** (the default selected) or **Adiabatic**.

CONSTRAINT SETTINGS

See Constraint Settings under the Thermoviscous Acoustic-Structure Boundary.

AEROACOUSTIC-STRUCTURE BOUNDARY

This section defines the physics involved in the multiphysics coupling. The **Aeroacoustics** and **Structure** lists include all applicable physics interfaces. See Acoustic-Structure Boundary for further details.

Acoustic-Thermoviscous Acoustic Boundary

The **Acoustic-Thermoviscous Acoustic Boundary** coupling ( ) is used to couple a thermoviscous acoustics domain to a pressure acoustics domain (in both frequency and time domain). As it is only necessary to solve the full detailed thermally conducting and viscous model near walls in the boundary layer region, it makes sense to switch to classical pressure acoustics outside this region. This saves a lot of memory and solution time due to the reduced number of degrees of freedom.

Note that, when using the coupling at places where pressure acoustics and thermoviscous acoustics interact at a common wall, the coupling becomes unphysical if no slip and isothermal conditions are used in the thermoviscous domain. The reason is that the conditions at the wall must match between the physics. The introduced error depends on the model size and the thickness of the acoustic boundary layer. If it makes physical sense then use the slip and adiabatic options on that specific wall (add an extra wall boundary condition).

The coupling prescribes continuity in the total normal stress and the total normal acceleration for the mechanical part. An adiabatic condition is prescribed for the total temperature:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t^{pa}-\mathbf{q})\right) = -\mathbf{n}\cdot i\omega\mathbf{u}_t \qquad \text{(frequncy domain)}$$

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t^{pa}-\mathbf{q})\right) = -\mathbf{n}\cdot\frac{\partial\mathbf{u}_t}{\partial t} \qquad \text{(time domain)}$$

$$\left[-p_t\mathbf{I}+\mu(\nabla\mathbf{u}_t+(\nabla\mathbf{u}_t)^T)-\left(\frac{2\mu}{3}-\mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -p_t^{pa}\mathbf{n}$$

$$-\mathbf{n}\cdot(-k\nabla T_t) = 0$$

The thermoviscous acoustic pressure variable is p_t and the pressure acoustic pressure is here denoted p_t^{pa} for clarity.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `atb1`.

ACOUSTIC-THERMOVISCOUS ACOUSTIC BOUNDARY

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Thermoviscous Acoustics** lists include all applicable physics interfaces. See Acoustic-Structure Boundary for further details.

Acoustic-Porous Boundary

The **Acoustic-Porous Boundary** coupling ( ) is used to couple a Pressure Acoustics, Frequency Domain interface to a Porous Material (Biot's model) domain from the Poroelastic Waves interface. The boundary coupling feature includes the following interaction between the fluid and the porous domains:

- Continuity of the fluid pressure on the boundaries where the fluid interacts with the porous domain. The pore pressure in the porous domain is set equal to the total pressure in the fluid domain

$$p_{pore} = p_t$$

- A pressure load from the fluid pressure is experienced by the elastic waves in the porous material

$$\mathbf{n} \cdot (\boldsymbol{\sigma}_d - \alpha_B p \mathbf{I}) = -\mathbf{n} p_t$$

where p_t is the total acoustic pressure in the fluid domain and the left hand side represents the total stress for the saturated porous domain.

- The pressure acoustic domain also experiences a normal acceleration that depends both on the acceleration of the porous matrix skeleton but also on the pore pressure. Because of the pressure boundary condition, which is a bidirectional constraint, this condition reduces to the fluid experiencing a normal acceleration

$$\mathbf{a}_n = (i\omega)^2 \mathbf{u}$$

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `apb1`.

ACOUSTIC-POROUS BOUNDARY

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Porous media** lists include all applicable physics interfaces. See Acoustic-Structure Boundary for further details.

Porous-Structure Boundary

The **Porous-Structure Boundary** coupling () is used to couple a Porous Domain (Biot's model) domain from the Poroelastic Waves interface to a Solid Mechanics, Shell, Membrane, or Multibody Dynamics interface. At the solid-porous boundaries continuity of the displacement field is applied.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `psb1`.

POROUS-STRUCTURE BOUNDARY

This section defines the physics involved in the multiphysics coupling. The **Porous media** and **Structure** lists include all applicable physics interfaces. See Acoustic-Structure Boundary for further details.

Background Potential Flow

The **Background Potential Flow** coupling ( ) is a one-way coupling that couples the background flow calculated by a Compressible Potential flow to the Linearized Potential Flow interfaces. The coupling provides the inputs for the Linearized Potential Flow Model.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `pfc1`.

BACKGROUND FLOW COUPLING

This section defines the physics involved in the multiphysics coupling. The **Source** and **Destinations** lists include all applicable physics interfaces. See Acoustic-Structure Boundary for further details.

Predefined Multiphysics Interfaces

The Acoustics Module includes several predefined multiphysics interfaces. When these interfaces are selected from the **Model Wizard** or the **Add Physics** window both contributing physics and the multiphysics couplings are added to the **Model Builder**. It is always possible to build a model sequentially adding one physics at the time and then coupling them. Note that far from all possible couplings are represented in the predefined physics interfaces. All the couplings are listed in the Coupling Features section.

When physics interfaces are added using the predefined multiphysics interfaces, for example **Thermoviscous Acoustic-Structure Interaction**, the **Selection** of the coupling feature, in this case the **Thermoviscous Acoustic-Structure Boundary**, is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Thermoviscous Acoustics** on one side and **Solid Mechanics** on the other.

If physics interfaces are added one at a time, followed by the coupling features, the selections are not automatically set. For example, if the single interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available coupling features, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

The Acoustics Module includes the following predefined multiphysics interfaces:

- The Acoustic-Solid Interaction, Frequency Domain Interface
- The Acoustic-Solid Interaction, Transient Interface
- The Acoustic-Piezoelectric Interaction, Frequency Domain Interface
- The Acoustic-Piezoelectric Interaction, Transient Interface
- The Acoustic-Solid-Poroelastic Waves Interaction Interface
- The Acoustic-Poroelastic Waves Interaction Interface
- The Acoustic-Shell Interaction, Frequency Domain Interface
- The Acoustic-Shell Interaction, Transient Interface
- The Acoustic-Thermoviscous Acoustic Interaction, Frequency Domain Interface
- The Thermoviscous Acoustic-Solid Interaction, Frequency Domain Interface
- The Thermoviscous Acoustic-Shell Interaction, Frequency Domain Interface

Another option is to create the coupled predefined multiphysics interface by adding it from the **Add Multiphysics** window after adding the separate physics interfaces.

Modeling with Multiphysics Couplings

Using the predefined multiphysics couplings has many advantages. It improves the flexibility and enables sequential modeling, increasing the complexity of a model step by step. Care is also taken to make the correct couplings both on the physics level but also from a numerics point of view. It is of course possible to couple physics manually and this is maybe also necessary in some cases when more advanced and exotic couplings are necessary.

The predefined multiphysics coupling approach used improves the flexibility and design options for your modeling. For for details, see The Multiphysics Node and Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual.*

In this section a few tips and tricks for modeling acoustics and vibration problems using the multiphysics couplings are presented.

- Use Selections
- The Override Behavior
- The Solvers
- Perfectly Matched Layers (PMLs)

Use Selections

When modeling it is always recommended to use selections to predefine and give names to several domains or boundaries. Define the selections under the **Definitions** node. Setting up these selections will for multiphysics modeling avoid adding two incompatible physics to the same domain. If acoustics and structures are coupled this happens at boundaries, where two distinct physics should meet. For, for example, piezoelectric devices the coupling between the electric and mechanical effect happen on the domain level. Operations can also be performed on the selections if the geometry is complex to find intersections, complement sets and so forth.

See Named Selections in the *COMSOL Multiphysics Reference Manual.*

The Override Behavior

When coupling acoustics, both Pressure Acoustics, Thermoviscous Acoustics, and Aeroacoustics, to a structure the behavior of the multiphysics coupling is such that it will override all boundary condition given on the acoustic side. Loads and constraints defined on the common boundaries in the structural physics interface will contribute with the coupling. In this way external forces that drive the system can be added here. It is also possible to constrain the structure on these boundaries while it will still experience the acoustic loads.

The Solvers

When solving a Multiphysics problem it is necessary to ensure that the solvers behave correctly. It is especially important to check if a model is solved **Fully Coupled** or in a **Segregated** way. COMSOL has built in logic to handle many multiphysics problems when the predefined multiphysics couplings are used ensuring that the model solves correctly. If many physics or couplings are added at the same time this logic can fail and manual settings should be performed. For large models COMSOL generates Iterative Solver Suggestions (see below).

See Introduction to Solvers and Studies and Solution Operation Nodes and Solvers in the *COMSOL Multiphysics Reference Manual*.

It is also possible to enable and disable both physics and couplings from within a Study Step. This can be done in the **Physics and Variables Selection** section in **Settings**. This functionality can be used to solve a model both coupled and uncoupled by adding two **Studies** : one where the Multiphysics coupling is active and one where it has been deactivated.

ITERATIVE SOLVER SUGGESTIONS

Multiphysics models can in 3D easily grow rapidly in size and become difficult to solve with a direct solver, simply because you run out of RAM. In this case it can be necessary to switch to an iterative solver approach. For certain multiphysics couplings iterative solver suggestions are automatically generated. Set up a new solver configuration (first time the model is solved) or right click the study node and select **Show Default Solver**, then expand the **Solver Configuration** tree under **Stationary Solver** or **Time-Dependent Solver**. The predefined iterative solver suggestions that are automatically generated can be seen here. In the predefined cases a direct solver is used as default and iterative

solvers are suggested and disabled (grayed out). To turn on one of these approached right-click the solver and select **Enable** (or press F4). Predefined suggestions exist for:

- **Acoustic-Structure Boundary**, gives one iterative solver suggestion.
- **Thermoviscous Acoustic-Structure Boundary**, generates two suggestions. The first is fast and can save 20% of memory while the second that uses domain decomposition (DD) is very robust and memory lean, but it can be slow.
- Combining **Acoustic-Structure Boundary** and **Piezoelectric Effect** yields a suggestion that can be used for modeling large piezoelectric transducer assemblies.

All solver suggestions have a tag that makes it possible to see which physics feature that has generated them, for example, *(asb1)* means that the suggestion comes from the *Acoustic-Structure Boundary 1* feature.

Perfectly Matched Layers (PMLs)

Care should be taken when setting up perfectly matched layers (PMLs) in multiphysics models where different waves propagate. A discussion about the Acoustic-Structure interaction situation is given in Configuration of Perfectly Matched Layers (PMLs) for Acoustic-Structure Interaction Models in Modeling with the Acoustic-Structure Interaction Branch. The approach described is valid in general for models where several sound speeds are present.

9

Structural Mechanics with the Acoustics Module

Elastic waves and vibrations are an important part of acoustics. The Acoustics Module is therefore equipped with an extended version of the base package Solid Mechanics interface shipped with COMSOL Multiphysics. The interface contain less features when compared to the full Solid Mechanics interface available with the Structural Mechanics Module. This chapter describes the features and applications of the Solid Mechanics and Piezoelectric Devices interface available with the Acoustics Module. The physics interfaces is found under the **Structural Mechanics** branch ().

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

The full documentation of all features in the full Solid Mechanics interface and the Piezoelectric Devices interface is found in the *Structural Mechanics Module User's Guide*, specifically in these sections:

- Solid Mechanics
- Structural Mechanics Modeling

- Structural Mechanics Theory
- Multiphysics Interfaces and Couplings and specifically The Piezoelectric Devices Interface

In this chapter:

- Vibroacoustic Applications
- The Solid Mechanics Interface
- The Piezoelectric Devices Interface
- Acoustic-Structure Multiphysics Interaction

Vibroacoustic Applications

The Solid Mechanics and Piezoelectric Devices interface are available with the Acoustics Module such that vibroacoustic problems involving solids, piezoelectric materials, and acoustic domains can be modeled. These multiphysics applications include piezo transducers, loudspeaker drivers, loudspeaker cabinets, automotive applications, and sound transmission/isolation problems. Pure elastic wave problems can also be modeled as the Solid Mechanics equations are formulated include the full dynamic behavior.

Additional multiphysics capabilities of the Acoustics Module are enabled when it is combined with the Structural Mechanics Module. The Shell interface, which is part of the Structural Mechanics Module, enables predefined acoustic-shell couplings. The Acoustic-Shell and Thermoacoustic-Shell interaction multiphysics interfaces are enabled. The option to couple to shells, membranes and multibody dynamics using the Acoustic-Structure Boundary is also enabled, see the Multiphysics Couplings chapter for further details.

The Solid Mechanics Interface

Included in the Acoustics Module is an extended version of the Solid Mechanics interface shipped with COMSOL Multiphysics. The interface contain less features when compared to the full Solid Mechanics interface available with the Structural Mechanics Module.

The Piezoelectric Devices Interface

The Piezoelectric Devices interface is also included in the Acoustics Module. This is a Multiphysics interface that couples the Electrostatics interface with the Solid Mechanics interface. See Modeling Piezoelectric Problems section in the *Structural Mechanics Module User's Guide* or further details.

Acoustic-Structure Multiphysics Interaction

As soon as an acoustics interface and a structural mechanics interface are present in the same model the **Multiphysics** node appear, if there exist a predefined coupling between the two interfaces. Under the **Multiphysics** node you can add the predefined multiphysics couplings that couple the fluid domain and to the solid domain. The couplings are described in detail in the Multiphysics Couplings chapter.

You can also find a set of predefined multiphysics interface that will automatically sets up the involved single physics and the multiphysics coupling. These are described in the Acoustic-Structure Interaction Interfaces chapter. Here you will also find modeling tips for acoustic-structure interaction problems in the Modeling with the Acoustic-Structure Interaction Branch section.

10

Glossary

This Glossary of Terms contains finite element modeling terms in an acoustics context. For mathematical terms as well as geometry and CAD terms specific to the COMSOL Multiphysics® software and documentation, see the glossary in the *COMSOL Multiphysics Reference Manual*. For references to more information about a term, see the index.

Glossary of Terms

acoustic impedance At a specified surface, the complex quotient of acoustic pressure by normal fluid velocity. SI unit: Pa/(m/s).

acoustic reactance The imaginary part of the acoustic impedance.

acoustic resistance The real part of the acoustic impedance.

acoustic-structure interaction 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.

adiabatic bulk modulus One over the compressibility β_s measured at constant entropy. The adiabatic bulk modulus is denoted K_s and gives a measure of the compressibility of the fluid and is directly related to the speed of sound c_s in the fluid. SI unit: Pa.

$$K_s = \frac{1}{\beta_s} = \rho_0 c_s^2$$

admittance The reciprocal of impedance.

aeroacoustics The scientific field of study used to couple acoustics and fluid dynamics.

anisotropy Variation of material properties with direction.

arbitrary Lagrangian-Eulerian (ALE) method A technique to formulate equations in a mixed kinematical description. An ALE referential coordinate system is typically a mix between the material (Lagrangian) and spatial (Eulerian) coordinate systems.

Bernoulli equation An integrated form of Euler's momentum equation along a line of flow. The equation gives an expression for an invariant quantity in an inviscid fluid. A decrease in the speed of the fluid translates to an increase in the fluid pressure and/or potential energy.

bulk modulus One over the compressibility. It gives a measure of the compressibility of the fluid and is related to the speed of sound in the fluid. SI unit: Pa. See also *adiabatic bulk modulus*.

characteristic impedance The product of the equilibrium density and the speed of sound in a medium. SI unit: Pa/(m/s).

compliance Reciprocal of *stiffness*.

compliance matrix The inverse of the elasticity matrix. See *elasticity matrix*.

Cauchy stress The most fundamental stress measure defined as force/deformed area in fixed directions not following the body.

constitutive equations Equations that relate two physical quantities. In thermoviscous acoustics both the stress tensor (relating velocity to stress) and Fourier's law of heat conduction (relating heat conduction to temperature) are constitutive relations. In structural mechanics this is the equation formulating the stress-strain relationship of a material. Constitutive equations are supplemented by equilibrium equations (mass, momentum, and energy) and an *equation of state* to provide a full physical description.

creep Time-dependent material nonlinearity that usually occurs in metals at high temperatures in which the effect of the variation of stress and strain with time is of interest.

damping Dissipation of energy in the fluid or a vibrating structure. The damping is typically due to viscous losses or thermal conduction. In acoustics this happens in structures with small geometrical dimensions, for example, small pipes or porous materials. In structures a common assumption is viscous damping where the damping is proportional to the velocity. See also *Rayleigh damping*.

decibel (dB) Logarithmic unit that indicates the ratio of a physical quantity relative to a reference value.

dipole source An acoustic source that behaves as a translational oscillating sphere.

Doppler effect Change in the observed frequency of a wave caused by a time rate of change in the effective length of the path of travel between the source and the observation point.

effective sound pressure RMS instantaneous sound pressure at a point during a time interval, T, long enough that the measured value is effectively independent of small changes in T. SI unit: Pa = N/m^2.

equation of state The thermodynamic relation between three independent thermodynamic variables. Typically in acoustics it is the density $\rho = \rho(p,s)$ given as function of the entropy s and the pressure p.

eigenmode A possible propagating mode of an acoustic wave.

elasticity matrix The matrix D relating strain to stresses:

$$\sigma = D\varepsilon$$

Eulerian Model described and solved in a coordinate system that is fixed (spatial). See also *Lagrangian* and *arbitrary Lagrangian-Eulerian method*.

Green-Lagrange strain Nonlinear strain measure used in large-deformation analysis. In a small strain, large rotation analysis, the *Green-Lagrange strain* corresponds to the engineering strain, with the strain values interpreted in the original directions. The *Green-Lagrange* strain is a natural choice when formulating a problem in the undeformed state. The conjugate stress is the *second Piola-Kirchhoff stress*.

impedance At a specified frequency, the quotient of a dynamic field quantity (such as force, sound, pressure) by a a kinematic field quantity (such as vibration velocity, particle velocity).

instantaneous sound pressure Total instantaneous pressure at a point in a medium minus the static pressure at the same point. SI unit: Pa = N/m^2.

irrotational background velocity field A velocity field $\mathbf{u}$ that has the property of having rotation $\nabla \times \mathbf{u} = \mathbf{0}$ everywhere, where the first term is the vorticity of the fluid. In such a fluid the viscous stress does not contribute to the acceleration of the fluid. The mean pressure in this fluid is described by *Bernoulli's equation*.

Lagrangian Model described and solved in a coordinate system that moves with the material. See also *Eulerian* and *arbitrary Lagrangian-Eulerian method*.

monopole source An acoustic source that behaves as a radially oscillating sphere.

particle velocity In a sound field, the velocity caused by a sound wave of a given infinitesimal part of the medium relative to the medium as a whole.

PML (perfectly matched layer) Domain adjoined at a system boundary designed to emulate a non-reflecting boundary condition independently of the shape and frequency of the incident wave front.

principle of virtual work States that the variation in internal strain energy is equal to the work done by external forces.

propagating acoustic modes The acoustic modes or wave shapes that propagate with no significant damping for a given frequency in a duct of a given cross-section.

Rayleigh damping A viscous damping model where the damping is proportional to the mass and stiffness through the mass and stiffness damping parameters.

reference sound pressure See definition in the entry for *sound pressure level*.

resonance frequency A frequency at which the system has the tendency to oscillate at a greater amplitude than at non-resonance frequencies. At the resonance frequencies the system can easily transfer energy from the actuation to the vibrating structure or acoustic wave.

RMS value Root-mean-square value; for the (complex) sound pressure, $p(t)$, over the time interval $T_1 < t < T_2$ defined as

$$p_{\mathrm{RMS}} = \sqrt{\frac{1}{T_2 - T_1}\int_{T_1}^{T_2} \mathrm{Re}[p(t)]^2 dt}$$

For a harmonic pressure wave, $p(t) = p_0 e^{i\omega t}$, the time interval is taken to be a complete period, resulting in $p_{\mathrm{RMS}} = p_0/\sqrt{2}$.

second Piola-Kirchhoff stress Conjugate stress to *Green-Lagrange strain* used in large deformation analysis.

sound energy Total energy in a given part of a medium minus the energy that would exist at the same part in the absence of sound waves. SI unit: J.

sound-energy flux density See *sound intensity*.

sound intensity Average rate of *sound energy* transmitted in a specified direction at a point through a unit area normal to this direction. SI unit: $\mathrm{W/m^2}$.

sound pressure See *effective sound pressure*.

sound pressure amplitude Absolute *instantaneous sound pressure* in any given cycle of a sound wave at some specified time. SI unit: W/m^2.

sound power density See *sound intensity*.

sound pressure level Ten times the logarithm (to the base ten) of the ratio of the time-mean-square pressure of a sound, in a stated frequency band, to the square of a *reference sound pressure*, p_{ref}. For gases, $p_{ref} = 20\ \mu Pa$, for other media (unless otherwise specified) $p_{ref} = 1\ \mu Pa$. Unit: dB (*decibel*).

sound source strength Maximum instantaneous rate of volume displacement produced by a source when emitting a harmonic sound wave. SI unit: m^3/s.

specific acoustic impedance At a point in a sound field, the quotient of sound pressure by particle velocity. SI unit: Pa/(m/s).

speed of sound The rate of change of particle displacement with distance for a sound wave. SI unit: m/s.

spin tensor The skew-symmetric part of the velocity gradient tensor.

static pressure Pressure that would exist at a point in the absence of a sound wave.

stiffness Ratio of change of force (or torque) to the corresponding change in translational (or rotational) displacement of an elastic element.

thermoviscous acoustics The interaction between thermodynamic and acoustic phenomena, which takes into account the temperature oscillations that accompany the acoustic pressure oscillations. The combination of these oscillations produces *thermoviscous acoustic* effects. Thermoviscous acoustic phenomena are modeled by solving the full linearized Navier-Stokes equation (momentum equation), the continuity equation, and the energy equation. Thermoviscous acoustics is also known as viscothermal acoustics or thermoacoustics.

velocity potential When a flow is irrotational $\nabla \times \mathbf{u} = \mathbf{0}$ the vector field (velocity field) can always be derived from a scalar potential $\phi(\mathbf{x})$ as $\mathbf{u} = \nabla\phi$, where ϕ is the velocity potential. See also *irrotational background velocity field*.

waveguide structures Structures that have the property of guiding sound waves. See also *propagating acoustic modes*.

Index