COMSOL MULTIPHYSICS

ACOUSTICS MODULE

USER'S GUIDE

VERSION 3.4

COMSOL

How to contact COMSOL:

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

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

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

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

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

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

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

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

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

United Kingdom
COMSOL Ltd.
UH Innovation Centre
College Lane
Hatfield
Hertfordshire AL10 9AB
Phone:+44-(0)-1707 284747
Fax: +44-(0)-1707 284746
info.uk@comsol.com
www.uk.comsol.com

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

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

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

info@comsol.com
www.comsol.com

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

Company home page
www.comsol.com

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

Acoustics Module User's Guide

Version: October 2007 COMSOL 3.4

CONTENTS

Chapter 1: Introduction

Chapter 2: Acoustics Module Overview

Chapter 3: Modeling Acoustics

Chapter 4: Application Mode Overview

Chapter 5: Pressure Acoustics

Chapter 6: Aeroacoustics

Chapter 7: Structural Mechanics Application Modes

Chapter 8: Piezoelectric Application Modes

Chapter 9: Glossary

1

Introduction

The Acoustics Module is an optional package that extends the COMSOL Multiphysics modeling environment with customized user interfaces and functionality optimized for the analysis of acoustics. Like all modules in the COMSOL family, it provides a library of prewritten ready-to-run models that make it quicker and easier to analyze discipline-specific problems.

This particular module solves problems in the general areas of acoustics, acoustic-structure interaction, and vibration. The application modes included here are fully multiphysics enabled, making it possible to couple them to any other physics application mode in COMSOL Multiphysics or the other modules. Explicit demonstrations of these capabilities are supplied with the product, a prominent example being a model of a loudspeaker involving both electromechanical and acoustic-structural couplings.

The documentation set for the Acoustics Module consists of two books. The one in your hands, the *Acoustics Module User's Guide*, introduces you to the basic functionality in the module, reviews basic modeling techniques, and includes reference material of interest to those working in acoustics. The second book in the set, the *Acoustics Module Model Library*, contains several ready-to-run models that illustrate real-world applications of the module. Each model comes with an introduction covering basic theory and the modeling purpose as well as

step-by-step instructions that illustrate how to set it up. Further, we supply these models as COMSOL Multiphysics Model MPH-files so you can import them into COMSOL Multiphysics for immediate execution. This way you can follow along with the printed discussion as well as use them as a jumping-off point for your own modeling needs.

We hope you find these models useful. If you have any feedback on the models in this set, please let us know. Likewise, we welcome your suggestions for additional models that we could add to the library. Finally, if in your work you have developed a model you think would be a good candidate for inclusion in this model set, please let us hear about it. In any case, feel free to contact us at info@comsol.com.

New Features in the Acoustics Module 3.4

This new release of the Acoustics Module includes a number of valuable new capabilities, including the following features:

- New application modes for piezoelectric modeling.
- Ultraweak variational formulation (UWVF) for efficient simulations of pressure acoustics.
- Piezoelectric Material Properties database with material properties for 23 common piezoelectric materials.
- Improved Model Library including new models of ultrasound scattering, using the ultraweak variational formulation, a SAW (surface acoustic wave) gas sensor, and a piezoacoustic transducer.

Typographical Conventions

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

- A **boldface** font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL graphical user interface (for toolbar buttons in the corresponding tooltip). For instance, we often refer to the **Model Navigator**, which is the window that appears when you start a new modeling session in COMSOL; the corresponding window on the screen has the title **Model Navigator**. As another example, the instructions might say to click the **Multiphysics** button, and

the boldface font indicates that you can expect to see a button with that exact label on the COMSOL user interface.

- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user interface during geometry modeling. However, nowhere on the screen will you see the term "Draw" referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).
- The symbol **>** indicates a menu item or an item in a folder in the **Model Navigator**. For example, **Physics>Equation System>Subdomain Settings** is equivalent to: On the **Physics** menu, point to **Equation System** and then click **Subdomain Settings**. **COMSOL Multiphysics>Heat Transfer>Conduction** means: Open the **COMSOL Multiphysics** folder, open the **Heat Transfer** folder, and select **Conduction**.
- A `Code` (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as "Type `1.25` in the **Current density** edit field." The monospace font also indicates COMSOL Script codes.
- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

2

Acoustics Module Overview

This manual describes the Acoustics Module, an optional add-on package for COMSOL Multiphysics designed to assist you in solving and modeling acoustical problems. Here you find an introduction to the modeling stages of the Acoustics Module, including some illustrative models as well as information that serves as a reference source for more advanced modeling.

Capabilities

What Can the Acoustics Module Do?

The Acoustics Module is a collection of application modes for COMSOL Multiphysics adapted to a broad category of acoustics simulations in fluids and solids. Those who are not familiar with computational techniques but have a solid background in acoustics should find this module extremely beneficial. It can serve equally well as an excellent tool for educational purposes.

Because the Acoustics Module is smoothly integrated with all of the COMSOL Multiphysics functionality, you can couple a simulation in this module to an arbitrary simulation defined in any of the COMSOL Multiphysics application modes. This forms a powerful *multiphysics* model that solves all the equations simultaneously.

You can transform any model developed with the Acoustics Module into a model described by the underlying partial differential equations. This offers a unique way to see the underlying physical laws of a simulation. You can also export the simulation data and results to COMSOL Script or MATLAB. Alternatively, save the model as a Model M-file, a script file that runs in both COMSOL Script and MATLAB. This enables you to incorporate models with other products in those technical computing environments and to efficiently run parametric studies.

Which Problems Can You Solve?

The Acoustics Module application modes handle acoustics in fluids and solids. The application modes for acoustics in fluids support transient, eigenfrequency, time-harmonic, and boundary modal analysis in pressure acoustics and aeroacoustics in compressible, irrotational velocity fields. The application modes for solids support static, transient, eigenfrequency, and frequency response analysis. Further, by coupling fluid and solid application modes, you can solve problems involving acoustic-structure interaction.

All categories are available in both 2D and 3D. In 2D the Acoustics Module offers in-plane application modes for problems with a planar symmetry as well as axisymmetric application modes for problems with a cylindrical symmetry. In addition, you can use the fluid acoustics application modes with 1D and 1D axisymmetric geometries.

Examples of applications that you can successfully simulate with the Acoustics Module include mufflers, loudspeakers, microphones, sound and noise in buildings, underwater acoustics, engine noise and vibration, pressure waves in geophysics, and ultrasonic sensors. For a more detailed description of many of these applications, refer to the matching book that comes with this product, the *Acoustics Module Model Library*.

Fundamentals of Acoustics

What Is Acoustics?

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 atmospheric pressure (about 100,000 pascals), which varies slowly. Associated with a sound pressure wave is a flow of energy. 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 hi-fi system. It is possible to see the movement of a bass speaker cone when it generates sound at a very low frequency. As the cone moves forward, it compresses the air in front of it, causing an increase in air pressure. Then it moves back past its resting position and causes a reduction in air pressure. This process continues, radiating a wave of alternating high and low pressure at the speed of sound.

Some Standard Acoustics Problems

An acoustics analysis can often be categorized as one of the following standard problems or scenarios:

- *The radiation problem*—A vibrating structure (a speaker, for example) radiates sound into the surrounding space. A far-field boundary condition or a PML (perfectly matched layer) is necessary to model the unbounded domain.
- *The scattering problem*—An incident wave impinges on a body and creates a scattered wave. A far-field radiation boundary condition or a PML is necessary.
- *The sound field in an interior space* (such as a room)—The acoustic waves stay in a finite volume so no radiation condition is necessary.
- *Coupled fluid-elastic structure interaction* (structural acoustics)—If the radiating or scattering structure consists of an elastic material, you must consider the interaction between the body and the surrounding fluid. In the multiphysics coupling, the acoustic analysis provides a load (the sound pressure) to the structural analysis, and the structural analysis provides accelerations to the acoustic analysis.

- *The transmission problem*—An incident sound wave propagates into a body, which can have different acoustic properties. Pressure and acceleration are continuous on the boundary.
- *Aeroacoustic problems*—Sound (noise) is generated by turbulent fluid motion or by the interaction between a fluid and a surface.

The Acoustics Module provides application modes with accompanying boundary conditions and example models for all these types of acoustics analyses.

Depending on the basic dependent variable used to model the acoustic field, the acoustical application modes can be divided in two main categories:

- *Pressure Acoustics*—The dependent variable is the acoustic pressure, p.
- A*eroacoustics*—The dependent variable is the potential, ϕ, for the acoustic particle-velocity field, $\mathbf{v} = \nabla\phi$. In the typical situation, the fluid is in motion with a total velocity, $\mathbf{v}_{\text{tot}} = \mathbf{V} + \mathbf{v}$, split into a stationary background-flow velocity, $\mathbf{V}$, and the particle velocity, $\mathbf{v}$, associated with the acoustic waves.

3

Modeling Acoustics

The goal of this chapter is to familiarize you with the modeling procedure in the Acoustics Module. Because this module is totally integrated with COMSOL Multiphysics, the modeling process is similar. This chapter also contains an example model illustrating the central aspects of the simulation process; it steps you through all the stages of modeling—from geometry creation to postprocessing. A number of additional models of differing complexity are presented—complete with step-by-step instructions—in the *Acoustics Module Model Library*. All example models are also provided as COMSOL Multiphysics MPH-files ready for execution.

Format for the Model Descriptions

The way COMSOL Multiphysics orders its toolbar buttons and menus mirrors the basic procedural flow during a modeling session. You work your way from left to right in the process of modeling, defining, solving, and postprocessing a problem using the COMSOL Multiphysics graphical user interface (GUI). Thus, this manual as well as the accompanying *Acoustics Module Model Library* manual and the *COMSOL Multiphysics Model Library* maintain a certain style convention when describing models. The format includes headlines that correspond to each major step in the modeling process; the headlines also roughly correspond to the various GUI modes and menus.

MODEL NAVIGATOR

The **Model Navigator** appears when you start COMSOL Multiphysics or when you restart completely within COMSOL Multiphysics by selecting **New** from the **File** menu or by clicking on the **New** toolbar button. On the **New** tab in the **Model Navigator** you specify the application mode, names of dependent variables, and the analysis type: static, time-harmonic, transient, mode analysis or eigenfrequency. You can also set up a combination of application modes from the Acoustics Module, COMSOL Multiphysics, or any other available module. See the section "Creating and Opening Models" on page 20 in the *COMSOL Multiphysics Quick Start and Quick Reference* for more information about the Model Navigator.

OPTIONS AND SETTINGS

This section reviews basic settings, for example, those for the axes and grid spacing. All settings are accessible from the **Options** menu, and some can be reached by double-clicking on the Status bar. It is often convenient to use the **Constants** dialog box to enter constant parameters for the model or use the dialog boxes that you reach by pointing to **Expressions** to enter expression variables. Advanced models may also need coupling variables. COMSOL Multiphysics maintains a user-defined library of materials accessible through the **Materials/Coefficients Library** dialog box.

GEOMETRY MODELING

The process of setting up a model's geometry requires knowledge of how to use the **Draw** menu and the Draw toolbar. For 2D the details appear in the section "Creating a 2D Geometry Model" on page 39 of the *COMSOL Multiphysics User's Guide*. For 3D you find them under "Creating a 3D Geometry Model" on page 56 of the same book.

BOUNDARY CONDITIONS

You specify the boundary conditions for a model in the **Boundary Settings** dialog box. For details, see "Specifying Boundary Conditions" on page 234 in the *COMSOL Multiphysics User's Guide*. You find the boundary conditions for each acoustics application mode in the chapters about the Acoustics Module application modes.

SUBDOMAIN SETTING

You specify equation parameters and material properties in the **Subdomain Settings** dialog box. For details see "Specifying Subdomain Settings and PDE Coefficients" on page 205 in the *COMSOL Multiphysics User's Guide*. The physical parameters of specific interest for acoustics modeling appear in the chapters about the Acoustics

Module application modes, where you can also learn about the derivation of the equations as well as the boundary conditions.

SCALAR VARIABLES

In the **Application Scalar Variables** dialog box you can examine and modify the values of predefined application-specific scalar variables such as the excitation frequency.

MESH GENERATION

The program must mesh the geometry before it can solve the problem. Sometimes it is sufficient to click the **Initialize Mesh** button on the Main toolbar. In other cases you need to adjust settings in the **Free Mesh Parameters** dialog box and the other mesh-generation tools on the **Mesh** menu. Read more about meshing in "Creating Meshes" on page 286 of the *COMSOL Multiphysics User's Guide*.

COMPUTING THE SOLUTION

To solve a problem, for most cases simply click the **Solve** button on the Main toolbar. In other cases it might be necessary to adjust the solver properties, which you do in the **Solver Parameters** dialog box. For details see "Selecting a Solver" on page 360 of the *COMSOL Multiphysics User's Guide*.

POSTPROCESSING AND VISUALIZATION

The powerful visualization of COMSOL Multiphysics tools are accessible in the program's Postprocessing mode, but to use them you must be familiar with the **Plot Parameters** dialog box and the other postprocessing tools on the **Postprocessing** menu. See "Postprocessing Results" on page 420 in the *COMSOL Multiphysics User's Guide* for details.

ADDITIONAL POSTPROCESSING

For further postprocessing calculations, you can export the solution to COMSOL Script or MATLAB. Details of modeling by programming are available in the *COMSOL Multiphysics Scripting Guide*.

Building and Solving an Acoustics Model

Introduction

This is an introduction to modeling in the Acoustics Module. The step-by-step instructions take you through the process of setting up a model of the sound field created by a cylindrical down-firing subwoofer placed on a floor. The important result of the simulation is the contribution to the system's mechanical impedance induced by the coupling between the speaker membrane and the air inside and outside the speaker enclosure. A polar plot of the far-field is also presented to demonstrate the subwoofer's uniform intensity distribution.

Model Definition

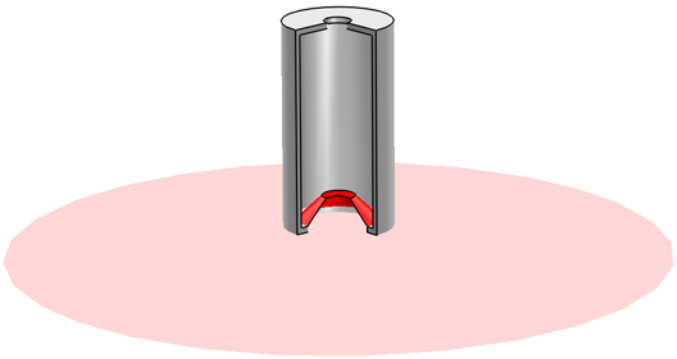


Figure 3-1: The geometry of the subwoofer with a slice removed to show its interior. In the illustration the floor is cut off but in the model it is assumed to extend to infinity.

Figure 3-1 shows the geometry simulated in this model. A down-firing cylindrical subwoofer is placed on a floor bounding an infinite half-space. The subwoofer enclosure has a height of 0.7 m and a diameter of 0.4 m. The bottom of the subwoofer is dominated by a 12-inch (0.3 m diameter) membrane. The top has a vent with a diameter of 0.1 m.

The membrane vibrates harmonically with the acceleration $a = a_0 e^{i\omega t}$ where $a_0 = 100$ m/s^2, and $\omega = 2\pi f$ is the angular frequency (rad/s). The frequencies considered is $f = 50$–70 Hz, which are in the high end of the frequency range that subwoofers are typically used for. It is assumed that the walls of the enclosure are perfectly rigid. The acoustic medium is air with a density of 1.25 kg/m^3 and a sound speed of 343 m/s.

The feet that the speaker would need to stand on are judged to have a negligible effect on the sound field. With this assumption, all geometric features and physics have a rotational symmetry with respect to the axis of the speaker. This makes it natural to set the model up in a 2D axisymmetric application mode.

Results and Discussion

By assuming the membrane is inflexible, you can write Newton's second law for the voice coil and membrane as

$$ma = F_{\text{ext}} + F_{\text{emf}} + F_m + F_a \tag{3-1}$$

where the forces acting on the system have been split in an applied force, F_{ext}, an electrical force opposing the movement of the coil, F_{emf}, a mechanical part, F_m, independent of the acoustic environment, and a term, F_a, containing only the effects of the fluid loading on the membrane. The external force is typically proportional to the applied voltage, while the latter three contributions are proportional to the velocity and directed to oppose the movement. Introducing corresponding mechanical impedances and using the time-harmonic assumption, it holds that

$$F_{\text{ext}} = v(i\omega m + Z_{\text{emf}} + Z_m + Z_a) \tag{3-2}$$

If you know the three impedances as functions of frequency, you can predict the system's electrical and mechanical behavior. The electrical force can be measured or simulated with a blocked voice coil, while Z_m is directly related to the mass of the voice coil and membrane as well as to the stiffness of the baffle keeping the voice coil centered. These contributions are therefore independent of the environment in which the speaker operates.

The acoustic contribution to the mechanical impedance, on the other hand, depends on the enclosure geometry and on reflecting surfaces in the immediate surroundings. In Figure 3-2 you can study the real and imaginary parts of Z_a as functions of frequency in the simulated range. The real part represents radiation and acts as a resistance in the electromechanical system while the imaginary part represents a reactance.

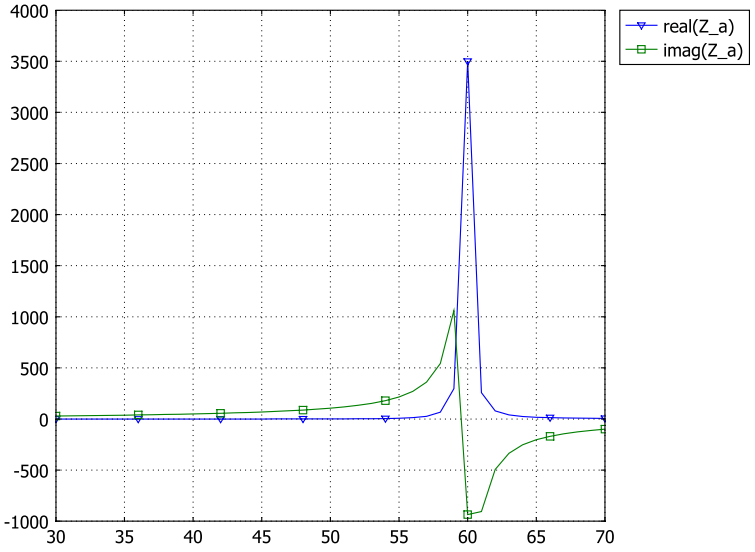


Figure 3-2: The resistive (real) and reactive (imaginary) parts of the contribution of the air to the speaker's mechanical impedance.

Note that the reactance switches sign from positive to negative at approximately 60 Hz. This means that at frequencies below this visible resonance the surrounding air acts as an added mass on the membrane, while at higher frequencies its action is spring-like.

Figure 3-3 displays the far-field sound pressure level in the *rz*-plane as a polar plot where the distance from the origin represents the intensity heard by a far-away observer

multiplied by the observation distance squared. Hence it is normalized so that, on average, it equals the local intensity at a distance of 1 m.

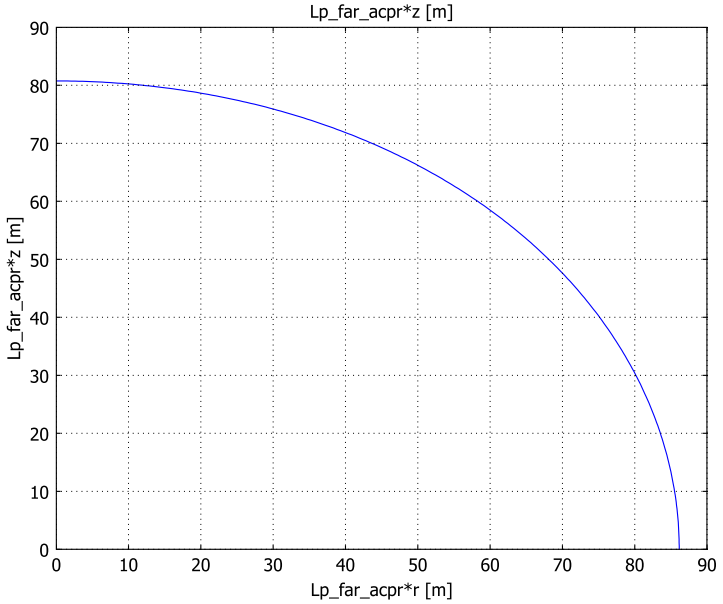


Figure 3-3: Far-field sound level in dB normalized to a distance of 1 m from the floor below the subwoofer. The large wavelength compared to the size of the speaker system makes the response very uniform.

Modeling in COMSOL Multiphysics

This model is set up in 2D axisymmetry using the Pressure Acoustics application mode, which is described in detail in Chapter 5. The modeled physical domain is a hemisphere with a radius of 1 m. To minimize the effect of nonphysical reflections at the exterior boundary of this domain, an absorbing perfectly matched layer (PML) is added outside of it. For information about PMLs in acoustics, see the subsection "Perfectly Matched Layers (PMLs)" on page 37.

The membrane is modeled with zero thickness. To allow the pressure field to be discontinuous across the membrane, the interior and the exterior of the loudspeaker are set up as an assembly with two different parts, connected only at the vent. To learn more about assemblies, see "Using Assemblies" on page 92 of the *COMSOL Multiphysics User's Guide*.

Model Library path: `Acoustics_Module/Tutorial_Models/cylindrical_subwoofer`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Start COMSOL Multiphysics.
2. In the **Model Navigator**, select **Axial symmetry (2D)** from the **Space dimension** list.
3. From the list of application modes select **Acoustics Module>Pressure Acoustics>Time-harmonic analysis**.
4. Click **OK**.

GEOMETRY MODELING

1. Choose **Draw>Specify Objects>Line**. Enter the following space-separated list of coordinates:

r	0 0.06 0.15 0.18 0.18 0.05 0
z	0.2 0.2 0.12 0.12 0.78 0.78 0.78

2. From the **Style** list, select **Closed polyline (solid)**, then click **OK**.
3. Choose **Draw>Specify Objects>Line** again. Enter the following coordinates:

r	0.15 0.2 0.2 0.05 0.05 0.18 0.18 0.15
z	0.1 0.1 0.8 0.8 0.78 0.78 0.12 0.12

4. From the **Style** list, select **Closed polyline (solid)**, then click **OK**.
5. Choose **Draw>Specify Objects>Circle**. Keep the default 1 for the **Radius** and click **OK**.
6. Create another circle, this time with a radius of `1.5`.
7. Choose **Draw>Specify Objects>Square**. Set the **Width** to `1.5`, then click **OK**.
8. Choose **Draw>Create Composite Object**. Type `SQ1*(C1+C2)+CO1-CO2` in the **Set formula** edit field, then click **OK**.
9. Click the **Zoom Extents** button on the Main toolbar.
10. Click the **Split Object** button on the Draw toolbar.

11 Select the objects CO1 and CO4 (by Shift-clicking in the geometry), then click the **Union** button on the Draw toolbar.

12 Choose **Draw>Use Assembly**.

Your geometry should now contain two objects, CO2 and CO3, and look like that in Figure 3-4.

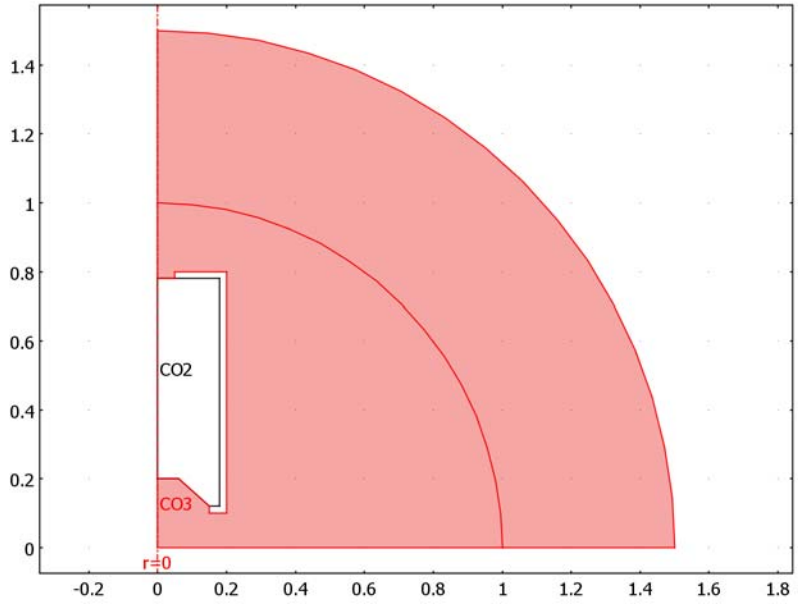


Figure 3-4: Model geometry.

OPTIONS AND SETTINGS

This model uses a constant to represent the membrane's peak acceleration, an integration variable to calculate the total force acting on the membrane, and expression variables to define the far-field sound pressure level and the mechanical impedance of the membrane-air system.

1 Open the **Constants** dialog box from the **Options** menu and define the constant `a0` according to the following table; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
a0	10[m/s^2]	Peak acceleration

The air load on the membrane acts in the direction normal to the membrane surface and with magnitude equal to the acoustic pressure. Due to the axial symmetry, the

resultant force must be in the axial direction and equal to the integral of the z component of the load vector

$$F_z = \int_S p n_z ds$$

where n_z is the z component of the surface unit normal (directed out from the acoustic domain) and the integral is taken over both sides of the membrane. Note that you must multiply the integrand by the factor $2\pi r$ to obtain an integral over the true 3D surface.

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

3 Select the membrane boundaries and create an integration coupling variable for the force acting on the membrane as follows:

BOUNDARIES	NAME	EXPRESSION	INTEGRATION ORDER	GLOBAL DESTINATION
3, 9, 17, 20	F_a	2*pi*r*p*nz	4	yes

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

5 Now you will enter the expression for the impedance (see Equation 3-2 on page 15). Choose **Options>Expressions>Global Expressions** and add the following expression:

NAME	EXPRESSION	DESCRIPTION
Z_a	-F_a/(a0/(i*2*pi*freq_acpr))	Mechanical impedance

6 Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 Choose **Physics>Subdomain Settings**.

2 Select Subdomain 2 and click the **PML** tab.

3 Select **Type of PML: Spherical**. Select the **Absorbing in radial dir.** check box.

The software's automatic detection of the PML's geometric properties correctly determines the PML's width and inner radius, and the default value for the PML center point applies. The evanescent part of the wave leaving from the vent is expected to decay with a characteristic length much shorter than the wavelength. Increasing the PML scaling exponent helps the PML resolve the damping of the evanescent wave.

4 Enter 2 in the **PML scaling exponent** edit field.

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

Boundary Settings

Because the geometry of the subwoofer model is represented as an assembly, you need to create identity pairs where the parts of the assembly are connected and interact with each other. In this case the pressure is continuous across the top vent, which therefore must be made into an identity pair.

1 Choose **Physics>Identity Pairs>Identity Boundary Pairs**.

2 Click the **New** button. Select the check box for Boundary 18 in the **Source boundaries** list and the check box for Boundary 5 in the **Destination boundaries** list. Click **OK** to close the dialog box.

You need to set the inward normal acceleration on both sides of the membrane to drive the model. Because the membrane's movement is in the axial direction only, the inward normal component of the acceleration at the surface equals the magnitude multiplied by the negative z component of the normal unit vector. This holds because the normal unit vector is directed out from the acoustics domain.

3 Choose **Physics>Boundary Settings** and apply the following boundary conditions; when done, click **OK**.

SETTINGS	BOUNDARIES 1, 4, 6, 16	BOUNDARIES 3, 9, 17, 20
Boundary condition	Axial symmetry	Normal acceleration
a_n	-	-a0*nz

Far-Field Settings

If you know the pressure and normal velocity on a closed surface containing all sources and objects in the model, you can evaluate the far-field radiation pattern as an explicit integral for each direction. The far-field pattern is defined as a limit

$$p_{\text{far}} = \lim_{\rho \to \infty} \rho e^{ik\rho} p$$

where ρ is the distance from the center of the coordinate system and k is the wave number. The magnitude of the far field is comparable to the pressure at a spherical surface with a radius of 1 m but shows the radiation pattern experienced by a far-away listener.

The hard floor is excluded from the integral by noting that the radiation pattern is exactly the same as if replacing the floor with a mirror image of the subwoofer that is

instead included in the integral. You need not worry about any of these details, however, because the automatic far-field calculation sets up the integral for you.

1. Still in the **Boundary Settings** dialog box, click the **Far-Field** tab.
2. Select all physical boundaries of the subwoofer by Shift-clicking in the geometry or Ctrl-clicking in the **Boundary selection** list. The selected boundary numbers should be 3, 7–12, 17, and 19–22. Note that you must select the boundaries representing both sides of the membrane in order to include all contributions to the far field.
3. Click the **Name** edit field and enter `p_far`. Select another edit field on the same row in the table to see the default values of the settings for this far-field variable.
4. Select the **Symmetry planes: z = 0** check box. Keep the default **Symmetric pressure** option. This means that the far field accounts for the perfectly reflecting floor by adding contributions from an imagined mirror copy (with respect to $z = 0$) of the subwoofer.
5. Click **OK**.

MESH GENERATION

Solutions to acoustics models are wave-like with a fixed wavelength that must be resolved. In addition, small features in the geometry can induce high local pressure gradients that must be resolved properly to obtain a consistent global solution. In this particular model, the wavelength is greater than 5m so the focus is on resolving the geometry.

1. Choose **Mesh>Free Mesh Parameters**.
2. Select **Finer** from the **Predefined mesh size** list.
3. Go to the **Boundary** page, select Boundaries 5, 7, and 18, then set the **Maximum element size** to `0.001`.
4. Click **OK**.
5. Click the **Initialize Mesh** button on the Main toolbar

COMPUTING THE SOLUTION

Time-harmonic acoustics can be solved either with the stationary solver for a single frequency or with the parametric solver when you specify a frequency range. The frequency parameter `freq_acpr` is defined in the **Application Scalar Variables** dialog box (select **Physics>Scalar Variables**), but the parametric solver lets you override this—and almost any other variable—during the solution.

1. Choose **Solve>Solver Parameters**.

2 Select the **Parametric** solver in the list on the left.

3 Set **Name of parameter** to `freq_acpr` and enter `30:1:70` in the **List of parameter values** edit field.

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

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

POSTPROCESSING AND VISUALIZATION

The default plot shows the pressure field for the last parameter value, 70 Hz. By default, the postprocessing algorithms try to smooth discontinuous fields. To avoid this happening at the membrane, where the field really should be discontinuous, follow these steps:

1 Click the **Plot Parameters** button on the Main toolbar to open the **Plot Parameters** dialog box.

2 Go to the **Surface** page and clear the **Smooth** check box.

3 Click **Apply** to update the plot, which now resembles this screen shot:

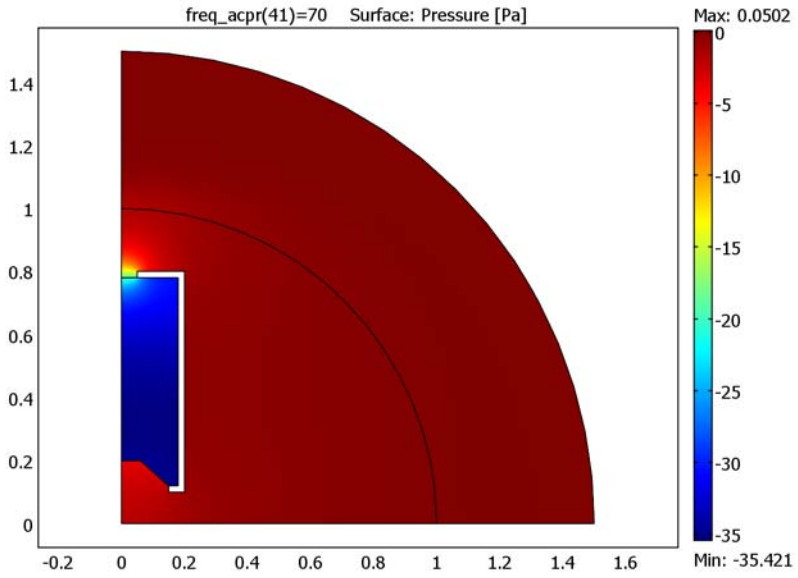


Figure 4: Default plot showing the acoustic pressure in and around the subwoofer.

Note that the pressure drops rapidly toward zero in the PML domain. To get a better view of the damping try plotting the sound pressure level in dB:

1. While still on the **Surface** page in the **Plot Parameters** dialog box, choose **Pressure Acoustics (acpr)>Sound pressure level** from the **Predefined quantities** list.
2. Click the **Height Data** tab, select the **Height Data** check box, then also choose **Pressure Acoustics (acpr)>Sound pressure level** from this **Predefined quantities** list.
3. Click **OK**.

With a visual inspection you can see that the pressure drop in the PML is roughly 50 dB. This means that the part of the wave that is reflected at the exterior boundary experiences a total of 100 dB damping before it returns to physical domain. In other words, the reflected wave has virtually no effect on the solution.

Proceed to display the previously defined impedance variable `Z_a` as function of frequency:

1. Choose **Postprocessing>Global Variables Plot**.
2. Enter `real(Z_a)` in the **Expression** edit field and click the **>** button immediately to the right of the field.
3. Enter `imag(Z_a)` in the same field and again click the **>** button.
4. Click the **Line Settings** button.
5. Select **Cycle** in the **Line marker** list and select the **Legend** check box.
6. Click **OK**, then **OK** again to close the **Global Variables Plot** dialog box and display the result, which should be similar to Figure 3-2.

Finally check that the far-field pressure distribution is indeed as uniform as you would expect from a subwoofer. The following trick gives you a polar plot where the distance from the origin represents the sound pressure level normalized to a distance of 1 m.

1. Choose **Postprocessing>Domain Plot Parameters**.
2. Select the 70 Hz solution in the **Solution to use** list.
3. Click the **Line/Extrusion** tab and select Boundary 14.
4. Type `Lp_far_acpr*z` in the **Expression** edit field.
5. Click the **Expression** option button and then click the **Expression** button on the **x-axis data** panel.
6. In the dialog box that appears, type `Lp_far_acpr*r` in the **Expression** edit field.

7 Click **OK**, then **OK** again to see the plot of the far-field pressure as a function of the polar angle. It takes a few seconds to appear, but when it does it should look like Figure 3-3.

To get a clearer picture of the subwoofer and the sound field, display only the physical boundaries in black and suppress the PML.

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

2 Select Boundaries 1, 4–6, 13–16, and 18, then click **OK**.

3 Choose **Options>Suppress>Suppress Subdomains**.

4 Select Subdomain 2, then click **OK**.

5 Choose **Postprocessing>Plot Parameters**.

6 On the **Height Data** tab of the **Surface** page, clear the **Height data** check box.

7 On the **General** page of the **Plot Parameters** dialog box, clear the **Geometry edges** check box and select the **Boundary** check box in the **Plot type** list.

8 On the **Boundary** page, click the **Uniform color** option button. Click the **Color** button and select a black color.

9 Click **OK** to see the plot of the acoustic pressure again, this time with only the physical boundaries and no PML displayed.

Some Aspects of Acoustics Modeling

There are certain difficulties that often arise when modeling acoustics, such as the rather severe requirements on the mesh resolution, the modeling of artificial boundaries, and the modeling of real-world damping materials. This section discusses how to deal with these issues and describes some of the features included in the Acoustics Module for simplifying the modeling process.

Resolving the Waves

Solutions to acoustic problems are wave like. 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.

For the solution on the discrete grid to have any meaning at all there has to be at least two *degrees of freedom* (DOFs) per wavelength in the direction of propagation, but such coarse a solution is useless in practice. 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 will be about

- 12 times the model length measured in wavelengths in 1D
- 144 times the model area measured in wavelengths squared in 2D
- 1728 times the model volume measured in wavelengths cubed in 3D

Before starting a new model, try to estimate the required number of DOFs using these rules of thumb. The maximum number of DOFs that can be solved for differs between computer systems, but a 32-bit system can usually deal with somewhere from a few hundred thousand up to a million DOFs. Even on a 64-bit system, more than a few million DOFs is cumbersome to handle.

USING LAGRANGE ELEMENTS (DEFAULT)

When creating an unstructured mesh for use with the default 2nd-order Lagrange elements, set the maximum element size, $h_{\max}$, to about 0.2λ. Because all elements in the constructed mesh are smaller than $h_{\max}$, the limit is set larger than the actual required element size. After meshing the model, check the total number of DOFs

against the model volume and the above rules of thumb. If the mesh turns out to be too coarse or too fine on average, try to change $h_{\max}$ accordingly.

Note that unstructured meshes are generally better than structured 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 differs very much between directions parallel to the grid lines and directions rotated 45 degrees about one of the axes.

USING THE ULTRAWEAK VARIATIONAL FORMULATION

In the ultraweak variational formulation (UWVF), the rules for how to determine a suitable mesh size are different. This formulation uses Ultraweak Helmholtz elements (`shuwhelm`), whose basis functions are free-space solutions to the Helmholtz equation, that is, plane waves. Because the finite elements in the UWVF thus contain information about the solution to the wave equation, the mesh elements must be larger in relation to the wavelength than when using ordinary Lagrange elements. As a rule of thumb, aim for a maximum mesh-element size, $h_{\max}$, of roughly 2–3 times the wavelength. A too fine mesh can lead to convergence problems, while a too coarse mesh reduces the accuracy of the solution. Also, for the UWVF to work well, the mesh should be reasonably uniform.

Because it requires a much coarser mesh, the UWVF is suitable for models where the wavelength is small, and for which ordinary Lagrange elements therefore would need an excessively dense mesh.

Note: Because of the coarse mesh, when visualizing a solution computed in the ultraweak variational formulation, you need to raise the element refinement on the **General** page of the **Plot Parameters** dialog box. An element-refinement factor around 20 gives good result for the default UWVF elements with 20 plane-wave basis directions.

Damping

Fluids with bulk 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.

POROUS ABSORBING MATERIALS

For frequency-domain modeling, the most convenient and compact description of a damping material (material here refers to the homogenization of a fluid and a porous solid) is given by its complex wave number, k_c, and complex impedance, Z_c, both functions of frequency. Knowing these properties, you can define a complex speed of sound as $c_c = \omega / k_c$ and a complex density as $\rho_c = k_c Z_c / \omega$.

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. This data can be used directly as input to COMSOL Multiphysics' interpolation functions to define k_c and Z_c in the application modes.

Sometimes, however, you cannot obtain acoustic properties 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. For a highly porous material with a rigid skeleton, the well-known model of Delany and Bazley can estimate the complex wave number and impedance as functions of frequency and flow resistivity.

For further details on how to implement damping in your acoustics model, see page 71 in the "Pressure Acoustics" chapter.

DAMPING AT BOUNDARIES

Acoustics in closed ducts and cavities appears to be easier to deal with than exterior problems because no artificial boundary condition is necessary. On the other hand, real-world cavity walls are usually either treated in some way (*lined*) or elastic in themselves.

The problem is that a liner typically reflects part of the wave and does so not at the interface with the domain but somewhere inside the liner or at its back wall against whatever structure is outside. This means that a liner boundary condition must contain more information about the outside world than an absorbing boundary. It also means that a real-world liner cannot be adequately described by a local boundary condition because waves at oblique incidence cause waves to propagate in the tangential direction inside the liner layer.

In fact, there seems to be no final answer as to how the process inside a porous liner is most accurately modeled. Various assumptions can be made about the interaction

between the fluid pressure waves and the liner material and about boundary conditions between liner and free fluid and at the back of the liner. The most accurate ways to deal with the situation includes modeling the actual liner layer. It is only possible to use a general impedance boundary condition for thin liners and when the angle of incidence is known for a liner that cannot be assumed locally reacting, an assumption that rarely holds with any justification. For an example that nevertheless uses this assumption, see "Flow Duct" on page 101 in the *Acoustics Module Model Library*.

Artificial Boundaries

In most cases, the acoustic wave pattern which is to be simulated is not contained in a closed cavity. That is, there are boundaries in the model which 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*.

Such conditions should ideally contain complete information about the outside world, but this is obviously not practical. After all, the artificial boundary was introduced to avoid spending 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. For obvious reasons, this is difficult in general.

One particular case which occurs frequently in acoustics concerns boundaries which 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 *nonreflecting boundary conditions*, of which two kinds are available in the Acoustic Module: *matched boundary conditions* and *radiation boundary conditions*. The former apply primarily to wave guide ports connected to a cavity, while the latter approximate the boundary at infinity in an exterior problem. A drawback of these boundary conditions is that they are not perfectly nonreflecting when subjected to a general incoming wave. They are described in more detail in the section "Boundary Conditions" on page 76 and onward.

The models "Absorptive Muffler" on page 74 and "Muffler with Perforates" on page 154 of the *Acoustics Module Model Library* both use a nonreflecting boundary condition of the radiation type.

Evaluating the Acoustic Field in the Far-Field Region

The Acoustics Module provides functionality for evaluating 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 on how to use the far-field functionality appear in the section "Far-Field Modeling" on page 90.

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 their 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 rule of thumb in line with the above definitions 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, the following definition of the far-field region can therefore be applied:

$$R > \frac{8a^2}{\lambda} = \frac{8}{2\pi} ka^2 .$$

In this formula, a is the radius of a sphere enclosing all objects and sources, λ is the wavelength, and k is the wave number. The second way of writing 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.

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 can be written as a boundary integral in terms of quantities evaluated on the surface. To evaluate this so-called *Helmholtz-Kirchhoff* integral, it is necessary to know both Dirichlet and Neumann values on the surface. Applied to acoustics, this means that if you know the pressure *and* its normal derivative (which is related to the normal velocity) on a closed surface, you can calculate the acoustic field 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 therefore is simply

$$G(\mathbf{R}, \mathbf{r}) = \frac{e^{-ik|\mathbf{r}-\mathbf{R}|}}{4\pi|\mathbf{r}-\mathbf{R}|}\,.$$

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

$$p(\mathbf{R}) = \frac{1}{4\pi}\int_S \frac{e^{-ik|\mathbf{r}-\mathbf{R}|}}{|\mathbf{r}-\mathbf{R}|}\left(\nabla p(\mathbf{r}) + p(\mathbf{r})\frac{(1+ik|\mathbf{r}-\mathbf{R}|)}{|\mathbf{r}-\mathbf{R}|^2}(\mathbf{r}-\mathbf{R})\right)\cdot \mathbf{n}\, dS\,, \qquad (4\text{-}3)$$

while in 2D, the Hankel function leads to a slightly different expression

$$p(\mathbf{R}) = -\frac{i}{4}\int_S \left(H_0^{(2)}(k|\mathbf{r}-\mathbf{R}|)\nabla p(\mathbf{r}) + kp(\mathbf{r})\frac{H_1^{(2)}(k|\mathbf{r}-\mathbf{R}|)}{|\mathbf{r}-\mathbf{R}|}(\mathbf{r}-\mathbf{R})\right)\cdot \mathbf{n}\, dS \quad (4\text{-}4)$$

For axially symmetric geometries, the full 3D integral must be evaluated. To this end, 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 4-3 and Equation 4-4, use the **Full integral** option in the settings for the far-field variables (see "Specifying Variables for Far-Field Postprocessing" on page 90).

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 4-3 when $|\mathbf{R}|$ goes to infinity and ignoring the rapidly oscillating phase factor, the far field, p_{far}, is defined as

$$p_{\text{far}}(\mathbf{R}) = -\frac{1}{4\pi}\int_S e^{ik\frac{\mathbf{r}\cdot\mathbf{R}}{|\mathbf{R}|}}\left(\nabla p(\mathbf{r}) - ikp(\mathbf{r})\frac{\mathbf{R}}{|\mathbf{R}|}\right)\cdot \mathbf{n}\, dS\,.$$

Note that 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 exponentials, 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 the circumferential wave number $m = 0$, the expression is

$$p_{\text{far}}(\mathbf{R}) \cong -\frac{1}{2}\int_S re^{ik\frac{zZ}{|\mathbf{R}|}}[J_0\left(\frac{krR}{|\mathbf{R}|}\right)\nabla p(\mathbf{r})\cdot\mathbf{n} - \frac{ikp(\mathbf{r})}{|\mathbf{R}|}\left(in_r RJ_1\left(\frac{krR}{|\mathbf{R}|}\right) + n_z ZJ_0\left(\frac{krR}{|\mathbf{R}|}\right)\right)]dS$$

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 for the far-field variables (see "Specifying Variables for Far-Field Postprocessing" on page 90).

THE ELKERNEL ELEMENT

The above 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 code that hides all details from the user. For further details, see the entry on `elkernel` on page 93 of the *COMSOL Multiphysics Reference Guide*.

Solving Large Acoustics Problems Using Multigrid

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

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 with simple smoothers such as Jacobi. 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 geometric multigrid as linear system solver together with simple smoothers, you must ensure that the Nyquist criterion is fulfilled on the coarsest mesh; see "Notes on the Efficiency of Smoothers" on page 522 in the *COMSOL Multiphysics Reference Guide*. 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 (see "The GMRES Iterative Solver" on page 512 in the *COMSOL Multiphysics Reference Guide*) as linear system solver with geometric multigrid as preconditioner. 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 linear system solver with GMRES as smoother or FGMRES (see "The FGMRES Iterative Solver" on page 513 in the *COMSOL Multiphysics Reference Guide*) as linear system solver with geometric multigrid as preconditioner (where GMRES is used as smoother). Using GMRES/FGMRES as outer iteration and smoother removes the requirements on the coarsest mesh. When you use GMRES as smoother for the multigrid preconditioner you must use FGMRES for the outer iterations because such a preconditioner is not constant (see Ref. 1). 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.

To summarize, the options you can try when solving large acoustics problems are (in increasing order of robustness and memory requirements):

- If the Nyquist criterion is fulfilled on the coarsest mesh, try to use geometric multigrid as linear system solver with default smoothers. The default smoothers are fast and have small memory requirements. You get this option if you select **Geometric multigrid** in the **Linear system solver** list on the **General** page of the **Solver Parameters** dialog box.
- An option more robust than the above is to use GMRES as linear system solver with geometric multigrid as preconditioner (where default smoothers are used). GMRES requires memory for storing search vectors. To get this option, select **GMRES** as **Linear system solver** and **Geometric multigrid** as **Preconditioner**.
- If the above step does not work, try to use geometric multigrid as linear system solver with GMRES as smoother. See below for recommended settings for the smoother. When using GMRES as smoother, the software stores search vectors on

all multigrid levels and for both presmoothing and postsmoothing. GMRES is also a slower smoother than the simple iterations.

- If the solver still has problems to converge, try to use FGMRES as linear system solver with geometric multigrid as preconditioner (where GMRES is used as smoother). See below for recommended settings for the smoother and FGMRES. In addition to the memory used by the previous step this option also stores search vectors for the outer FGMRES iteration.

Note: The second option of the above four can sometimes be used successfully even when the Nyquist criterion is not fulfilled on coarser meshes. Because GMRES is not used as smoother, this option might find a solution faster than the last two options even if a large number of outer iterations are needed for convergence.

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 (see "Constructing a Multigrid Hierarchy" on page 522 in the *COMSOL Multiphysics Reference Guide*).

If the coarse mesh still is too fine for a direct solver, try using an iterative solver with 5–10 iterations as coarse solver (see "Settings for the Coarse Solver" on page 527 in the *COMSOL Multiphysics Reference Guide*).

Use the V-cycle as **Multigrid cycle** (the default). If this does not work, try first the F-cycle and then the W-cycle.

To use GMRES as presmoother, select **Presmoother** in the tree in the **Linear System Solver Settings** dialog box. Try to set the **Number of iterations** to 2 or 3. It is also recommended that you turn off the preconditioner for the smoother by selecting **None** from the **Preconditioner** list.

When using GMRES as postsmoother, try to use somewhere between 10 and 40 postsmoothing iterations. This makes each iteration slow, but using too few postsmoothing steps might decrease the convergence rate considerably. It is also recommended that you turn off the preconditioner for the smoother by selecting **None** from the **Preconditioner** list.

When using FGMRES as an outer iteration, if too much memory is needed by the algorithm try decreasing the **Number of iterations before restart** from the default 50 to, say, 20. If the memory requirements still are too large, and many GMRES

postsmoothing steps are used, try decreasing the **Number of iterations before restart** for the postsmoother to 5–10. Note that this might slow down the convergence rate considerably.

Reference

1. Saad, Y., "A flexible inner-outer preconditioned GMRES algorithm," *SIAM J. Sci. Statist. Comput.*, vol. 14, pp. 461–469, 1993.

Perfectly Matched Layers (PMLs)

In many scattering and waveguide-modeling problems, you cannot describe the incident radiation as a plane wave with a well-known direction of propagation. In such situations, consider using *perfectly matched layers* or PMLs (see Ref. 1). A PML is strictly speaking not a boundary condition but an additional domain that absorbs the incident radiation without producing reflections. It provides good performance for a wide range of incidence angles and is not particularly sensitive to the shape of the wave fronts. The PML formulation introduces a complex-valued coordinate transformation under the additional requirement that the wave impedance should not be affected.

The following section describes how to use the PMLs in the Acoustics Module to create planar, cylindrical, and spherical PMLs. For information about PMLs for elastic waves in solids, see the section "Perfectly Matched Layers (PMLs)" on page 171 in the "Structural Mechanics Application Modes" chapter.

PML Implementation

For a PML that absorbs waves in the coordinate direction ξ, the Acoustics Module uses the following coordinate transformation inside the PML:

$$\xi' = \mathrm{sign}(\xi - \xi_0)|\xi - \xi_0|^n \frac{L}{\delta\xi^n}(1 - i) \tag{4-5}$$

The scaled PML width, L; the coordinate of the inner PML boundary, ξ_0; and the (actual) width of the PML, $\delta\xi$, are input parameters for each orthogonal absorbing coordinate direction. The scaling exponent, n is an input parameter for each PML subdomain.

The default value for L is one wavelength, $\lambda = c_s/f$, which is appropriate for acoustic waves propagating along the absorbing coordinate direction, ξ. To preserve the attenuation level for obliquely incident waves, you need to adjust the scaled PML width; for a wave with wave vector $\mathbf{k}$, the optimal value for L in the coordinate direction ξ is $2\pi/|\mathbf{k} \cdot \mathbf{e}_\xi| = \lambda/|\cos\vartheta|$, where $\mathbf{e}_\xi$ is a unit vector in the ξ direction, and ϑ is the angle between $\mathbf{k}$ and $\mathbf{e}_\xi$. Thus, you should multiply the default value for the scaled PML width by the factor $|\cos\vartheta|^{-1}$. Note that if you increase L, you need to make sure that the mesh resolution is sufficient to resolve the number of wavelengths that fit inside the adjusted scaled width.

The default value of the scaling exponent is 1, which gives a linear scaling that works well in most cases. For scattering problems and models where different wavelengths should be absorbed (outside waveguides, for example) you can increase the exponent somewhat, the useful range for n being roughly between 1 and 2. Increasing the exponent allows you to use fewer mesh elements to resolve wavelengths much smaller than the scaled PML width.

The parameters ξ_0 and $\delta\xi$ get default settings that the software deduces from the drawn geometry and stores in so-called guess variables. You can inspect the values of the guess variables on the **Variables** page of the **Subdomain Settings - Equation System** dialog box or at the corresponding node of the **Model Tree**.

The default settings defined by the guess variables work nicely in most cases, but they might fail for PML subdomains of a nonstandard shape. Examples of geometries that work nicely are shown in the following figures for each of the available PML types:

- **Cartesian**—PMLs absorbing in Cartesian coordinate directions.
- **Cylindrical**—PMLs absorbing in cylindrical coordinate directions from a specified axis. For axisymmetric geometries the cylinder axis is the z-axis.
- **Spherical**—PMLs absorbing in the radial direction from a specified center point.

For each of the above PML types, you can choose the coordinate directions in which the PML absorbs waves, that is, for which directions a coordinate transformation of the type Equation 4-5 applies. To allow complete flexibility in defining a PML there is, in addition, a fourth option:

- **User defined**—General PMLs or domain scaling with user-defined coordinate transformations.

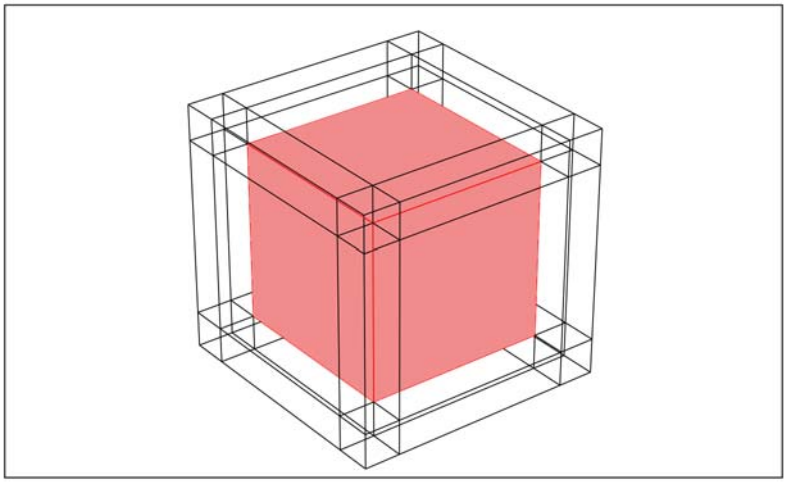


Figure 4-1: A cube surrounded by typical PML regions of the type "Cartesian."

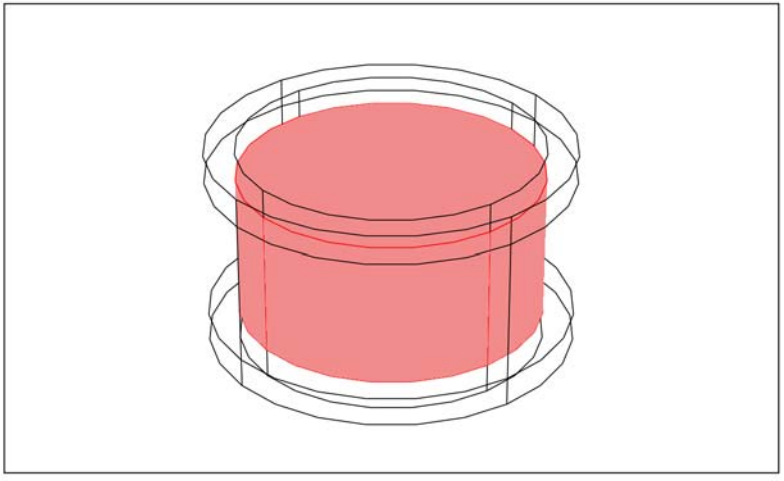


Figure 4-2: A cylinder surrounded by typical cylindrical PML regions.

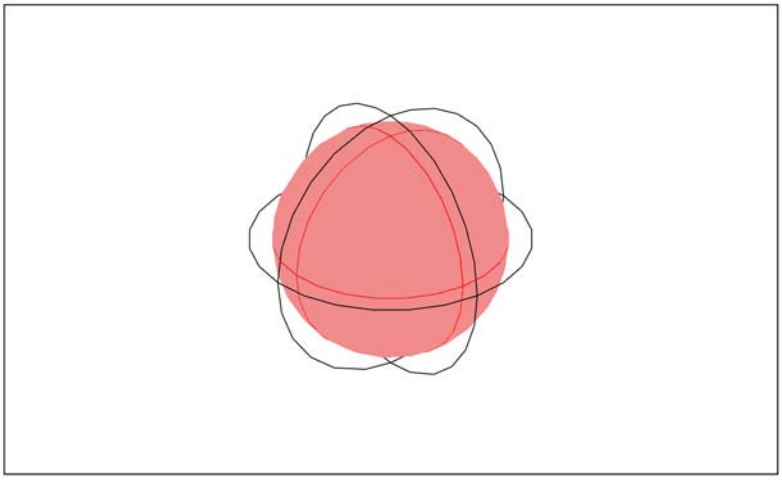


Figure 4-3: A sphere surrounded by a typical spherical PML region.

How to Set Up a PML

This is a tutorial example showing how to add PMLs. In the Acoustic Module you can define perfectly matched layers by adding additional subdomains outside the boundaries that you want to model as absorbing. This model shows a wave propagating from a vibrating cylinder in two dimensions.

DOMAIN EQUATIONS

The model solves the following equation for the acoustic pressure:

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

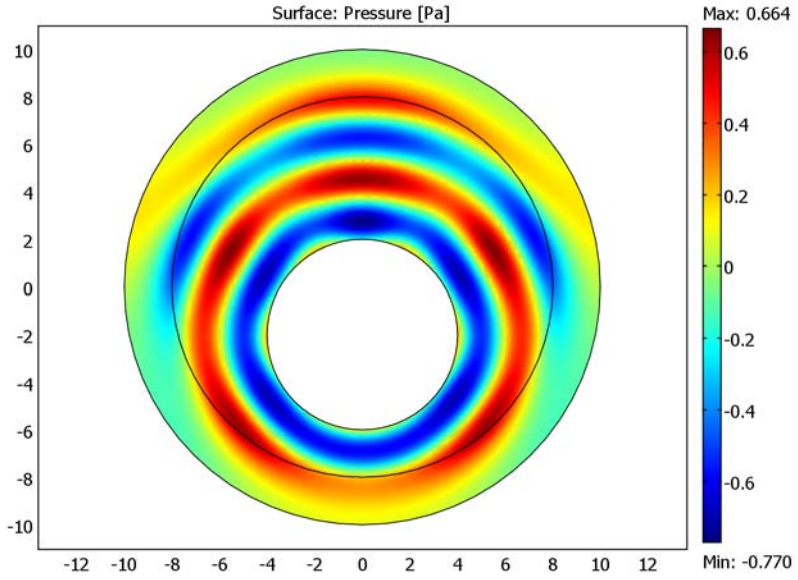
To define the PMLs you introduce a new subdomain around the air domain representing absorbing layers. In this model the cylinder is positioned slightly off center in the air domain because this geometry gives a better view of how the PML absorbs the waves.

BOUNDARY CONDITIONS

The cylinder vibrates with a normal acceleration amplitude equal to 1 m/s^2.

RESULTS

The following figure shows how the wave is absorbed by the surrounding PML. Notice how the amplitude quickly decreases inside the PML. Notice also that there is no distortion of the circular wave fronts inside the air domain because of reflection.



Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator** open the **Acoustics Module** folder, then select **Pressure acoustics>Time-harmonic analysis**.
2. Click **OK**.

GEOMETRY MODELING

1. Draw a circle by selecting **Specify Objects>Circle** from the **Draw** menu.
2. Set the **Radius** to 8 and the **Center** to $x = 0$ and $y = 2$. Click **OK**.
3. Draw a new circle with radius 10 centered at $x = 0$ and $y = 2$.
4. Press Ctrl+A to select both circles. Click the **Union** button on the Draw toolbar.

5 Draw a new circle with a radius of 4 and its center at the origin.

6 Press Ctrl+A to select both objects, then click the **Difference** button.

PHYSICS SETTINGS

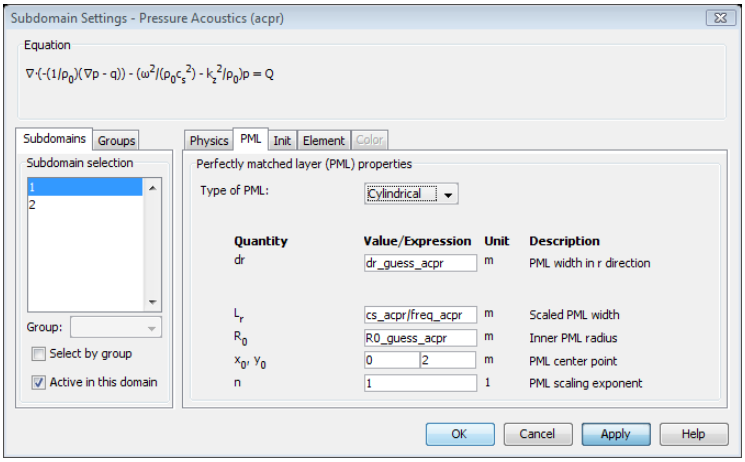
Subdomain Settings

1 Open the **Subdomain Settings** dialog box.

2 Click the **PML** tab, and select Subdomain 1.

3 Choose **Cylindrical** from the **Type of PML** list.

This activates a set of PML-related edit fields in the dialog box; see the following picture.

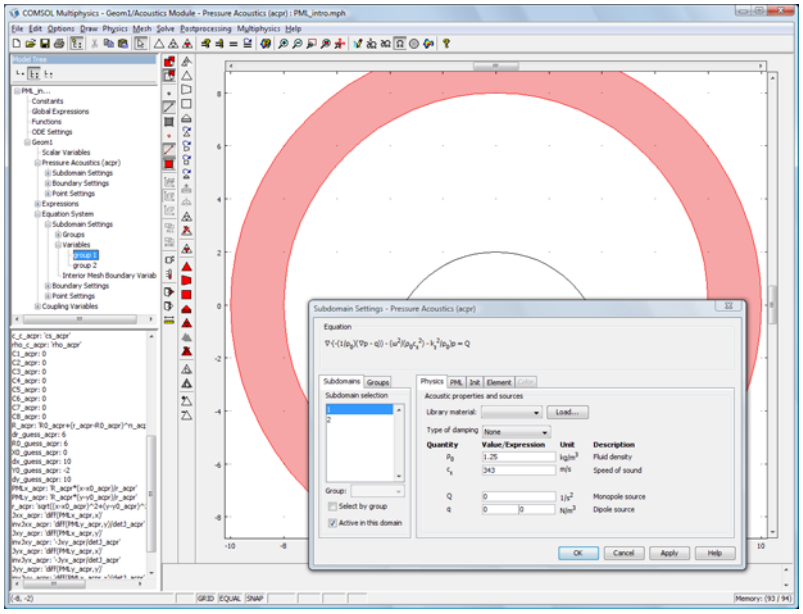


4 Set the *y*-coordinate of the **PML center point** to 2.

For the **Scaled PML width** and the **PML scaling exponent** you can use the default values, and the software correctly deduces the geometric PML parameters (the **PML width in *r* direction**, and the **Inner PML radius**) using the so-called guess variables.

To inspect the values of the applicable guess variables, `R0_guess_acpr`, do as follows: If the model tree is not already open, click the **Model Tree** button on the Main toolbar and select the **Detail** view. Choose **[untitled]>Geom1>Equation System>Subdomain Settings>Variables>group1** in the model tree. All variables defined on Subdomain 1 (the annular PML region) appear

in the area below the model tree together with their respective values; to see the guess variables you need to scroll down the list.



5 Click **OK** to confirm the PML properties and close the dialog box.

Boundary Conditions

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

2 Select Boundaries 5, 6, 9, and 10.

3 Select **Normal acceleration** in the **Boundary condition** list and set the acceleration to `1`.

4 Click **OK**.

MESH GENERATION

To resolve the waves, you need to use a finer mesh than the default. The frequency is 100 Hz which gives a wavelength of about 3.4 m.

1 Open the **Free Mesh Parameters** dialog box, click the **Custom mesh size** button, and set the **Maximum element size** to `3.4*0.2`.

2 Click **OK**.

3 Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

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

Reference

1. J.-P. Bérenger, "A perfectly matched layer for the absorption of electromagnetic waves," *J. Comput. Phys.*, vol. 114, pp. 185–200, 1994.

Scattered-Field Formulation

Most common applications of acoustics can be loosely classified into cavity and duct problems, radiation problems, and scattering problems. The latter type differs from the rest in that the real quantity of interest is not the total pressure field but the, usually small, part of an incident wave reflected from a structure. When modeling a scattering problem in terms of the total pressure field, numerical noise in the large-amplitude incident wave tends to mask much of the detail in the all-important scattered wave.

To avoid the numerical problems associated with resolving contributions of wildly different amplitudes, you can choose to model only the scattered part of the pressure field. The basic idea is simple: write the pressure field as the sum of a known part—the incident wave p_i—and an unknown part—the scattered field p_s. Then insert this sum into the equation governing the total pressure field, and you obtain an equation for the scattered field only.

The scattered-field formulation implemented in the Acoustics Module makes one further assumption: that the incident field p_i by itself solves the original equation with the scattering object removed. When the incident field is simple, like a plane wave, it can be prescribed explicitly, but, for example, for scattering from an object inside a cavity, the incident wave must be solved for as a separate dependent variable.

Selecting the scattered-field formulation alters the implementation of most boundary conditions. Nonreflecting boundary conditions are simplified because no incident wave must be accounted for. The incident field instead appears in the boundary conditions used at the surface of the scattering target.

When using PMLs to replace nonreflecting exterior boundary conditions in a scattering problem, you are forced to choose the scattered-field formulation rather than the full-field formulation. The reason is that it is quite difficult to apply an incident field on the boundary between the actual domain and the PML, and pushing the incident wave through the PML from the outside distorts the wave fronts if the PML is not planar.

Application Modes with Scattered Fields

The scattered-field formulation can be selected for the Pressure Acoustics application mode. It is accessible directly in the **Model Navigator** as the analysis type **Time-harmonic**

analysis, scattered wave. With the application mode already running, you can choose **Scattered wave** from the **Solve for** list in the **Application Mode Properties** dialog box.

The chosen name for the dependent variable (the default is p) in the **Model Navigator** represents either the total field or the scattered field according to the setting in **Application Mode Properties** dialog box. The scattered pressure is always available as `p_s`. If you solve for the total pressure, this variable is the difference between the total pressure and the incident field specified in the **Application Scalar Variables** dialog box. Conversely, the total pressure field is always available as `p_t`, which is defined as the sum of the incident field and the scattered field when solving only for the latter.

Example Model

The benchmark model "Scattering from a Plate with Ribs" on page 214 in the *Acoustics Module Model Library* uses PMLs to truncate the domain and is therefore set up using the scattered-field formulation.

Material Libraries

A useful feature in COMSOL Multiphysics is the Materials/Coefficients library. In addition to the Basic Material Properties library the Acoustics Module extends this library with two extra material libraries:

- MEMS Material Properties, an extended solid material library for MEMS applications. See "MEMS Material Properties Library" on page 48.
- Piezoelectric Material Properties, a material library with 23 common piezoelectric materials. See the section "Piezoelectric Material Properties Library" below.

The Basic Material Properties library is included with COMSOL Multiphysics and contains properties for a limited number of basic solid materials, given as constants, and temperature-dependent properties for air and water, given as functions.

For more information about using the **Materials/Coefficients Library** dialog box, see "Using the Materials/Coefficients Library" on page 223 in the *COMSOL Multiphysics User's Guide*.

Piezoelectric Material Properties Library

The Piezoelectric Material Properties library ships with the Acoustics Module, MEMS Module, and Structural Mechanics Module. It contains the following piezoelectric materials:

MATERIAL
Barium Sodium Niobate
Barium Titanate
Barium Titanate (poled)
Lithium Niobate
Lithium Tantalate
Lead Zirconate Titanate (PZT-2)
Lead Zirconate Titanate (PZT-4)
Lead Zirconate Titanate (PZT-4D)
Lead Zirconate Titanate (PZT-5A)
Lead Zirconate Titanate (PZT-5H)
Lead Zirconate Titanate (PZT-5J)

MATERIAL
Lead Zirconate Titanate (PZT-7A)
Lead Zirconate Titanate (PZT-8)
Quartz
Rochelle Salt
Bismuth Germanate
Cadmium Sulfide
Gallium Arsenide
Tellurium Dioxide
Zinc Oxide
Zinc Sulfide
Ammonium Dihydrogen Phosphate
Aluminum Nitride

All materials define the following material properties needed for piezoelectric modeling:

MATERIAL PROPERTY	DESCRIPTION
c_E	Elasticity matrix
e	Coupling matrix, stress-charge
ε_{rS}	Relative permittivity, stress-charge
s_E	Compliance matrix
d	Coupling matrix, strain-charge
ε_{rT}	Relative permittivity, strain-charge
ρ	Density

MEMS Material Properties Library

The MEMS Material Properties library ships with the Acoustics Module, MEMS Module, and Structural Mechanics Module. It contains 33 materials commonly used in MEMS applications. The materials are divided into the following groups: Metals, Semiconductors, Insulators, and Polymers.

The basic structure of this library comes from the book *Microsensors, MEMS, and Smart Devices* (Ref. 3). The material properties come from two primary sources: the *CRC Handbook of Chemistry and Physics* (Ref. 1) and *MacMillan's Chemical and Physical Data* (Ref. 2). Some of the mechanical properties in the library are instead

more MEMS-specific values from *The MEMS Handbook* (Ref. 4), and most of the semiconductor properties are values from Ref. 5. Ref. 6 provides a valuable resource for cross-checking the insulation material properties.

The table below lists the materials and their corresponding groups:

MATERIAL	GROUP
Aluminium (Al)	Metals
Silver (Ag)	Metals
Gold (Au)	Metals
Chrome (Cr)	Metals
Indium (In)	Metals
Titanium (Ti)	Metals
Iron (Fe)	Metals
Nickel (Ni)	Metals
Lead (Pb)	Metals
Palladium (Pd)	Metals
Platine (Pt)	Metals
Antimon (Sb)	Metals
Tungsten (W)	Metals
C [100]	Semiconductors
GaAs	Semiconductors
Ge	Semiconductors
InSb	Semiconductors
Si(c)	Semiconductors
Poly-Si	Semiconductors
Silicon (single-crystal)	Semiconductors
Al2O3	Insulators
SiC (6H)	Insulators
Si3N4	Insulators
SiO2	Insulators
ZnO	Insulators
Borosilicate	Insulators
Nylon	Polymers
PMMA	Polymers

MATERIAL	GROUP
Polymide	Polymers
Polyethylene	Polymers
PTFE	Polymers
PVC	Polymers

REFERENCES

1. D.R. Lide (Editor-in-chief), *CRC Handbook of Chemistry and Physics*, 84th edition, CRC Press, 2003.

2. A.M. James and M.P. Lord, *MacMillan's Chemical and Physical Data*, MacMillan's Press, 1992.

3. J.W. Gardner, V.K. Varadan, and O.O. Awadelkarim, *Microsensors, MEMS, and Smart Devices*, John Wiley & Sons, 2001.

4. M. Gad-el-Hak (editor), *The MEMS Handbook*, CRC Press, 2002.

5. *New Semiconductor Materials. Characteristics and Properties*, http://www.ioffe.ru/SVA/NSM, 2003.

6. *Ceramics WebBook*, http://www.ceramics.nist.gov/srd/scd/scdquery.htm, 2003.

4

Application Mode Overview

The Acoustics Module provides a set of application modes that facilitate the modeling of acoustics phenomena in a wide variety of application contexts. This chapter gives an overview of the application modes you have at your disposal when creating models using the Acoustics Module.

Application Mode Guide

Table 4-1 lists the application modes available in the Acoustics Module. For a detailed description of any of the modes, see the section on the page listed in the corresponding table entry.

In the table, the application modes are classified in five categories:

- *Acoustics*—the module's core application modes which model the sound waves
- *Structural Mechanics*—application modes which allow you to model acoustic-structure interaction
- *Piezoelectricity*—application modes for modeling of piezoelectric effects.
- *Fluid Dynamics*—an application mode tailored for modeling irrotational flow
- *Predefined Multiphysics*—combinations of application modes with predefined couplings

The **Name** column shows the default name that appears as a label when you use an application mode. The labels are particularly important in multiphysics simulations where they distinguish between the application modes included in the model; each model defines its own set of variables, whose names are appended by an underscore and the unique application mode name.

The **Dependent Variables** column lists the dependent variables for which the mode formulates the underlying PDEs. Note that using weak constraints introduces additional dependent variables on boundaries.

Finally, the **Analysis Capabilities** columns indicate which analysis types each application mode supports.

Note: For historical reasons, the frequency domain analysis type is called *time-harmonic* in the acoustics application modes but *frequency response* in the structural mechanics context. These analysis types are however fully compatible.

TABLE 4-1: ACOUSTICS MODULE APPLICATION MODES

APPLICATION MODE	DEFAULT NAME	PAGE	DEPENDENT VARIABLES	ANALYSIS CAPABILITIES						
				STATIC	TIME-HARMONIC	TIME-HARMONIC, SCATTERED WAVE	TRANSIENT	EIGENFREQUENCY/-MODE	DAMPED EIGENFREQUENCY/-MODE	FREQUENCY RESPONSE
ACOUSTICS										
Pressure Acoustics	acpr	64	p		√	√	√			
Pressure Acoustics, Boundary modal analysis	acbm	99	p					√		
Aeroacoustics	acae	112	ϕ		√		√	√		
Aeroacoustics, Boundary modal analysis	acab	124	ϕ					√		
STRUCTURAL MECHANICS										
Solid, Stress-Strain	acsld		u, v, w	√			√	√	√	√
Axial Symmetry, Stress-Strain	acaxi		uor, w	√			√	√	√	√
Plane Strain	acpn		u, v	√			√	√	√	√
PIEZOELECTRICITY										
Piezo Solid	smpz3d		u, v, w, V	√			√	√	√	√
Piezo Axial Symmetry	smpaxi		uor, w, V	√			√	√	√	√
Piezo Plane Strain	smppn		u, v, V	√			√	√	√	√
FLUID DYNAMICS										
Compressible Potential Flow	acpf	130	Φ, ρ	√			√			

TABLE 4-1: ACOUSTICS MODULE APPLICATION MODES

APPLICATION MODE	DEFAULT NAME	PAGE	DEPENDENT VARIABLES	ANALYSIS CAPABILITIES						
PREDEFINED MULTIPHYSICS				STATIC	TIME-HARMONIC	TIME-HARMONIC, SCATTERED WAVE	TRANSIENT	EIGENFREQUENCY/-MODE	DAMPED EIGENFREQUENCY/-MODE	FREQUENCY RESPONSE
Aeroacoustics with Flow	acpf, acae	135	Φ, ρ, ϕ		√		√			

In addition, you can perform parametric analyses using the parametric solver. Typical parameters to vary include the frequency and the out-of-plane or axial wave number.

When using the axisymmetric modes it is important to note that the horizontal axis represents the *r* direction and the vertical axis the *z* direction. Further, you must create the geometry in the right half plane, that is, only for positive *r*.

Analysis Capabilities

The Acoustics Module is primarily designed for frequency-domain simulations, including related eigenvalue and modal problems. Transient analysis is possible but less efficient from the computational point of view. The Compressible Potential Flow application mode is tailored to model a stationary background flow to be used in a subsequent time-harmonic aeroacoustics simulation. In the solid structural mechanics application modes, the static analysis type is included merely as a convenience.

The analysis types require different solvers and equations. In the **Application Mode Properties** dialog box you select one of the analysis types, each of which has a predefined solver. You can disable the choice of a predefined solver by clearing the **Auto**

select solver check box in the **Solver Parameters** dialog box. The following table lists the different analysis types with their predefined solver:

ANALYSIS TYPE	DEFAULT SOLVER	APPLICATION MODE TYPES
Static	Stationary	Structural mechanics, Piezoelectric, Fluid dynamics
Time-harmonic	Stationary	Acoustics
Frequency response	Parametric	Structural mechanics, Piezoelectric,
Eigenfrequency	Eigenvalue	Acoustics, Structural mechanics, Piezoelectric
Damped eigenfrequency	Eigenvalue	Structural mechanics, Piezoelectric
Modal analysis	Eigenvalue	Acoustics
Transient	Time dependent	All

Note that the acoustics' application modes time-harmonic analysis type and the structural mechanics frequency response analysis are fully compatible. The only difference between them is the default solver choice. Both analysis types work well with either the stationary or parametric solver. To manually change to a different solver, make a new selection in the **Solver Parameters** dialog box.

Read through the following analysis type descriptions to help find good candidates for your application.

STATIC ANALYSIS

A static analysis solves for stationary displacements or a steady-state condition. All loads and constraints are constant.

TIME-HARMONIC AND FREQUENCY-RESPONSE ANALYSIS

Acoustic wave propagation is modeled by equations from linearized fluid dynamics and solid dynamics. The full equations are time dependent, but noting that a harmonic excitation with a time dependence of the form $f = \hat{f}e^{i\omega t}$ gives rise to an equally harmonic response with the same frequency, the time can be eliminated completely from the equations. Instead the angular frequency, ω, enters as a parameter.

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.

Frequency domain simulation suites the finite element method, on which COMSOL Multiphysics and the Acoustics Module are based, very well. Therefore, choose the time-harmonic or frequency response analysis types over the transient whenever possible. Certain important software features, notably PMLs and damping due to porous media, are only present when using the time-harmonic and frequency response analysis types.

The result of a frequency response analysis is a complex time-dependent field, which can be interpreted as an amplitude u_{amp} and a phase angle u_{phase}. The actual displacement at any point in time is the real part of the solution:

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

You can visualize the amplitudes and phases as well as the solution at a specific angle (time). The **Solution at angle** 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** edit 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 `freq` and `omega` are available variables.

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

EIGENFREQUENCY ANALYSIS

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 frequencies* of the structure. When approaching a resonance frequency in a harmonically-driven problem, you need a weaker and weaker source 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 analysis type when you are interested in the resonance frequencies of the structure, whether you want to exploit them, as in a musical instrument, or avoid them, as in a reactive muffler or inside a hi-fi speaker system. To the trained 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, you can specify whether to look at the mathematically more fundamental eigenvalue, λ, or the eigenfrequency, f, which is more commonly used in an acoustics context.

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

You control the way to specify eigenvalues from the **Application Mode Properties** dialog box from the **Physics** menu.

DAMPED EIGENFREQUENCY ANALYSIS

A damped eigenfrequency analysis solves for the damped eigenfrequencies and the shape of the eigenmodes in structural and piezoelectric models. When performing a damped eigenfrequency analysis, you can specify whether to look at the mathematically more fundamental eigenvalue, λ, or the eigenfrequency, f, which is more commonly used in a structural-mechanics context.

MODAL ANALYSIS

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 direction, or more easily by inserting the assumption that the mode is harmonic in space, $u = ue^{-ik_z z}$ and eliminating all z dependence. The axial wave number, k_z, is a parameter in the 2D acoustics application modes.

Similarly 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 of propagating or evanescent waveguide modes. The eigenvalue solver can solve for these propagation constants together with the corresponding mode shapes. Note that the propagation constant is a function of the frequency. The relation between the two is commonly referred to as a *dispersion curve*.

Modal analysis is available as an analysis type in plane 2D and axially symmetric 1D acoustics application modes. There are also special boundary modal application modes available on boundaries in 3D and 2D axisymmetry.

The most common use for the modal analysis is in defining sources for a subsequent time-harmonic simulation. If you have a component with one or more waveguide connections, you can describe its behavior by simulating its response to the discrete set of propagating modes on the waveguide port cross sections. The frequency-dependent transfer matrix thus obtained can be used in an external simulation, using COMSOL Script for instance, of a complete waveguide system. See, for example, "Flow Duct" on page 101 in the *Acoustics Module Model Library*.

TRANSIENT ANALYSIS

The complete equations behind the theory of acoustic wave propagation are time dependent, as noted above. 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, however, if not for efficiency so for convenience.

Note that some central modeling techniques, such as the use of PMLs, are not available for the transient analysis type. Further, you have to be careful when defining your sources to avoid, as far as possible, to excite waves at frequencies that the mesh cannot resolve. For an example, see "Transient Gaussian Explosion" on page 55 in the *Acoustics Module Model Library*.

Application Mode Documentation Notes

The following chapters contain the details necessary to get full insight into the different application modes, that is, the physical assumptions and mathematical considerations upon which we base them and the functionality they offer.

In the *Application Mode Variables* sections you find all the variables available for formulating equations and for postprocessing (when you define a function of these

variables for plots and graphs). In a multiphysics setting, it is also possible to use these variables in expressions for physical properties in the other application modes.

We have organized the tables that detail the application mode variables as follows:

- The **Name** column gives the names of variables you can use in equations or for postprocessing. The indices `i` and `j` (using an italic font) in the variable names can mean any of the spatial coordinates. For example, `vi` means `vx`, `vy`, or `vz` in 3D when the spatial coordinates are x, y, and z. In 2D axisymmetry, `pi` stands for `vr` or `vz`.

  In a COMSOL Multiphysics model, the name of each application mode variable gets an underscore plus the application mode name appended as a suffix. For example, the default name of the Pressure Acoustics application mode is acpr, so the variable for the x-component of the local velocity is called `vx_acpr`.
- The **Domain** column indicates whether the variable is defined on subdomains (S), boundaries (B), edges (E), or points (P). The column indicates the top level where the variable is defined. Many variables that are available on subdomains are also available on boundaries, edges, and points, but they then take the average value of the values in the subdomains around the boundary, edge, or point. In other words, to get a value on an interior boundary between two subdomains, you take the average of the value in the left subdomain and the value in the right subdomain.
- The **Analysis** column specifies for which type of analysis a variable is defined. The available analysis types might be, for example, transient (T), time-harmonic (H), eigenfrequency (E), or modal (M). The available analysis types are application-mode dependent; some variables are defined differently depending on the analysis type or are available only for some analysis types.
- The **Description** column gives a textual description of the variables.
- The **Expression** column gives the expression of the variables in terms of other physical quantities. In these expressions, the subscripts i and j of vector and tensor components stand for one of the spatial coordinates, usually referring to a corresponding index in the **Name** column.

5

Pressure Acoustics

This chapter describes the two application modes in the Acoustics Module in which the acoustic pressure, p, is the basic dependent variable:

- Pressure Acoustics (acpr)
- Pressure Acoustics, Boundary modal analysis (acbm)

Because the Modal analysis type of the Pressure Acoustics application mode is intimately related to the Pressure Acoustics, Boundary modal analysis application mode, these are described in a separate section following the discussion of the other analysis types of the Pressure Acoustics application mode. The chapter begins, however, with a brief review of the underlying mathematics.

Theory Background

Mathematical Model

Sound waves in a lossless medium are governed by the following equation for the acoustic pressure, p (with SI unit Pa):

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

Here ρ_0 (kg/m^3) refers to the density, and c_s (m/s) denotes the speed of sound. The *dipole source* $\mathbf{q}$ (N/m^3) and the *monopole source* Q (1/s^2) are both optional. The combination $\rho_0 c_s^2$ is called the *adiabatic bulk modulus*, commonly denoted K (Pa).

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, with f (Hz) 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:

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

You can alternatively treat this equation as an eigenvalue PDE to solve for eigenmodes and eigenfrequencies.

Typical boundary conditions are:

- Sound-hard boundaries (walls)
- Sound-soft boundaries (zero acoustic pressure)
- Specified acoustic pressure
- Specified normal acceleration
- Impedance boundary conditions
- Radiation boundary conditions

Their respective purposes in pressure acoustics modeling and their implementations in the Pressure Acoustics application mode are described in more detail in the next section, "Boundary Conditions" on page 76.

In lossy media, it is necessary to introduce an additional term of first order in the time derivative to account for attenuation of the sound waves:

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

For transient analysis, the damping term in Equation 5-1 is absent from the standard PDE formulations in the Pressure Acoustics application modes. However, in line with COMSOL Multiphysics' general modeling philosophy, you can access the d_a coefficient from the user interface through the **Subdomain Settings - Equation System** dialog box.

In the frequency domain, the 1st-order damping term gives a purely imaginary contribution to the Helmholtz equation that can be interpreted as a complex speed of sound. A selection of more general models for damping by means of complex material parameters is available for time-harmonic and eigenfrequency analysis; the details are described in the next section, "Subdomain Settings" on page 70.

Finally, note that 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.

Application Mode Description

Variables and Space Dimensions

The Pressure Acoustics application modes solve for the acoustic pressure, p. It is suitable for modeling acoustics phenomena that do not involve fluid flow.

The Pressure Acoustics application mode is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

PDE Formulation

The Pressure Acoustics application mode provides four distinct PDE formulations—or analysis types:

- Transient analysis
- Time-harmonic analysis
- Eigenfrequency analysis
- Modal analysis

This section discusses the former three options, while the Modal analysis type is described a separate section "Pressure Acoustics, Modal Analysis" on page 99.

TRANSIENT ANALYSIS

Use the Transient analysis type to model transient acoustic phenomena in a stationary fluid. With this choice the software solves the wave equation

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

for the acoustic pressure, $p = p(\mathbf{r}, t)$. Here c_s is the speed of sound and ρ_0 denotes the equilibrium density, while $\mathbf{q}$ and Q are dipole and monopole sources, respectively. The density and speed of sound can both be nonconstant in space. In contrast, they are assumed to vary with time on scales much larger than the period for the acoustic waves, and they are therefore considered time independent in the previous equation. For information about how to specify these properties, see "Subdomain Settings" on page 70.

In the **Model Navigator** or in the **Application Mode Properties** dialog box (see "Application Mode Properties" on page 68) you can select between solving for the total wave or for the scattered wave. The default is to solve for the total wave.

In the scattered-field formulation, the total acoustic pressure, p_t, is written as the sum of a known incident field, p_i, and an unknown scattered field, p_s. Inserting this sum in the standard acoustic wave equation and assuming that the incident field by itself is a solution to the source-free equation, it follows that p_s—which is what p refers to in this formulation—satisfies the source-free version of Equation 5-2. You can set the expression for the incident wave, p_i, in the **Application Scalar Variables** dialog box. The default expression is a Gaussian pulse of width $\Delta t = 0.01$ s traveling in the x direction:

$$p_i(\mathbf{r}, t) = \exp\left(-\frac{1}{2}\frac{(t - x/c_s)^2}{(\Delta t)^2}\right).$$

TIME-HARMONIC ANALYSIS

In the frequency domain, Equation 5-2 corresponds to the inhomogeneous Helmholtz equation

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{\omega^2}{\rho_0 c_s^2} p = Q \tag{5-3}$$

where $p = p(\mathbf{x}, \omega)$ (the dependence on ω is henceforth not explicitly indicated). With this formulation you can compute the frequency response by using the parametric solver to sweep over a frequency range using a harmonic load. You specify ω through the frequency, f, which is an application scalar variable; see page 69.

When there is damping, ρ_0 and c_s are complex quantities. The available damping models and how to apply them is described in the section "Subdomain Settings" on page 70.

In the **Model Navigator** or in the **Application Mode Properties** dialog box you can choose between solving for the total wave or for the scattered wave. The default setting is to solve for the total wave.

In the scattered-field formulation the default expression for the incident wave is a plane wave traveling in the x direction:

$$p_i(\mathbf{r}) = e^{-ikx}$$

The wave number is $k = \omega/c_s$, where there angular frequency, $\omega = 2\pi f$, is defined in terms of the frequency, f.

Equation 5-3 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 $p(\mathbf{r}) = p(x,y)e^{-ik_z z}$, which inserted in Equation 5-3 gives

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{1}{\rho_0}\left(\frac{\omega^2}{c_s^2} - k_z^2\right)p = Q \tag{5-4}$$

The *out-of-plane wave number*, k_z, is an application scalar variable that you can supply as a parameter. By default its value is zero. In the Modal analysis type, $-ik_z$ is used as the eigenvalue; see the section "Pressure Acoustics, Modal Analysis" on page 99.

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,

$$p(r,\phi,z) = p(r,z)e^{-im\varphi} \tag{5-5}$$

where m denotes the *circumferential wave number*. Because the azimuthal coordinate is periodic, m must be an integer. Just like k_z in the 2D case, it is an application scalar variable.

As a result of Equation 5-5, the equation to solve for the acoustic pressure in 2D axisymmetric geometries becomes

$$\frac{\partial}{\partial r}\left[-\frac{r}{\rho_0}\left(\frac{\partial p}{\partial r} - q_r\right)\right] + r\frac{\partial}{\partial z}\left[-\frac{1}{\rho_0}\left(\frac{\partial p}{\partial z} - q_z\right)\right] - \left[\left(\frac{\omega}{c_s}\right)^2 - \left(\frac{m}{r}\right)^2\right]\frac{rp}{\rho_0} = rQ\,.$$

1D Axisymmetry

In 1D axisymmetric geometries, $p(r,\phi,z) = p(r)e^{-i(k_z z + m\varphi)}$, leading to the radial equation

$$\frac{\partial}{\partial r}\left[-\frac{r}{\rho_0}\left(\frac{\partial p}{\partial r} - q_r\right)\right] - \left[\left(\frac{\omega}{c_s}\right)^2 - \left(\frac{m}{r}\right)^2 - k_z^2\right]\frac{rp}{\rho_0} = rQ$$

where both the circumferential wave number, m, and the *axial wave number*, k_z appear as parameters.

1D

The equation for the 1D case is obtained by taking the pressure to depend on a single Cartesian coordinate, x:

$$\frac{d}{dx}\left(-\frac{1}{\rho_0}\left(\frac{dp}{dx}-q\right)\right)-\frac{\omega^2}{\rho_0 c_s^2}p = Q.$$

EIGENFREQUENCY ANALYSIS

You can treat Equation 5-3 as an eigenvalue problem. In the eigenvalue formulation you solve for the eigenmodes and the eigenvalues or eigenfrequencies:

$$\nabla\cdot\left(-\frac{1}{\rho_0}\nabla p-\mathbf{q}\right)+\frac{\lambda^2 p}{\rho_0 c_s^2} = Q. \tag{5-6}$$

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 dipole and monopole source terms are ignored by the solver unless you are solving a coupled eigenvalue problem.

Equation 5-6 applies to the 3D case. The equations solved in eigenfrequency analyses in lower dimensions and for axisymmetric geometries are obtained from their time-harmonic counterparts, given in the previous subsection, by the substitution $\omega^2 \rightarrow -\lambda^2$.

You can switch between specifying the eigenvalues, the eigenfrequencies, or the angular frequencies by choosing **Properties** from the **Physics** menu and changing the value of the property **Specify eigenvalues using** in the **Application Mode Properties** dialog box. There you can also change the analysis type.

Application Mode Properties

Table 5-1 lists the properties you can set in the **Application Mode Properties** dialog box, which you reach from the **Model Navigator** when selecting or adding application modes or by choosing **Physics>Properties** during a modeling session.

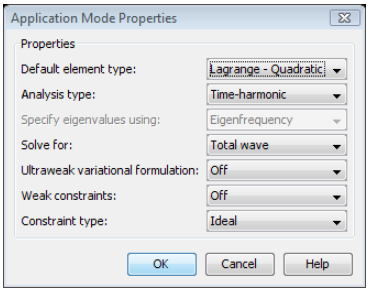


TABLE 5-1: PRESSURE ACOUSTICS APPLICATION MODE PROPERTIES

PROPERTY	VALUES	DESCRIPTION
Default element type	Lagrange elements of order 1–5 (the default value is 2). If Ultraweak variational formulation is On, Ultraweak Helmholtz elements.	Specifies which type of finite elements to use.
Analysis type	Time-harmonic | Eigenfrequency | Transient	Specifies which type of analysis to perform.
Specify eigenvalues using	Eigenfrequency | Eigenvalue | Angular frequency	Specifies the quantity in which solver parameters and output should be given; applies only to eigenfrequency analysis.
Solve for	Total wave | Scattered wave	Specifies if the dependent variable, p, describes the total wave or the scattered wave; applies only to time-harmonic and transient analysis.
Ultraweak variational formulation	On | Off	Specifies if the Ultraweak variational formulation should be used.
Weak constraints	On | Off	Specifies if weak constraints should be used.
Constraint type	Ideal | Non-ideal	Specifies the type of constraint.

For 2D Cartesian and 1D axisymmetric geometries, the Modal analysis type is also available from the **Application Mode Properties** dialog box; for a corresponding list of it application mode properties see Table 5-7 on page 101.

Some of these entries have already been brought up in this discussion. For a review of element types and weak constraints refer to the sections "Understanding the Finite Element Method" on page 452 of the *COMSOL Multiphysics Reference Guide* and "Using Weak Constraints" on page 300 in the *COMSOL Multiphysics Modeling Guide*.

Application Scalar Variables

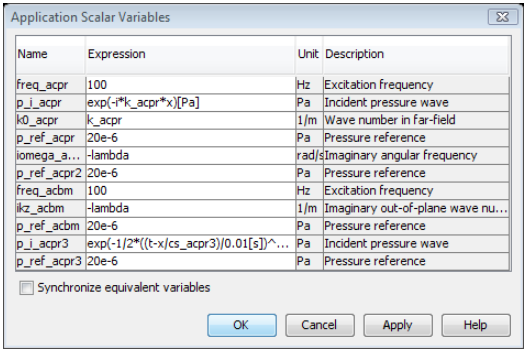
Table 5-2 lists the predefined scalar variables in the Pressure Acoustics application mode. The following abbreviations are used to label the analysis types discussed in this section:

T	Transient
H	Time-harmonic
E	Eigenfrequency

TABLE 5-2: PRESSURE ACOUSTICS APPLICATION SCALAR VARIABLES

QUANTITY	VARIABLE	ANALYSIS	GEOMETRIES	DESCRIPTION
f	freq	H	All	Frequency
$i\omega$	iomega	E	All	Imaginary angular frequency
ik_z	ikz		2D	Imaginary out-of-plane wave number
ik_z	ikz		1D axi	Imaginary axial wave number
m	m		2D axi, 1D axi	Circumferential wave number
p_i	p_i	H T		Incident pressure wave
p_ref	p_ref	All		Pressure reference for the sound pressure level

To inspect a list of all the application scalar variables defined in the model you are working with, open the **Application Scalar Variables** dialog box by choosing **Physics>Scalar Variables**.



Note that each variable name is appended by a tag that identifies the application mode to which it belongs. For the Pressure Acoustics application mode the default application mode name is `acpr`. If there are two or more application modes of the same type in a model, they are by default distinguished by a number after the application mode name.

You can modify the value of any variables in the corresponding **Expression** edit field. By first selecting the **Synchronize equivalent variables** check box you ensure that a change in, for instance, the value of the frequency for one application mode automatically propagates to the frequencies for all other application modes.

Subdomain Settings

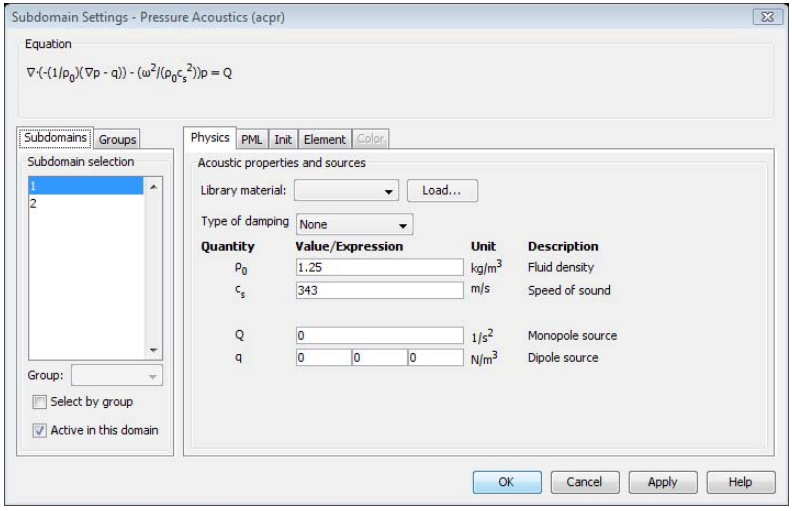
The **Subdomain Settings** dialog box contains the following pages, each accessible by clicking the corresponding tab:

- **Physics**: This is where you specify acoustic properties and sources defined at the subdomain level.
- **PML**: In time-harmonic and eigenfrequency analyses you can include auxiliary subdomains in the geometry that serve as perfectly matched layers; use this page to designate the selected subdomains as PMLs and to specify their properties in the manner described in the subsection “Perfectly Matched Layers (PMLs)” on page 73.
- **Init**: On this page you can set the initial value, $p(t_0)$, for the acoustic pressure in transient analyses.

- **Elements**: Here you can change the specifications for the finite elements to be used in the subdomain; for further details refer to the subsection "Specifying the Finite Element Type" on page 211 of the *COMSOL Multiphysics User's Guide*.
- **Color**: This page is active only when the **Groups** page is open. Use it to visually distinguish among groups of subdomains with common settings by giving them different colors in the user interface.

PHYSICS

On the **Physics** page you can specify the equilibrium density, ρ_0, and the speed of sound, c_s. The default values are appropriate for air at atmospheric pressure and room temperature. This is also where you specify dipole and monopole sources, if present.



Damping

In all analysis types except transient analysis you have the option of including damping in a model. The fluid density and the speed of sound in Equation 5-3 are then complex quantities, denoted by ρ_c and c_c, respectively:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p - \mathbf{q})\right) - \frac{\omega^2}{\rho_c c_c^2} p = Q\,.$$

These variables are frequency dependent, defined by the equations

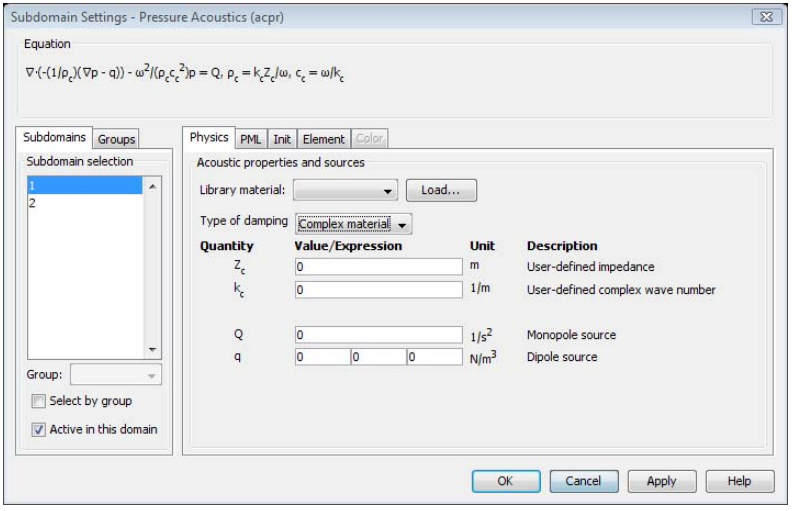
$$\rho_c = \frac{Z_c k_c}{\omega} \qquad c_c = \frac{\omega}{k_c}$$

where the expressions for the complex wave number, k_c, and the complex impedance, Z_c, depend on the selected type of damping. To turn on the damping select the type of damping you want in the **Type of damping** list in the **Subdomain Settings** dialog box. The following options for damping are available:

- **None**: No damping, that is, $\rho_c = \rho_0$ and $c_c = c_s$. This is the default option.
- **General damping**: This type of damping is specified by means of the real-valued attenuation coefficient, α (1/m):

$$k_c = \frac{\omega}{c_s} - i\alpha \qquad Z_c = \rho_0 c_s .$$

  The attenuation coefficient is not directly related to the fluid's physical properties. Instead it represents a guessed or measured relative attenuation per unit distance for a propagating wave without making any assumptions as to which physical process is responsible for the attenuation.
- **Complex material**: This is the type of damping to choose if you want to enter your own expressions for the complex impedance and the complex wave number in a porous medium. Given a sample of the material, you can measure these quantities as functions of frequency using an impedance tube. For details on how to use such data to define interpolation functions, see the section "Interpolation of Measured Data and Nonlinear Materials" on page 229 of the *COMSOL Multiphysics User's Guide*.



- **Delany-Bazley**: This option specifies damping for porous media according to the model of Delany and Bazley:

$$k_c = \frac{\omega}{c_s}\left[1 + C_1\left(\frac{\rho_0 f}{R_f}\right)^{-C_2} - iC_3\left(\frac{\rho_0 f}{R_f}\right)^{-C_4}\right]$$

$$Z_c = \rho_0 c_s\left[1 + C_5\left(\frac{\rho_0 f}{R_f}\right)^{-C_6} - iC_7\left(\frac{\rho_0 f}{R_f}\right)^{-C_8}\right]$$

  The advantage of this scheme is that the flow resistivity R_f is easy to measure and is independent of frequency. Other authors have refined the Delany-Bazley model for particular materials by changing the C_i coefficients. You can modify these values to suit individual needs using the **Subdomain Settings - Equation System** dialog box. For an example model involving Delany-Bazley damping see "Absorptive Muffler" on page 74 of the *Acoustics Module Model Library*.

- **Bulk viscosity**: Here you specify damping using a bulk viscosity, μ_B, with SI unit Pa·s/m, according to the equations:

$$k_c = \frac{\omega}{c_s}\frac{1}{\sqrt{1 + \dfrac{i\omega\mu_B}{\rho_0 c_s^2}}} \qquad Z_c = \rho_0 c_s \frac{1}{\sqrt{1 + \dfrac{i\omega\mu_B}{\rho_0 c_s^2}}}.$$

  This choice is most appropriate for situations where the damping takes place in free space and is not related to interaction between the fluid and a solid skeleton.

PERFECTLY MATCHED LAYERS (PMLS)

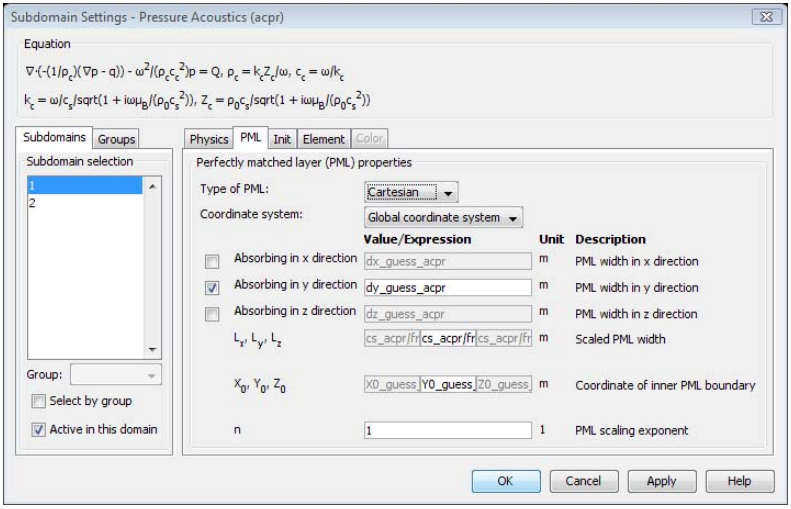
As described in more detail in the subsection "Boundary Conditions" on page 76, the Pressure Acoustics application mode offers two closely related types of absorbing boundary conditions: the radiation boundary conditions and the matched boundary condition. The former are perfectly absorbing for plane, cylindrical, and spherical waves, whereas the latter is perfectly absorbing only for guided modes provided that the correct value of the propagation constants are supplied.

However, in situations where you cannot describe the outgoing radiation as a simple waveform with a well-known wave number and direction of propagation, *perfectly matched layers* (PMLs) provide a powerful alternative. While a PML serves the same purpose, it is not a boundary condition but an additional domain that absorbs incident radiation without producing reflections. PMLs provide good performance for a wide range of incidence angles and are not very sensitive to the shape of the wave fronts. PMLs are thus capable of emulating nonreflecting boundaries.

The following types of PMLs are available:

TYPE	DESCRIPTION
None	Not absorbing
Cartesian	Absorbing in the specified Cartesian coordinate directions
Cylindrical	Absorbing in the radial or axial direction from a specified axis (the z-axis for axisymmetric geometries)
Spherical	Absorbing in the radial direction from a specified point
User defined	User-defined PML coordinates

To implement a PML, begin by adding a subdomain on the outside of the boundary where the waves radiate out into the surroundings. Then specify the PML properties on the **PML** page of the **Subdomain Settings** dialog box, starting by selecting the **Type of PML**. For the PML type **Cartesian** you have the option of selecting which coordinate system to use. Unless you have defined an alternative coordinate system in the **Coordinate System Settings** dialog box (that you open from the **Options** menu), the only available option is the default **Global coordinate system**.



In the example of the previous figure, you specify a PML of type **Cartesian** in a 3D Cartesian geometry with the following properties:

- The PML subdomain, which you select from the **Subdomain selection** list or by clicking directly in the drawing area of the user interface.

- The coordinate directions in which waves should be absorbed. In this example, the PML absorbs waves traveling in the y direction.
- The *PML width*, d_i, in each absorbing coordinate direction, x_i. COMSOL Multiphysics attempts to extract a value for the PML width and assigns it to a variable referred to as a "guess variable." You can inspect this value on the **Variables** page in the **Subdomain Settings - Equation System** dialog box or at the corresponding node of the **Model Tree**. The default PML width in a given direction is the actual width of the PML subdomain in the direction in question. You can replace this variable by explicitly entering the actual value, but the predefined variable works well in most cases.
- The *scaled PML width*, L_i, in each absorbing coordinate direction; choose this parameter so that any unwanted reflections are suppressed to below the general numerical noise level in the model. Provided that the mesh resolution is sufficient, the attenuation of reflections measured in dB is proportional to the scaled PML width. By default the scaled width is set to one wavelength, which is `cs_acpr/freq_acpr` using the default application mode name. In theory, the attenuation of a plane wave entering a rectilinear PML in the normal direction is then approximately 109 dB. In practice this level of attenuation is neither necessary nor obtained.

  For obliquely incident waves, you need to adjust the scaled PML widths to preserve the attenuation level. If the angle between the absorbing coordinate direction ξ_i and the incident wave's direction of propagation is ϑ_i, multiply the default value of L_i by the factor $|\cos\vartheta_i|^{-1}$. Conversely, if you keep the default L_i value, the attenuation level decreases by the factor $|\cos\vartheta_i|$.

  If you increase the scaled PML width, you must increase the mesh resolution in the PML domain accordingly because the mesh must resolve the number of wavelengths that fit inside the scaled width. Conversely, if you do not need the high level of attenuation provided by the default setting, you can save mesh elements by reducing the scaled PML width. To estimate the efficiency of a given PML, look at the difference between the sound pressure level of the incident wave and the level at the outside boundary of the PML. The attenuation of the reflected wave is roughly twice this difference.
- The *coordinates of the inner PML boundary*, x_{i0}, which COMSOL Multiphysics also provides as predefined variables, `X0_guess_acpr`, `Y0_guess_acpr`, and

`Z0_guess_acpr`. You can replace these variables with your own coordinate values, but the predefined values should work well in most cases.

- The *PML scaling exponent* defines the type of scaling to use for the coordinates in the PML. The default value is 1, which gives a linear scaling that works well in most cases. For scattering problems and models where different wavelengths should be absorbed (outside waveguides, for example) you can increase the exponent somewhat, the useful range for n roughly being between 1 and 2. Increasing the exponent allows you to use fewer mesh elements to resolve wavelengths much smaller than the scaled PML width.

Cylindrical and spherical PMLs have the following properties:

- In 3D, a cylindrical PML can be absorbing in the r direction and the z direction, and there are predefined variables for the PML width in both directions. Spherical PMLs and cylindrical PMLs in 2D always absorb in the radial direction from the center point, and that is the only PML width.
- The scaled PML width (see the previous type).
- The coordinates of the inner PML boundary (see the previous type).
- The PML scaling exponent (see the previous type).
- The center point location, (x_0, y_0, z_0) (in 3D) or (x_0, y_0) (in 2D). For cylindrical PMLs this refers to an arbitrary point on the center axis. The default location is the origin. For axisymmetric geometries the only parameter that you can specify is z_0 for spherical PMLs because the center point always lies on the z-axis.
- The center axis direction, $\mathbf{r}_{\text{axis}}$ (for cylindrical PMLs in 3D only).

For user-defined PMLs, you supply the full expressions for the PML coordinates for each coordinate direction of your model geometry. This option gives you full control over the scaling to use in the PML subdomain. The default is no scaling.

The **PML** page is available for time-harmonic and eigenfrequency analysis.

Example models in the *Acoustics Module Model Library* using PMLs include "Flow Duct" on page 101, "Loudspeaker" on page 131, "Open Pipe" on page 200, and "Scattering from a Plate with Ribs" on page 214 of this accompanying volume.

Boundary Conditions

This section describes the boundary conditions available for the Pressure Acoustics application mode. You specify them in the **Boundary Settings** dialog box.

SOUND-HARD BOUNDARY (WALL)

A *sound-hard boundary* is one where the normal component of the particle velocity vanishes. Because there is no acoustic drift velocity, this condition is equivalent to the normal acceleration being equal to zero:

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

For a zero dipole source, this means that the normal derivative of the pressure at the boundary vanishes:

$$\frac{\partial p}{\partial n} = 0 .$$

Use the sound-hard boundary condition to model rigid surfaces. Because this boundary condition is of a homogeneous Neumann type, it is also the neutral boundary condition needed on boundaries where the pressure is controlled by a periodic boundary condition (see "Using Periodic Boundary Conditions" on page 245 of the *COMSOL Multiphysics User's Guide*) or is coupled to another application mode.

SOUND-SOFT BOUNDARY

At a *sound-soft boundary* the acoustic pressure vanishes:

$$p = 0 .$$

This boundary condition is an appropriate approximation for a liquid-gas interface and in some cases for external waveguide ports.

PRESSURE SOURCE

This condition means that you specify the acoustic pressure at a boundary:

$$p = p_0 .$$

For time-harmonic analysis, p_0 is the amplitude of a harmonic pressure source. For transient analysis you must explicitly specify the time dependence of the pressure source. In both cases p_0 can be a function of the position at the boundary.

IMPEDANCE BOUNDARY CONDITION

The impedance boundary condition is a generalization of the sound-hard and sound-soft boundary conditions:

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

In the frequency domain the corresponding equation reads

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

Here Z (SI unit Pa·s/m) is the acoustic input impedance of the external domain. From a physical point of view the acoustic input impedance is the ratio between pressure and normal particle velocity. It can be expressed in terms of the characteristic impedance inside the domain, $Z_0 = \rho c$, as $Z = \zeta Z_0$, where the dimensionless quantity ζ is called the specific acoustic impedance. The default setting corresponds to $\zeta = 1$, that is, $Z = Z_0$.

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

Note that in the two opposite limits $Z \to \infty$ and $Z \to 0$ the sound-hard and sound-soft boundary conditions are obtained.

SPECIFIED NORMAL ACCELERATION

With this condition you specify the inward normal acceleration, a_n, at the boundary:

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

In this equation, a_n represents an external source term.

You can use this boundary condition to couple an acoustics model to a structural analysis. Examples of this usage in the *Acoustics Module Model Library* include "Hollow Cylinder" on page 19 and "Loudspeaker" on page 131.

AXIAL SYMMETRY

This condition is available only for axisymmetric geometries where it is the correct choice for the symmetry boundary at $r = 0$.

RADIATION BOUNDARY CONDITIONS

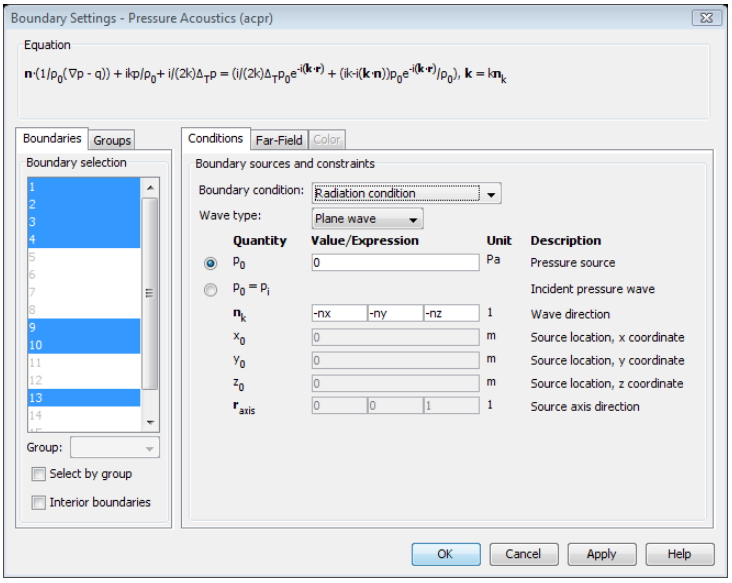
The radiation conditions allow an outgoing wave to leave the modeling domain with no or minimal reflections. In specifying a boundary condition of this kind you have the choice between three wave types:

- Plane
- Cylindrical
- Spherical

You can thus adapt the condition to the geometry of the modeling domain. In addition, you have the option of including an incoming wave. How to specify a radiation boundary condition is described in more detail in a following section.

Radiation boundary conditions are available for all analysis types. For the case of time-harmonic analysis, 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 the software uses the corresponding 2nd-order expressions from Bayliss, Gunzburger, and Turkel (Ref. 2). The Transient and Eigenfrequency analysis types implement the same expansions to the first order. Because the precise expressions for the boundary conditions thus differ between the analysis types this discussion covers them under separate headings in a following section. The procedure for specifying a radiation boundary condition is, however, essentially the same in the three cases.

The next figure shows the layout of the **Boundary Settings** dialog box when you have selected the option **Radiation condition** from the **Boundary condition** list and chosen **Plane wave** as the **Wave type**.



To include an incoming plane wave, $p_0 e^{-i\mathbf{k}\cdot\mathbf{r}}$, select the **$p_0$** option button and then supply the amplitude, p_0, in the **Pressure source** edit field. Also supply the components of the wave-direction vector, $\mathbf{n}_k$, in the three **Wave direction** edit fields. Only the direction of the vector you enter matters because the software normalizes the components to make $\mathbf{n}_k$ a unit vector. The wave vector is then defined as $\mathbf{k} = k\mathbf{n}_k$, where $k = \omega/c_s$ is the wave number.

By default $p_0 = 0$. The default value of $\mathbf{n}_k$ is the inward normal vector, $-\mathbf{n}$, which is the natural direction for waveguides and similar structures. For wave propagation in open space $\mathbf{k}$ can point in any direction.

Alternatively, you can set an incident wave equal to the application scalar variable p_i by instead clicking the **$p_0 = p_i$** option button. By modifying the expression for this variable in the **Application Scalar Variables** dialog box—the default setting is the plane wave e^{-ikx}—you can choose a form of the incident wave that suits your particular model.

In more detail, the options for the outgoing wave are:

- **Plane wave**: You can use this boundary condition 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.

  In the time-harmonic case, the plane-wave boundary condition reads

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

  or

$$\mathbf{n}\cdot\left(\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)+i\frac{k}{\rho_0}p+\frac{i}{2k}\Delta_{\mathrm{T}}p \;=\; \frac{i}{2k}\Delta_{\mathrm{T}}p_i+i\frac{k}{\rho_0}p_i+\frac{i}{2k}\Delta_{\mathrm{T}}p_i$$

  depending on how the optional incoming wave is specified. In these equations, Δ_{T} at a given point on the boundary denotes the Laplace operator in the tangent plane at that particular point.

  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**: This boundary condition is based on a series expansion of the outgoing wave in cylindrical coordinates (Ref. 2), and it assumes the field is independent of the axial coordinate. You 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.

  With an incoming plane wave $p_0 e^{-i\mathbf{k}\cdot\mathbf{r}}$ included, the boundary condition reads

$$\mathbf{n}\cdot\left(\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right) \;=\; \frac{1}{8r}\left(\frac{3+12ikr-8k^2r^2}{1+ikr}\right)\frac{(p_0 e^{-i\mathbf{k}\cdot\mathbf{r}}-p)}{\rho_0}$$
$$-i(\mathbf{k}\cdot\mathbf{n})p_0 e^{-i\mathbf{k}\cdot\mathbf{r}}-\frac{(r\Delta_{\mathrm{T}}p_0 e^{-i\mathbf{k}\cdot\mathbf{r}}+r\Delta_{\mathrm{T}}p)}{2(1+ikr)\rho_0}$$

  If you specify an incoming wave using the application scalar variable p_i, the corresponding expression becomes

$$\mathbf{n}\cdot\left(\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right) = \frac{1}{8r}\left(\frac{3+12ikr-8k^2r^2}{1+ikr}\right)\frac{(p_i-p)}{\rho_0}+\mathbf{n}\cdot\frac{1}{\rho_0}\nabla p_i-\frac{(r\Delta_{\mathrm{T}}p_i+r\Delta_{\mathrm{T}}p)}{2(1+ikr)\rho_0}$$

In both equations, for a given point on the boundary, the value of r denotes the shortest distance from the source to the point in question.

- **Spherical wave**: Use this option to allow a radiated or scattered wave—emanating from an object centered at the point (x_0, y_0, z_0) that you specify—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.

  The expression for the boundary condition reads

$$\mathbf{n}\cdot\left(\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)+\left(ik+\frac{1}{r}\right)\frac{p}{\rho_0}-\frac{r\Delta_{\mathrm{T}}p}{2\rho_0(ikr+1)} = \left(\frac{-r\Delta_{\mathrm{T}}p_0}{2\rho_0(ikr+1)}+\left(ik+\frac{1}{r}-i\,\mathbf{k}\cdot\mathbf{n}\right)\frac{p_0}{\rho_0}\right)e^{-i\,\mathbf{k}\cdot\mathbf{r}}$$

  with an incoming plane wave included, or

$$\mathbf{n}\cdot\left(\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)+\left(ik+\frac{1}{r}\right)\frac{p}{\rho_0}-\frac{r\Delta_{\mathrm{T}}p}{2\rho_0(ikr+1)} = \frac{-r\Delta_{\mathrm{T}}p_i}{2\rho_0(ikr+1)}+\left(ik+\frac{1}{r}\right)\frac{p_i}{\rho_0}+\mathbf{n}\cdot\frac{1}{\rho_0}\nabla p_i$$

  if you use the application scalar variable p_{i} to specify the incoming wave.

Eigenfrequency Analysis

For eigenfrequency analysis, the different 1st-order radiation boundary conditions can be summarized in the expression

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

where, again, $k=\omega/c_{\mathrm{s}}$ is the wave number while $\kappa(r)$ 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 latter two 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 the optional incoming plane pressure wave with amplitude p_0 and wave vector $\mathbf{k} = k\mathbf{n}_k$, where $\mathbf{n}_k$ denotes the unit vector in the direction of propagation.

In eigenfrequency analysis you specify a radiation boundary condition in much the same way as for time-harmonic analysis (see page 80) with the exception that the option of using an application mode variable to specify the incident wave is absent. In addition to the wave type you can thus specify

- p_0—the pressure source amplitude
- $\mathbf{n}_k$—the wave-direction vector
- $\mathbf{r}_0 = (x_0, y_0, z_0)$—a point on the source axis (for a cylindrical wave) or the source location (for a spherical wave)
- $\mathbf{r}_{axis}$—the source axis direction (only for cylindrical waves)

Transient Analysis

The radiation boundary condition for transient analysis is the 1st-order expression

$$\mathbf{n} \cdot \left(\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) + \frac{1}{\rho_0}\left(\frac{1}{c_s}\frac{\partial p}{\partial t} + \kappa(r)p\right) = \frac{1}{\rho_0}\left(\frac{1}{c_s}\frac{\partial p_0}{\partial t} + \kappa(r)p_0 + \mathbf{n} \cdot \nabla p_0\right)$$

where $\kappa(r)$ is the same wave-type dependent function as for the eigenfrequency case discussed on page 82. The right-hand side describes an incident wave, $p_0(\mathbf{r}, t)$, that you specify by selecting the **p_0** option button in the **Boundary Settings** dialog box and then supplying the expression in the associated edit field. Note that you can use different expressions for different boundaries.

If a single expression for the incident wave suffices for the model setup, you can instead select the **$p_0 = p_i$** option button. With this choice the right-hand side in the above equation describes an incident wave determined by the application scalar variable p_i. Its default expression is a Gaussian pulse of width $\Delta t = 0.01$ s traveling in the x direction:

$$p_i(\mathbf{r}, t) = \exp\left[-\frac{1}{2}\left(\frac{x}{c_s} - t\right)^2 / (0.01s)^2\right]$$

To change the shape of the incoming wave, choose **Scalar Variables** from the **Physics** menu and edit the expression for the variable `p_i_acpr`.

MATCHED BOUNDARY

Just as does the radiation boundary conditions, the matched boundary condition belongs to the class of nonreflecting boundary conditions (NRBCs). In fact, it is based on the same 2nd-order approximation of Givoli and Neta's NRBC scheme (Ref. 1) as the time-harmonic, plane-wave radiation condition discussed earlier.

Properly set up, the matched boundary condition allows two modes—characterized by their wave numbers, k_1 and k_2—to leave the modeling domain with minimal reflections. It is given by the equation

$$\mathbf{n}\cdot\left(\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)+\frac{i\left(\left(\frac{\omega}{c_s}\right)^2+k_1k_2\right)p+\frac{\Delta_T p}{c_s}}{\rho_0(k_1+k_2)}=$$

$$=\left(\frac{i\left(\left(\frac{\omega}{c_s}\right)^2+k_1k_2\right)p_0+\frac{\Delta_T p_0}{c_s}}{\rho_0(k_1+k_2)}-\frac{i}{\rho_0}(\mathbf{k}\cdot\mathbf{n})p_0\right)e^{-i\,\mathbf{k}\cdot\mathbf{r}}$$

Here Δ_T, for a given point on the boundary, refers to the Laplace operator in the tangential plane at that point, while p_0 is the amplitude of an optional incoming plane wave with wave vector $\mathbf{k}$. As described in more detail on page 79 in the context of radiation boundary conditions, in addition to p_0 you specify the propagation direction, $\mathbf{n}_k$, whereas the wave number is not specified in this dialog box; instead, it is defined by $k = \omega/c_s$.

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 $k_1 = \omega/c_s$ and $k_2 = \omega_1/c_s$, where $\omega_1 = 2\pi f_1$, and f_1 is the cutoff frequency for the first excited mode. With the default settings, $k_1 = k_2 = \omega/c_s$, the matched boundary condition reduces to the plane-wave option of the time-harmonic radiation boundary condition.

The matched boundary condition is available in time-harmonic and eigenfrequency analysis.

NEUTRAL

To impose periodic boundary conditions you need a homogeneous Neumann condition. When solving for the total acoustic pressure the sound-hard boundary condition is the homogeneous Neumann boundary condition. In models where you solve for the scattered pressure field, the incident pressure, $p_i \equiv p_t - p$, shows up in the

sound-hard boundary condition as the effective dipole-source term $-\nabla p_i$. For this reason, a separate neutral boundary condition is needed in such situations:

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

Here p denotes the scattered pressure. This boundary condition is available in the **Boundary condition** list when the application-mode property **Solve for** is set to **Scattered wave**.

INTERFACE CONDITIONS ON INTERIOR BOUNDARIES

By default only exterior boundaries are active in the **Boundary Settings** dialog box. However, if you select the **Interior boundaries** check box you can specify the interface conditions for boundaries inside the modeling domain that serve as partitions between subdomains.

The following options are available in the Pressure Acoustics application mode (subscripts 1 and 2 refer to the two sides of the boundary):

- **Continuity**:

$$\mathbf{n} \cdot \left[\left(-\frac{1}{\rho_0} (\nabla p - \mathbf{q}) \right)_1 - \left(-\frac{1}{\rho_0} (\nabla p - \mathbf{q}) \right)_2 \right] = 0$$

  This condition, which expresses the continuity of the normal acceleration, is the default setting. It corresponds to a situation where the interior boundary has no direct effect on the acoustic pressure field.

- **Sound soft boundary**: $p = 0$
- **Pressure**: $p = p_0$

INTERFACE CONDITIONS ON PAIRS

If you have pairs connecting different parts of an assembly, you can set boundary conditions on the pairs. To do so, click on the **Pairs** tab on the **Boundary Settings** dialog box to activate the **Pairs selection** list and the associated **Boundary condition** list.

Because pairs can form interior boundaries, the boundary conditions available for interior boundaries listed in the previous paragraph are available also for pairs. In addition, pairs can have *slit boundary conditions*, that is, boundary conditions for which the dependent variable need not be continuous across the boundary.

The following slit boundary conditions are available in the Pressure Acoustics application mode (subscripts 1 and 2 refer to the two sides of the boundary):

- **Sound hard boundary (wall)**:

$$\mathbf{n}\cdot\left(\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)_1 = 0 \qquad \mathbf{n}\cdot\left(\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)_2 = 0$$

- **Impedance boundary condition**:

  This boundary condition allows you to specify the impedance, Z, of an interior boundary. For transient analysis, the relevant conditions read:

$$\mathbf{n}\cdot\left(-\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)_1 = \frac{1}{Z}\frac{\partial}{\partial t}(p_1-p_2) \qquad \mathbf{n}\cdot\left(-\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)_2 = \frac{1}{Z}\frac{\partial}{\partial t}(p_2-p_1)$$

  In the frequency domain, the corresponding equations are:

$$\mathbf{n}\cdot\left(-\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)_1 = \frac{i\omega(p_1-p_2)}{Z} \qquad \mathbf{n}\cdot\left(-\frac{1}{\rho_0}(\nabla p-\mathbf{q})\right)_2 = \frac{i\omega(p_2-p_1)}{Z}$$

  For time-harmonic analysis, you have the option of specifying, instead of Z directly, the characteristic properties for a perforated plate. The software then calculates the impedance using the following expression (Ref. 3):

$$\frac{Z}{\rho_0 c_s} = \left(\frac{1}{\sigma}\sqrt{\frac{8\mu k}{\rho_0 c_s}}\left(1+\frac{t_p}{d_h}\right)+\theta_f\right)+i\frac{k}{\sigma}(t_p+\delta_h)$$

  Here μ is the dynamic viscosity, σ is the porosity of the perforated plate (that is, the holes' fraction of the boundary surface area—a dimensionless number between 0 and 1), t_p is the thickness of the perforated plate, and d_h is the diameter of the holes in the plate. Furthermore, δ_h is an end correction to the reactance with the default value $0.25d_h$, and θ_f is a contribution to the resistive part of the impedance that you can use, for example, to include the effects of a mean flow; by default this term is set to zero.

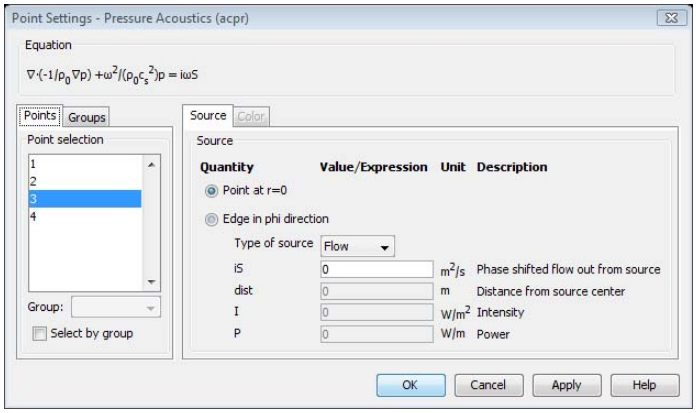
  The perforated plate model is not available for transient analysis.

  For an application of the perforated-plate impedance boundary condition, see the model "Muffler with Perforates" on page 154 of the *Acoustics Module Model Library*.

Point and Edge Conditions

Point and edge conditions are available in time-harmonic and transient analysis. Use them to include point and line sources for the acoustic pressure field.

In 3D, you specify point sources in the **Point Settings** dialog box and line sources in the **Edge Settings** dialog box. In 2D, the **Point Settings** dialog box is used to specify line sources. The 2D axisymmetric case is somewhat special in that you specify both point sources and line sources in the **Point Settings** dialog box as displayed in the next figure. For points on the symmetry axis select the **Point at r=0** option button to specify a point source. For points off the symmetry axis instead select the **Edge in phi direction** option button to specify a circular line source.



TRANSIENT ANALYSIS

Point sources are monopoles, so Equation 5-2 gives

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) + \frac{1}{\rho_0 c_s^2}\frac{\partial^2 p}{\partial t^2} = \frac{\partial S}{\partial t}\delta^{(3)}(\mathbf{r} - \mathbf{r}_0)$$

where $\mathbf{r}_0$ is the source location and S is the source strength with SI unit m^3/s. For physical line sources—that is, those defined on edges in 3D, on edges in the φ direction in 2D axisymmetry, and at points in 2D—the same equation applies with the modification that the Dirac δ function is defined in the 2D planes perpendicular to the source. In this case the unit for S is m^2/s.

When specifying a transient point source you have two options for the **Type of source**:

- **Flow**: The general option that allows you to specify the source strength, S, as an arbitrary function of time, t, by typing the relevant expression in the **S** edit field.
- **Gaussian pulse**: This option provides a convenient way to specify a Gaussian pulse; instead of typing in the expression for $S(t)$, you supply three characteristic properties:
  - The amplitude, A
  - The frequency bandwidth, f_0
  - The pulse peak time, t_p

  These properties define a pulse of the form

$$S(t) = Ae^{-\pi^2 f_0^2 (t-t_p)^2} \qquad \tau - t_\mathrm{p} < t < \tau + t_\mathrm{p}$$

  where $\tau = 1/f_0$ is the pulse half width.

TIME-HARMONIC ANALYSIS

When a point source is located at $\mathbf{r} = \mathbf{r}_0$, Equation 5-3 is modified to read

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) - \frac{\omega^2}{\rho_0 c_s^2} p = i\omega S \delta^{(3)}(\mathbf{r} - \mathbf{r}_0)$$

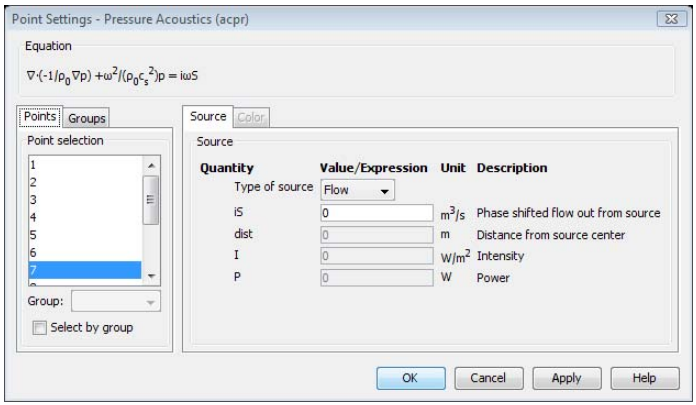
where S is the source amplitude.

In time-harmonic analysis there are three ways to specify the source strength depending on which characteristic property you have quantitative information of: flow, intensity, or power. Below follows a detailed description of each option.

Flow

If you choose the **Flow** option (the default) from the **Type of source** list in the **Point Settings** or the **Edge Settings** dialog box, the **iS** edit field is activated, allowing you to specify the source-strength amplitude. Note that the expression to enter in the edit field is i times S, not S itself. The software includes this default phase shift of the source to produce a nonzero result when you visualize the resulting pressure field

using the default value (0) in the **Solution at angle** edit field in postprocessing dialog boxes.



The flow option is the only one that allows you to specify not only the source's amplitude but also its complex phase. This can be useful if you have two or more sources that are mutually out of phase. For an interesting application involving an array of phase-shifted sources see the model "Bessel Panel" on page 8 of the *Acoustics Module Model Library*.

Intensity

Using this condition you can set a desired intensity, I, at a specified distance, dist, from the source. In a homogeneous medium you get the specified intensity, but with other objects and boundaries present the actual intensity is different.

This condition differs slightly between edges in 3D and 2D axisymmetric geometries and 2D points on the one hand and 3D and 2D axisymmetric points on the other. In the latter case the following equation holds:

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) = 4\pi(\mathrm{dist})\sqrt{\frac{Ic_s}{\rho_0}}\delta^{(3)}(\mathbf{r}-\mathbf{r}_0)$$

For an edge in a 3D or 2D axisymmetric geometry or a point in 2D the corresponding equation reads

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) = 4\pi\sqrt{\frac{fI(\mathrm{dist})}{\rho_0}}\delta^{(2)}(\mathbf{r}-\mathbf{r}_0)$$

In both cases I has the default SI unit $\mathrm{W/m^2}$.

Power

You can specify the source's strength by stating the total power it would radiate into a homogeneous medium. Again, this condition differs slightly between edges in 3D and 2D points on the one hand and 3D points on the other. For a point in 3D the following condition applies:

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

Here P is the radiated power with the default SI unit W.

For an edge in 3D or a point in 2D the relevant condition reads

$$\nabla \cdot \left(-\frac{1}{\rho_0}(\nabla p - \mathbf{q})\right) = 2\sqrt{\frac{P\omega}{\rho_0}}\delta^{(2)}(\mathbf{r} - \mathbf{r}_0)$$

where P now denotes the radiated power per unit length measured in W/m.

Far-Field Modeling

The Pressure Acoustics application mode supports far-field postprocessing for the Time-harmonic analysis type. This section describes how to use this functionality; for some background information about far-field modeling and its implementation in the Acoustics Module, see "Evaluating the Acoustic Field in the Far-Field Region" on page 30.

SPECIFYING VARIABLES FOR FAR-FIELD POSTPROCESSING

To create a variable representing the acoustic pressure field in the far-field region, follow these steps:

1. In the **Boundary Settings** dialog box, click the **Far-Field** tab. Select the boundaries over which you want the software to integrate the near field, typically a closed surface where the pressure field and its normal derivative are available.
2. Type the name of the variable in the **Name** column.
3. Press Enter or click **Apply** to update the values in the **Field** and **Normal derivative** columns. The default settings are the dependent variable for the pressure, p, and its normal derivative, $\mathbf{n} \cdot \nabla p$.

   In most cases, there is no reason to change these settings, but if you have set the pressure on any of the boundaries you can type the same value for the pressure on that boundary in the **Field** edit field. Similarly, if you use a Neumann boundary

condition at the far-field boundaries, typing the corresponding expression for $\mathbf{n} \cdot \nabla p$ into the **Normal derivative** column provides the best accuracy. For example, if you set the normal acceleration you can set the field derivative to ρa_n where ρ is the density and a_n is the normal acceleration.

4 Click the **Integral approximation at $r \to \infty$** button to compute the value in the far-field limit (see "The Far-Field Limit" on page 32) or click the **Full integral** button to compute the full Helmholtz-Kirchhoff integral (see "The Helmholtz-Kirchhoff Integral Representation" on page 31).

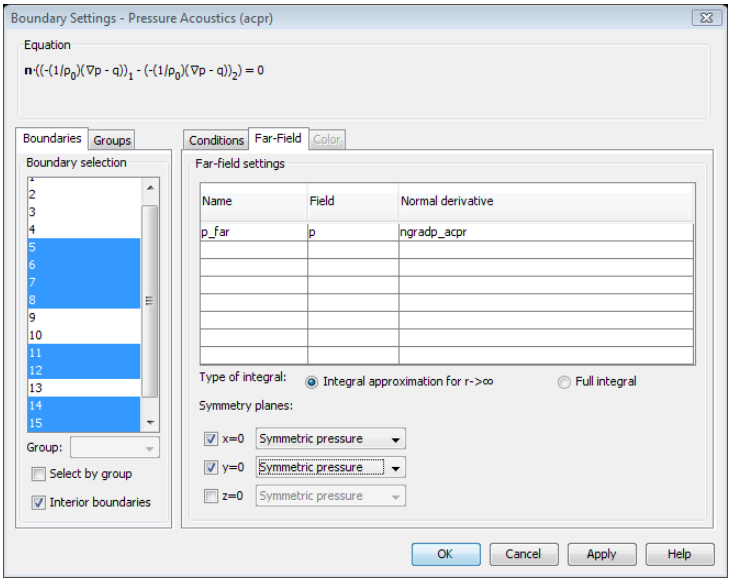
The far-field variables you define become available as predefined quantities for postprocessing on surfaces (boundaries) and edges in 3D and on boundaries in 2D. For each far-field variable the software also generates a function that represents the acoustic far field. The input to the function is the coordinates. If the far field at infinite distance is calculated, the coordinates you specify determine the direction from the source. If the full integral is calculated, the function gives the acoustic field at the point with those coordinates. For example, you can type `p_far(100,0,0)`, in the **Expression** field in the **Global Data Display** dialog box to evaluate the far-field variable `p_far` at $x = 100$ along the x-axis.

Note: The function only gives the correct answer at points farther from the field source than the boundaries where the near field is calculated.

For the sound pressure level (SPL), L_p, of a far-field variable, an additional variable is available as a predefined quantity. Its name is the same as the name for the far-field variable with the capital letter `L` as a prefix and the application mode name as a suffix. For example, the name of the variable for the sound pressure level of the far-field variable `p_far` (with the default application mode name, `acpr`) is `Lp_far_acpr`.

5 When applicable, make use of symmetries to reduce the computational effort. The symmetry planes must coincide with one of the Cartesian coordinate planes. You specify the symmetry planes by selecting those of the check boxes **x=0**, **y=0**, and **z=0** that apply. For each plane, select the type of symmetry to use—either **Symmetric pressure** or **Antisymmetric pressure**. The choice should match the boundary condition you set for the symmetry boundary. With these settings you can include regions excluded from the model for symmetry reasons in the far-field analysis.

6 Click **OK**.



The Far-Field page in the Boundary Settings dialog box for the Pressure Acoustics application mode.

MODELS USING FAR-FIELD POSTPROCESSING

Models applying the far-field postprocessing features include "Cylindrical Subwoofer" on page 14 of this book as well as "Bessel Panel" on page 8, "Hollow Cylinder" on page 19, and "Scattering from a Plate with Ribs" on page 214 of the *Acoustics Module Model Library*.

The Ultraweak Variational Formulation

By default, the Pressure Acoustics application mode uses 2nd-order Lagrange elements. For time-harmonic analysis, you can alternatively set the application mode property **Ultraweak variational formulation** to **On** to use the *ultraweak variational formulation* (UWVF). The software then switches to using Ultraweak Helmholtz elements. The basis functions for these elements are plane waves, which are solutions to the Helmholtz equation in free space. Thus, in the UWVF the finite elements contain information about the solution to the wave equation Equation 5-3. Therefore, for a given wavelength, this formulation requires a much coarser mesh, making it suitable for ultrasound modeling.

For guidelines on how to determine the mesh size when modeling with the UWVF, see the section "Using the Ultraweak Variational Formulation" on page 27. A tutorial example model, "Ultrasound Scattering Off a Cylinder," is available on page 64 in the *Acoustics Module Model Library*.

The UWVF is available in 1D, 2D, and 3D (not for axisymmetric geometries) for time-harmonic analysis only.

For detailed theoretical information on the ultraweak variational formulation, see, for example, Ref. 4 and Ref. 5.

APPLICATION MODE VARIABLE

If the application mode property **Ultraweak variational formulation** is set to **On**, there is an extra application mode variable in addition to those listed in Table 5-2 on page 69:

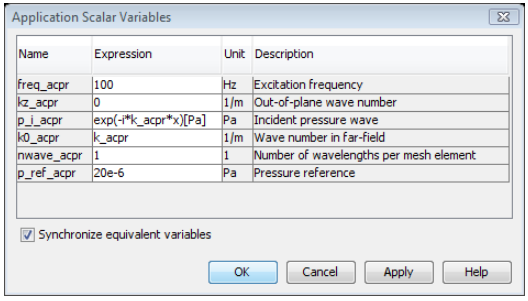
TABLE 5-3: UWVF-SPECIFIC APPLICATION MODE VARIABLE

QUANTITY	VARIABLE	ANALYSIS	GEOMETRIES	DESCRIPTION
n	nwave	H	3D, 2D, 1D	Number of wavelengths per mesh element

The software uses this variable to determine the number of plane-wave basis directions per element, N, using the formula

$$N_\varepsilon = 2\pi n + 1.8(-\log(\varepsilon))^{2/3}(2\pi n)^{1/3}$$

where $\varepsilon = 0.01$ is the tolerance for the relative error. You can modify the value of **nwave** in the **Application Scalar Variables** dialog box; the default value is 1.



The value of N used for the elements is the number from a dimension-dependent set of integers that is closest to N_ε; see Table 5-4.

TABLE 5-4: AVAILABLE NUMBERS OF PLANE-WAVE BASIS DIRECTIONS FOR THE UWVF FINITE ELEMENTS

SPACE DIMENSION	AVAILABLE N VALUES	N VALUE FOR NWAVE = 1
1D	2	2
2D	15, 20, 25, 30, 35, 40	20
3D	12, 24, 30, 32, 50, 72	24

Application Mode Variables

Table 5-5 lists the variables available for postprocessing and for use in equations and boundary conditions. The following abbreviations are used for the analysis and domain types:

T	Transient
H	Time-harmonic
E	Eigenfrequency
M	Modal
S	Subdomain
B	Boundary

TABLE 5-5: PRESSURE ACOUSTICS APPLICATION MODE VARIABLES

NAME	SYMBOL	DOM.	ANALYSIS	DESCRIPTION	EXPRESSION
p_t	p_t	S	H T tot.	Total pressure	p
p_t	p_t	S	H T sc.	Total pressure	$p+p_\mathrm{i}$
p_s	p_s	S	H T tot.	Scattered pressure	$p-p_\mathrm{i}$
p_s	p_s	S	H T sc.	Scattered pressure	p
omega	ω	S	H M	Angular frequency	$2\pi f$
omega	ω	S	E	Angular frequency	$-i\lambda$
a*i*	a_i	S	E M	Acceleration, x_i component	$-\frac{1}{\rho_0}\frac{\partial p}{\partial x_i}+q_{x_i}$
a*i*	a_i	S	H T	Acceleration, x_i component	$-\frac{1}{\rho_0}\frac{\partial p}{\partial x_i}+q_{x_i}$
v*i*	v_i	S	E	Velocity, x_i component	$\left(-\frac{1}{\rho_0}\frac{\partial p}{\partial x_i}+q_{x_i}\right)\frac{1}{i\omega}$

TABLE 5-5: PRESSURE ACOUSTICS APPLICATION MODE VARIABLES

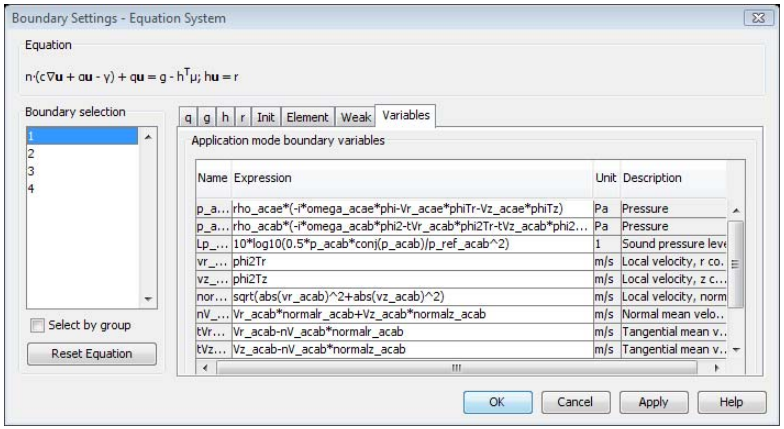
NAME	SYMBOL	DOM.	ANALYSIS	DESCRIPTION	EXPRESSION
v*i*	v_i	S	H	Velocity, x_i component	$\left(-\frac{1}{\rho_0}\frac{\partial p}{\partial x_i}+q_{x_i}\right)\frac{1}{i\omega}$
Lp	L_p	S	E M	Sound pressure level	$10\log_{10}(pp^*/(2p_{\mathrm{ref}}{}^2))$
Lp	L_p	S	H T	Sound pressure level	$10\log_{10}(p_{\mathrm{t}}p_{\mathrm{t}}{}^*/(2p_{\mathrm{ref}}{}^2))$
Lp_s	$L_{p,s}$	S	H T	Sound pressure level, scattered pressure	$10\log_{10}(p_{\mathrm{s}}p_{\mathrm{s}}{}^*/(2p_{\mathrm{ref}}{}^2))$
k	k	S	H	Wave number	ω/c
normv	$\|\mathbf{v}\|$	S	H M E	Local velocity, norm	$\sqrt{\mathbf{v}\cdot\mathbf{v}}$
norma	$\|\mathbf{a}\|$	S	all	Local acceleration, norm	$\sqrt{\mathbf{a}\cdot\mathbf{a}}$
delta	δ	S	H	Scaling factor	$1/\omega^2$
I*i*	I_i	S	H M E	Intensity, x_i component	$v_i{}^* p$
normI	$\|\mathbf{I}\|$	S	H M E	Intensity, norm	$\sqrt{\mathbf{I}\cdot\mathbf{I}}$
PMLx*i*	$\mathrm{PML}x_i$	S	H M E	PML coordinate x_i, Cartesian PML	$\mathrm{sign}(x_i-X_{i0}+\mathrm{eps})\|x_i-X_{i0}+\mathrm{eps}\|^n \times L_{x_i}(1-i)/dx_i^n$ $n \equiv$ PML scaling exponent
rx	$\mathrm{r}x$	S	H M E	r vector in PML cylinder, x-coord., cylindrical PML (3D)	$(y_{\mathrm{axis}}{}^2+z_{\mathrm{axis}}{}^2)(x-x_0)-(y_{\mathrm{axis}}(y-y_0)+z_{\mathrm{axis}}(z-z_0))x_{\mathrm{axis}}$
ry	$\mathrm{r}y$	S	H M E	r vector in PML cylinder, y-coord., cylindrical PML (3D)	$(z_{\mathrm{axis}}{}^2+x_{\mathrm{axis}}{}^2)(y-y_0)-(z_{\mathrm{axis}}(z-z_0)+x_{\mathrm{axis}}(x-x_0))y_{\mathrm{axis}}$
rz	$\mathrm{r}z$	S	H M E	r vector in PML cylinder, z-coord., cylindrical PML (3D)	$(x_{\mathrm{axis}}{}^2+y_{\mathrm{axis}}{}^2)(z-z_0)-(x_{\mathrm{axis}}(x-x_0)+y_{\mathrm{axis}}(y-y_0))z_{\mathrm{axis}}$
normr		S	H M E	r vector in PML cylinder, norm, cylindrical PML (3D)	$[\mathrm{rx}^2+\mathrm{ry}^2+\mathrm{rz}^2]^{1/2}$
R	R	S	H M E	Scaled radial coordinate, cylindrical PML (3D)	$R_0+(\mathrm{normr}-R_0)^n L_r(1-i)/dr^n$ $n \equiv$ PML scaling exponent

TABLE 5-5: PRESSURE ACOUSTICS APPLICATION MODE VARIABLES

NAME	SYMBOL	DOM.	ANALYSIS	DESCRIPTION	EXPRESSION
PMLx*i*	$\mathrm{PML}x_i$	S	H M E	PML coordinate x_i, cylindrical PML (3D)	$x_i + (-1+R/\mathrm{normr})\mathrm{r}x_i + (\mathbf{r}_{\mathrm{axis}}\cdot(\mathbf{x}-\mathbf{x}_0))/\lvert\mathbf{r}_{\mathrm{axis}}\rvert - Z_0)^n L_z(1-i)/dz^n - \mathbf{r}_{\mathrm{axis}}\cdot(\mathbf{x}-\mathbf{x}_0)x_{i\mathrm{axis}}/\lvert\mathbf{r}_{\mathrm{axis}}\rvert^2$
R	R	S	H M E	Scaled radial coordinate, cylindrical PML (2D)	$R_0 + (\delta_0 - R_0)^n L_r(1-i)/dr^n$ $n \equiv$ PML scaling exponent
PMLx*i*	$\mathrm{PML}x_i$	S	H M E	PML coordinate x_i, cylindrical PML (2D)	$R(x_i - x_{0i})/\delta_0$, $\delta_0 \equiv [(x-x_0)^2+(y-y_0)^2]^{1/2}$
R	R	S	H M E	Scaled radial coordinate, spherical PML	$R_0 + (\Delta_0 - R_0)^n (1-i) L_r/d_r^{\,n}$ $\Delta_0 \equiv [(x-x_0)^2+(y-y_0)^2+(z-z_0)^2]^{1/2}$ $n \equiv$ PML scaling exponent
PMLx*i*	$\mathrm{PML}x_i$	S	H M E	PML coordinate x_i, spherical PML	$R(x_i - x_{0i})/\Delta_0$
Jx*i*x*j*	$J_{x_i x_j}$	S	H M E	PML transformation matrix, element $x_i x_j$	$\frac{\partial}{\partial x_j}\mathrm{PML}x_i$
invJx*i*x*j*		S	H M E	PML inverse transformation matrix, element $x_i x_j$	$(\mathbf{J}^{-1})_{x_i x_j}$
detJ	$\lvert\mathbf{J}\rvert$	S	H M E	Determinant of PML transformation matrix	$\det(\mathbf{J})$
pPMLx*i*		S	H M E	Pressure derivative in PMLx_i direction	$\frac{\partial p}{\partial x_j}\cdot(\mathbf{J}^{-1})_{x_j x_i}$
nk		B	All	Normal component of wave vector	$\mathbf{n}\cdot\mathbf{k}$
na		B	All	Normal acceleration	$\mathbf{n}\cdot\mathbf{a}$
nv		B	H E	Normal velocity	$\mathbf{n}\cdot\mathbf{v}$

You can find the complete list of application mode variables defined at each domain level in the corresponding equation-system dialog box. So, for example, to see which variables you can use for postprocessing at the boundary level, open the **Physics** menu and select **Equation System>Boundary Settings**. This action launches the **Boundary Settings - Equation System** dialog box in which you find a tab labeled **Variables**. Clicking

this tab you find a list containing the application mode variables defined on boundaries for all application modes included in your model.



Note: In some cases the variables page is absent. There is, for example, no variables page at the edge level in 3D. However, you can still plot variables on edges as long as they are defined on the adjacent boundaries. In such situations, for each point on the edge the software calculates the average value of the quantity you want to plot at the nearest elements on each side of the edge.

References

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 Journal on Applied Mathematics*, 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. O. Cessenat and B. Després, "Using plane waves as base functions for solving time harmonic equations with the ultra weak variational formulation," *J. Comput. Acoustics*, vol. 11, pp. 227–238, 2003.

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

Pressure Acoustics, Modal Analysis

When you feed an acoustic wave of a given angular frequency, ω, 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

$$p(\mathbf{r}) = \sum_{j=0}^{N} p_j(x,y)e^{-ik_{zj}z}.$$

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 modal analysis capabilities in the Pressure Acoustics application mode allow you to solve such eigenvalue problems.

For an example model involving modal analysis, see "Absorptive Muffler" on page 74 in the *Acoustics Module User's Library*.

Variables and Dimensions

The modal analysis types of the Pressure Acoustics application modes solve for the transverse eigenmodes for the acoustic pressure, p, and the associated propagation constants, k_z.

The Pressure Acoustics, Boundary modal analysis application mode (acbm) is available for 3D Cartesian and 2D axisymmetric geometries, while the Pressure Acoustics application mode (acpr) supports modal analysis on 2D Cartesian and 1D axisymmetric geometries.

PDE Formulation

In 3D boundary modal analysis and in 2D modal analysis, the eigenvalue solver computes a specified number of solutions $\{p_j, \lambda_j\}$ to the equation

$$\nabla \cdot \left(-\frac{1}{\rho}(\nabla p - \mathbf{q})\right) - \left(\frac{\omega^2}{\rho_0 c^2} - \frac{k_z^2}{\rho_0}\right) p = Q \tag{5-7}$$

defined on a 2D boundary of the modeling domain (in 3D) or on the 2D domain itself, with $\lambda = -ik_z$ as the eigenvalue. In this equation p is the pressure, ρ is the density, c is the speed of sound, ω is the angular frequency, and k_z is the propagation constant, in this context also referred to as the out-of-plane wave number.

Note: Although the out-of-plane wave number is denoted k_z, the two-dimensional surface on which Equation 5-7 is defined does not necessarily have to be normal to the z-axis for 3D geometries.

The dipole source, $\mathbf{q}$, and the monopole source, Q, are normally ignored by the solver because they are independent of the dependent variable, p; thus they are not considered further in this section. When solving a coupled eigenvalue problem, however, these terms can be of interest.

Note that the above equation is identical to the 2D time-harmonic Equation 5-4, except that k_z is interpreted as an eigenvalue and not as a parameter.

For axisymmetric geometries, both 2D and 1D, the relevant eigenvalue equation to solve for the radial pressure modes and the eigenvalues, λ, is

$$\frac{d}{dr}\left(\frac{r}{\rho_0}\frac{dp}{dr}\right) + \left[\left(\frac{\omega}{c_s}\right)^2 + \lambda^2 - \left(\frac{m}{r}\right)^2\right]\frac{rp}{\rho_0} = 0$$

Here m, the circumferential wave 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 $\lambda = -ik_z$.

Application Scalar Variables

The application scalar variables defined in modal analysis are given in Table 5-6.

TABLE 5-6: APPLICATION SCALAR VARIABLES FOR PRESSURE ACOUSTICS MODAL ANALYSIS

QUANTITY	VARIABLE	DEFAULT	UNIT	GEOMETRIES	DESCRIPTION
f	freq	100	Hz	All	Frequency
ik_z	ikz	-lambda	1/m	3D, 2D	Imaginary out-of-plane wave number

TABLE 5-6: APPLICATION SCALAR VARIABLES FOR PRESSURE ACOUSTICS MODAL ANALYSIS

QUANTITY	VARIABLE	DEFAULT	UNIT	GEOMETRIES	DESCRIPTION
ik_z	ikz	-lambda	1/m	2D axi, 1D axi	Imaginary axial wave number
m	m	0		2D axi, 1D axi	Circumferential wave number
p_{ref}	p_ref	20e-6	Pa		SPL pressure reference

To inspect or modify the current settings, choose **Physics>Scalar Variables** to open the **Application Scalar Variables** dialog box.

Application Mode Properties

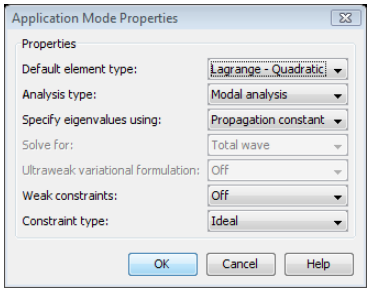


Table 5-7 lists the application mode properties defined for modal analysis.

TABLE 5-7: APPLICATION MODE PROPERTIES FOR PRESSURE ACOUSTICS MODAL ANALYSIS

PROPERTY	VALUES	DESCRIPTION
Default element type	Lagrange elements of order 1–5 (the default value is 2)	Specifies which type of finite elements to use.
Specify eigenvalues using	Propagation constant | Eigenvalue | Phase velocity	Specifies the quantity in which the solver parameters should be given.
Weak constraints	On | Off	Specifies if weak constraints should be used.
Constraint type	Ideal | Non-ideal	Specifies the type of constraint.

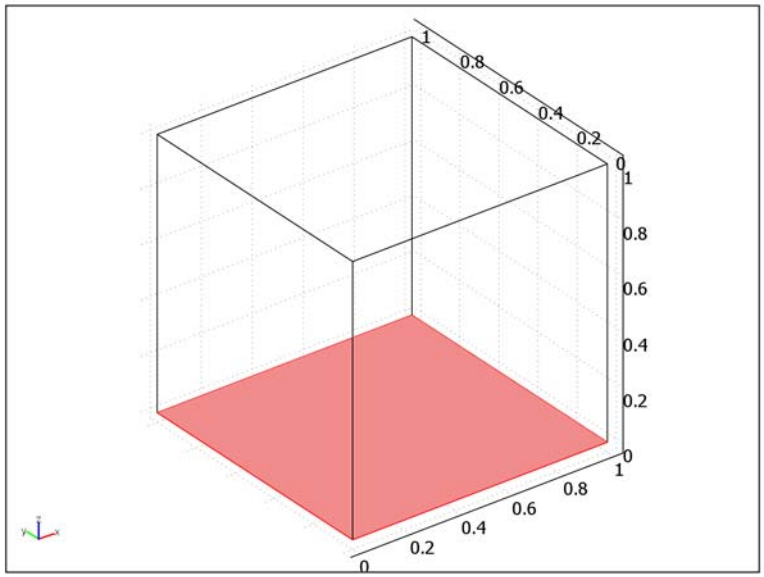
Where damping is available, the same damping models as for time-harmonic and eigenfrequency analysis apply; see page 71.

Subdomain Settings

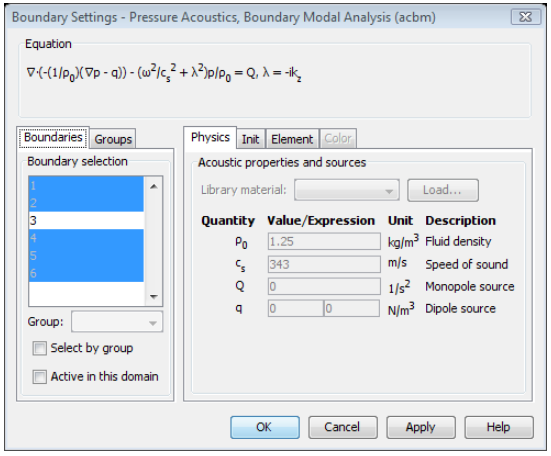
The topic of this subsection is how to set the properties on the domain where the relevant eigenvalue equation is defined. For 3D and 2D axisymmetric geometries this domain is part of the boundary of the geometry in the drawing area. In the 2D and 1D axisymmetric cases, on the other hand, the eigenvalue equation is defined at the subdomain level. For this reason you specify the coefficients appearing in the eigenvalue equation in the **Boundary Settings** dialog box for the former two geometry types and in the **Subdomain Settings** dialog box for the latter two. Once you have found the appropriate dialog box, the procedure for specifying equation parameters and damping models (when applicable) are identical. Thus see the section "Subdomain Settings" on page 70 for further information and instructions.

There is another consequence of the fact that the boundary modal analysis types in 3D and 2D axisymmetric geometries are defined in one dimension lower than that of the geometry in the drawing area: you must explicitly deactivate the acbm application mode on the boundaries where you do not want to solve the eigenvalue problem. Typically these boundaries correspond to the duct or waveguide walls.

Consider, as an example, the cube in the next figure and assume that a Pressure Acoustics, Boundary modal analysis application mode is defined on this 3D geometry. Further assume that you want to find the boundary modes on the bottom surface.



The following image shows the **Boundary Settings** dialog box for this application mode.



At the bottom left of the dialog box, inside the **Boundary Selection** area, is the **Active in this domain** check box. To deactivate the application mode on all boundaries except the bottom face, select the remaining five boundaries in the **Boundary selection list** (or

by Ctrl-clicking in the drawing area), then clear the **Active in this domain** check box. Once this is done the edges of the bottom face turn from interior to exterior boundaries. Consequently, the full set of boundary conditions becomes available, allowing you to specify conditions on these edges that reflect the boundary conditions on the cube's inner walls.

Boundary Conditions

Table 5-8 lists the availability of different boundary conditions for modal analysis.

TABLE 5-8: BOUNDARY CONDITIONS, PRESSURE ACOUSTICS MODAL ANALYSIS

BOUNDARY CONDITION	3D	2D AXI	2D	1D AXI
Sound-hard	E	P	B	B
Sound-soft	E	P	B	B
Pressure	E	P	B	B
Normal acceleration	E	P	B	B
Impedance	E	P	B	B
Axial	-	P	-	B
Radiation	-	-	B	B
Matched boundary	-	-	B	-

Here the term "boundary condition" refers to conditions that you can impose on the boundary of the domain where the eigenvalue equation is defined. The letters in the table indicate which dialog box you should open to set or inspect such boundary conditions:

B	Boundary Settings
E	Edge Settings
P	Point Settings

The expressions for the boundary conditions agree almost fully with those presented for the nonmodal analysis types in the subsection "Boundary Conditions" on page 76. The last two conditions in Table 5-8 deserve some further comments, though.

RADIATION CONDITION

The radiation conditions for 2D and 1D axisymmetric geometries are the same 1st-order expressions as those used for eigenfrequency analysis:

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

Here $k = \omega / c_s$ is the wave number, and $\kappa(r) = 0$ for an outgoing plane wave (only in 2D) and $\kappa(r) = 1/(2r)$ for a cylindrical wave (the spherical-wave option does not apply). For further details, see page 82.

MATCHED BOUNDARY

The matched boundary condition for modal analysis in 2D is identical to the one for time-harmonic analysis. Therefore see the discussion in the subsection "Matched Boundary" on page 84.

Application Mode Variables

The application mode variables for the modal analysis types are included in Table 5-5 on page 94, which lists the variables for all Pressure Acoustics analysis types.

6

Aeroacoustics

This chapter describes the two Aeroacoustics application modes in the Acoustics Module:

- Aeroacoustics (acae)
- Aeroacoustics, Boundary modal analysis (acab)

As in the previous chapter on pressure acoustics, the latter mode is discussed in a separate section together with the modal analysis type of the Aeroacoustics application mode.

In addition, the chapter contains a presentation of the application mode Compressible Potential Flow (acpf), which is included in the Acoustics Module to allow fully dynamic simulations of the fluid in which the acoustic waves propagate.

The chapter begins with a brief theory review.

Theory Background

For a mathematical description of the noise generated in fluids in turbulent motion or by the aerodynamic forces acting in the interface between a moving fluid and a structural boundary, the acoustic pressure is not the most suitable dependent variable to use. Instead, to couple acoustics and fluid dynamics, a formulation based on the potential field for the particle velocity, has been developed. This is the scientific field of *aeroacoustics*.

This section presents the basic mathematical framework for aeroacoustics, starting with the equations for the acoustic waves in a given mean-flow velocity field. Then follows a presentation of the equations describing the dynamics of this background flow, which is assumed to be compressible, inviscid, barotropic, and irrotational.

Descriptions of the associated application modes appear in the subsequent sections.

Aeroacoustics

In aeroacoustics, 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

$$\mathbf{v}_{\text{tot}}(\mathbf{r}, \mathbf{t}) = \mathbf{V}(\mathbf{x}) + \mathbf{v}(\mathbf{r}, t) \tag{6-1}$$

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 equation for the velocity potential, ϕ, governing acoustic waves in a background flow with mean velocity, $\mathbf{V}$, mean density, ρ, and mean speed of sound, c_s is

$$-\frac{\rho}{c_s^2}\frac{\partial}{\partial t}\left(\frac{\partial\phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right) + \nabla\cdot\left[\rho\nabla\phi - \frac{\rho}{c_s^2}\left(\frac{\partial\phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right)\mathbf{V}\right] = 0 \tag{6-2}$$

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 6-1. Then, linearizing the resulting equations in the acoustic perturbation and eliminating all acoustic variables except the velocity potential gives Equation 6-2. Thus, the density, ρ, in this equation is the time-independent part. The corresponding acoustic part is $\rho_a(\mathbf{r},t) = p(\mathbf{r},t)/c_s^2$ where p is the acoustic pressure, given by

$$p(\mathbf{r}, t) = -\rho\left(\frac{\partial \phi}{\partial t} + \mathbf{V} \cdot \nabla \phi\right)$$

Hence, once Equation 6-2 has been solved for the velocity potential the acoustic pressure can easily be calculated.

When transformed to the frequency domain, the wave equation 6-2 reads

$$-\frac{\rho}{c_s^2} i\omega(i\omega\phi + \mathbf{V} \cdot \nabla\phi) + \nabla \cdot \left[\rho\nabla\phi - \frac{\rho}{c_s^2}(i\omega\phi + \mathbf{V} \cdot \nabla\phi)\mathbf{V}\right] = 0$$

while the acoustic pressure is

$$p(\mathbf{r}) = -\rho(i\omega\phi + \mathbf{V} \cdot \nabla\phi)$$

Typical boundary conditions include:

- Sound-hard boundaries or walls
- Sound-soft boundaries
- Impedance boundary conditions
- Radiation boundary conditions

These, and a few additional options available in the Aeroacoustics application mode, are described in more detail in the next section, on page 117.

Compressible Potential Flow

The previous section presented the equations for aeroacoustic waves in a background mean-flow field characterized by its velocity, density, and sound speed. This section discusses the equations of motion and state for the fluid in some detail.

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:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0$$
$$\rho\left(\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla)\mathbf{V}\right) + \nabla p = \mathbf{f} \qquad (6\text{-}3)$$
$$\frac{\partial e}{\partial t} + \nabla \cdot ((e + p)\mathbf{V}) = 0$$

Here a volume force, **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,

$$p = p_0\left(\frac{\rho}{\rho_0}\right)^{\gamma}$$

where $\gamma = c_p / c_V$ is the ratio between the specific heats at constant pressure and constant volume, while p_0 and ρ_0 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

$$\frac{dp}{dt} = \frac{dp}{d\rho}\frac{d\rho}{dt} \equiv c_s^{\,2}\frac{d\rho}{dt}$$

where, using the equation of state,

$$c_s = \sqrt{\gamma \frac{p}{\rho}}$$

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 Equations 6-3 can be integrated to yield the *Bernoulli equation*

$$\frac{\partial \Phi}{\partial t} + \left(\frac{1}{2} |\nabla \Phi|^2 + p_0 \frac{\gamma \rho^{\gamma - 1}}{(\gamma - 1) \rho_0^{\gamma}} + \Psi \right) = \frac{1}{2} v_0{}^2 + \frac{p_0 \gamma}{(\gamma - 1) \rho_0} + \Psi_0$$

In this equation, two additional reference quantities have entered: the velocity, v_0, and the force potential, Ψ_0, both valid at the same reference point as p_0 and ρ_0. Note, in particular, that neither the pressure, p, nor the energy per unit volume, e, appears in this equation.

Collecting the results, the equations governing the compressible, inviscid, irrotational flow of an ideal fluid are

$$\frac{\partial \Phi}{\partial t} + \frac{1}{2} |\nabla \Phi|^2 + p_0 \frac{\gamma \rho^{\gamma - 1}}{(\gamma - 1) \rho_0^{\gamma}} + \Psi = \frac{1}{2} v_0{}^2 + \frac{p_0 \gamma}{(\gamma - 1) \rho_0} + \Psi_0$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \nabla \Phi) = 0 \qquad c_s = \sqrt{\gamma \frac{p}{\rho}} \qquad \gamma \equiv c_p / c_V$$

Application Mode Description

Variables and Space Dimensions

The Aeroacoustics application mode (acae) solves for the velocity potential, ϕ. It is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

Note that Aeroacoustics, Boundary modal analysis (acab) counts as a separate application mode, available for 2D axisymmetric and 3D geometries. This application mode is described in the next section together with the modal analysis type for the Pressure Acoustics application mode in 2D.

The general discussion uses equations and settings that apply to the 3D case. Where the lower-dimensional geometries differ, this is explicitly noted.

PDE Formulation

The Aeroacoustics application mode provides three different analysis types:

- Transient analysis
- Time-harmonic analysis
- Modal analysis (for 2D and 1D axisymmetric geometries)

This section treats the former two cases, while modal analysis is discussed in the section "Aeroacoustics—Modal Analysis" on page 124.

TRANSIENT ANALYSIS

The following equation governs acoustic waves in a mean flow:

$$-\frac{\rho}{c_s^2}\frac{\partial}{\partial t}\left(\frac{\partial \phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right) + \nabla\cdot\left[\rho\nabla\phi - \frac{\rho}{c_s^2}\left(\frac{\partial \phi}{\partial t} + \mathbf{V}\cdot\nabla\phi\right)\mathbf{V}\right] = 0 \qquad (6\text{-}4)$$

Here ρ (kg/m^3) is the density, $\mathbf{V}$ (m/s) denotes the mean velocity, and c_s (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, $\mathbf{V}$, and c_s are approximately constant in time, while they may be functions of the spatial coordinates.

TIME-HARMONIC ANALYSIS

For harmonic waves of the form $\phi(\mathbf{r}, t) = \phi(\mathbf{r})e^{i\omega t}$, each time-derivative in Equation 6-4 can be replaced by a factor $i\omega$, leading to the frequency-domain equation

$$-\frac{\rho}{c_s^2}i\omega(i\omega\phi + \mathbf{V}\cdot\nabla\phi) + \nabla\cdot\left[\rho\nabla\phi - \frac{\rho}{c_s^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\mathbf{V}\right] = 0$$

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:

$$-i\omega\frac{\rho}{c_s^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi - ik_z V_z\phi) + \nabla\cdot\left(\rho\nabla\phi - \mathbf{V}\frac{\rho}{c_s^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi - ik_z V_z\phi)\right)$$

$$+\rho k_z^2\phi + ik_z V_z\frac{\rho}{c_s^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi - ik_z V_z\phi) = 0$$

The out-of-plane wave number is an application scalar variable (see Table 6-1) whose value you can specify; the default value is 0, that is, no wave propagation perpendicular to the 2D plane. Alternatively, by using the eigenvalue solver you can solve for k_z in a modal analysis; see the next section.

For 2D axisymmetric models, $\phi(\mathbf{r}, t) = \phi(r, z)e^{i(\omega t - m\varphi)}$, the circumferential wave number, m, similarly appears in the equation as a parameter:

$$-i\omega\frac{\rho}{c_s^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi) + \nabla\cdot\left(\rho\nabla\phi - \mathbf{V}\frac{\rho}{c_s^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\right) + \rho\frac{m^2}{r^2}\phi = 0$$

Note that the background velocity field, $\mathbf{V}$, cannot have a circumferential component because the flow is irrotational.

Application Scalar Variables

The application scalar variables are given in the following table.

TABLE 6-1: AEROACOUSTICS APPLICATION MODE SCALAR VARIABLES

NAME	SYMBOL	ANALYSIS TYPES/GEOMETRIES	DEFAULT	UNIT	DESCRIPTION
freq	f	Time-harmonic, Modal	100	Hz	Excitation frequency
pref	p_{ref}	All	20e-6	Pa	Pressure reference
ikz		Time-harmonic /2D	0	1/m	Imaginary out-of-plane wave number

TABLE 6-1: AEROACOUSTICS APPLICATION MODE SCALAR VARIABLES

NAME	SYMBOL	ANALYSIS TYPES/GEOMETRIES	DEFAULT	UNIT	DESCRIPTION
ikz		Time-harmonic / 1D Axi	0	1/m	Imaginary axial wave number
m		Time-harmonic, Modal / 2D Axi, 1D Axi	0	1	Circumferential wave number
ikz		Modal / 3D	`-lambda`	1/m	Imaginary out-of-plane wave number
ikz		Modal / 2D axi	`-lambda`	1/m	Imaginary axial wave number

You can change the settings from their default values in the **Application Scalar Variables** dialog box, which you open by selecting the menu item **Scalar Variables** from the **Physics** menu.

Application Mode Properties

Table 6-2 lists the properties that can be set in the **Application Mode Properties** dialog box that you can reach from the **Model Navigator** when selecting or adding application modes or by choosing **Physics>Properties** during a modeling session.

TABLE 6-2: AEROACOUSTICS APPLICATION MODE PROPERTIES

PROPERTY	VALUES	DESCRIPTION
Default element type	Lagrange elements of order 1–5 (the default value is 2)	Specifies which type of finite elements to use
Analysis type	Time-harmonic | Modal analysis | Transient	Specifies which type of analysis to perform
Specify eigenvalues using	Eigenvalue | Propagation constant | Phase velocity	Specifies the quantity in which the solver parameters should be given; applies only to modal analysis
Weak constraints	On | Off	Specifies if weak constraints should be used
Constraint type	Ideal | Non-ideal	Specifies the type of constraint

For details on the Modal analysis type, see the section “Aeroacoustics—Modal Analysis” on page 124. For a review of element types and weak constraints refer to the sections “Understanding the Finite Element Method” on page 452 of the *COMSOL Multiphysics Reference Guide* and “Using Weak Constraints” on page 300 in the *COMSOL Multiphysics Modeling Guide*.

Subdomain Settings

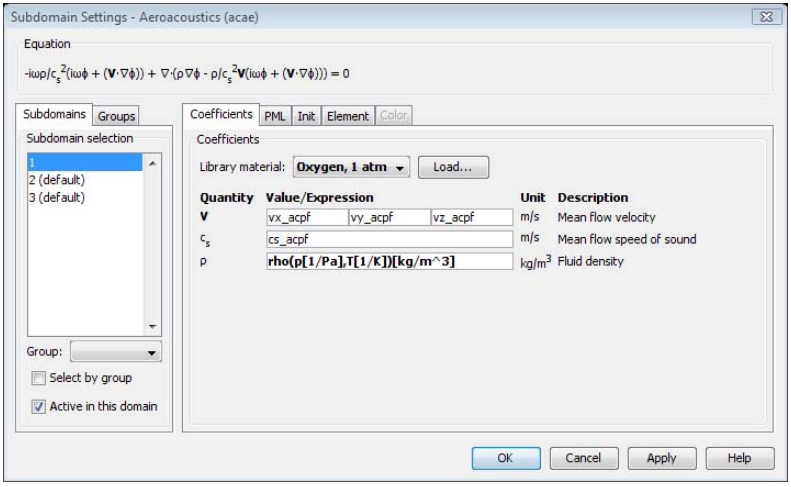
The **Subdomain Settings** dialog box contains the following pages, each accessible by clicking the corresponding tab:

- **Coefficients**—This is where you specify acoustic properties pertaining to subdomains.
- **PML**—In time-harmonic and modal analysis you can include auxiliary subdomains in the geometry that serve as perfectly matched layers; use this page to specify their properties.
- **Init**—For transient analysis, use this page to supply initial values for the velocity potential and its first-order time derivative. For the other analysis types, you can specify a nontrivial starting solution for the nonlinear solver, a feature that can be relevant when solving coupled multiphysics models.
- **Elements**—Here you can change the specifications for the finite elements to be used in the subdomain; for further details, refer to the subsection "Specifying the Finite Element Type" on page 211 of the *COMSOL Multiphysics User's Guide*.
- **Color**—This page is only active when the **Groups** page is open. Use it to visually distinguish between groups of subdomains with common settings by giving them different colors in the user interface. For 1D axisymmetric and 1D geometries, the tab is labeled **Color/Style** because there you can also set the line style.

COEFFICIENTS

On the **Coefficients** page you can specify the mean-flow velocity, $\mathbf{V}$, the mean-flow speed of sound, c_s, and the fluid density, ρ. The default values for the latter two quantities apply to air at atmospheric pressure and room temperature. If these default settings do not apply in your model, you can search the materials libraries for the relevant medium. To do so, select any subdomain, click the **Load** button to activate the **Library material** list, and then browse the available libraries. For detailed information

on how to use the materials libraries, consult "Using the Materials/Coefficients Library" on page 223 of the *COMSOL Multiphysics User's Guide*.



The default mean-flow velocity is zero unless you are using the application mode Aeroacoustics with Flow—a combination of the Aeroacoustics (acae) and Compressible Potential Flow (acpf) application modes; in such a case the default setting is $\mathbf{V} = \mathbf{v}_{acpf}$, as exemplified in the previous figure.

PERFECTLY MATCHED LAYERS (PMLS)

The Aeroacoustics application mode offers an absorbing boundary condition, the radiation boundary condition, which is perfectly absorbing for a plane wave. For models where you cannot describe the incident radiation as a plane wave with a well-known direction of propagation, you can use perfectly matched layers (PMLs) to emulate nonreflecting boundaries. For further details, see the discussion under the heading "Perfectly Matched Layers (PMLs)" on page 37.

The PML page is available for time-harmonic and modal analysis. The following types of PMLs are available:

TYPE	GEOMETRY	DESCRIPTION
None	All	Not absorbing
Cartesian	3D, 2D, 1D	Absorbing in the specified coordinate directions
Cylindrical	3D, 2D axi, 1D, 1D axi	Absorbing in the radial or axial direction

TYPE	GEOMETRY	DESCRIPTION
Spherical	3D, 2D axi,	Absorbing in the radial direction
User defined	All	User-defined PML coordinates

Boundary Conditions

To complete the mathematical formulation of your model you must supply a set of boundary conditions that properly reflects the physics at the interface between the modeling domain and its surroundings. This subsection describes the options at your disposal in the Aeroacoustics application mode.

SOUND-HARD BOUNDARIES (WALLS)

For modeling rigid boundary surfaces, or walls, use the *sound-hard* condition. 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:

$$-\mathbf{n}\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{V}\cdot\nabla\phi\right)\right)=0$$

The sound-hard boundary condition is available for all analysis types. The equation above applies to the transient case; to obtain the corresponding condition for time-harmonic acoustic waves, simply replace $\partial/\partial t$ by $i\omega$.

SOUND-SOFT BOUNDARIES

At a *sound-soft* boundary, the acoustic pressure vanishes:

$$p = 0$$

Use this condition to model the interface between a liquid and a gas.

NORMAL MASS FLOW

The natural boundary condition for the total wave has the meaning of a mass flow through the boundary surface:

$$-\mathbf{n}\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{V}\cdot\nabla\phi\right)\right)=m_n$$

To specify a normal mass flow boundary condition, type the value of the *inward* mass flow, m_n, in the $\mathbf{m_n}$ edit field.

VELOCITY POTENTIAL

When coupling two Aeroacoustics application modes together it can sometimes be necessary to set the velocity potential:

$$\phi = \phi_0$$

For an example model using the velocity potential boundary condition, see the model "Flow Duct" on page 101 of the *Acoustics Module Model Library*.

IMPEDANCE

In time-harmonic analysis you can define the input impedance of the external domain as the ratio of pressure to normal velocity, $Z_i = p/(\mathbf{n}\cdot\mathbf{v})$ at the boundary. The associated impedance boundary condition reads

$$\mathbf{n}\cdot\left(\rho\nabla\phi - \mathbf{V}\frac{\rho}{c_s^2}(i\omega\phi + \mathbf{V}\cdot\nabla\phi)\right) = \rho\left(\frac{p}{Z} + \frac{1}{i\omega}(\mathbf{V}\cdot\nabla)\frac{p}{Z}\right)$$

RADIATION CONDITION

This is a class of non-reflecting boundary conditions which assumes that there is an outgoing plane wave, and optionally also an incoming exciting wave.

For transient analysis the boundary condition is

$$-\mathbf{n}\cdot\left(\rho\nabla\phi - \mathbf{V}\frac{\rho}{c_s^2}\left(\frac{\partial\phi}{\partial t} + (\nabla\phi\cdot\mathbf{V})\right)\right) - \rho k_n\frac{\partial\phi}{\partial t} - \mathbf{n}\cdot\mathbf{V}\frac{\rho}{c_s^2}\left(\frac{\partial\phi}{\partial t} - k_n\frac{\partial\phi}{\partial t}\mathbf{n}\cdot\mathbf{V}\right) =$$

$$\rho\frac{\partial}{\partial t}\phi_0 k_k(\mathbf{n}\cdot\mathbf{n_k}) - \mathbf{n}\cdot\mathbf{V}\frac{\rho}{c_s^2}k_k\frac{\partial\phi_0}{\partial t}\mathbf{n_k}\cdot\mathbf{V} - \rho k_n\frac{\partial\phi_0}{\partial t} + \mathbf{n}\cdot\mathbf{V}\frac{\rho}{c_s^2}\left(k_n\frac{\partial\phi_0}{\partial t}\mathbf{n}\cdot\mathbf{V}\right)$$

$$k_k = \frac{1}{c_s + \mathbf{V}\cdot\mathbf{n_k}} \qquad k_n = \frac{1}{c_s + \mathbf{V}\cdot\mathbf{n}}$$

while the corresponding time-harmonic equation reads

$$-\mathbf{n}\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^2}(i\omega\phi+\nabla\phi\cdot\mathbf{V})\right)-\rho\,ik_n\phi-\mathbf{n}\cdot\mathbf{V}\frac{\rho}{c_s^2}(i\omega-ik_n\mathbf{n}\cdot\mathbf{V})\phi\;=$$

$$\rho\left(ik_k\mathbf{n}\cdot\mathbf{n}_\mathbf{k}-\mathbf{n}\cdot\frac{\mathbf{V}}{c_s^2}(ik_k\mathbf{n}_\mathbf{k}\cdot\mathbf{V})-ik_n+\mathbf{n}\cdot\frac{\mathbf{V}}{c_s^2}(ik_n\mathbf{n}\cdot\mathbf{V})\right)\phi_0 e^{i(\mathbf{k}\cdot\mathbf{r})}$$

$$k_k=\frac{\omega}{c_s+\mathbf{V}\cdot\mathbf{k}}\qquad k_n=\frac{\omega}{c_s+\mathbf{V}\cdot\mathbf{n}}\qquad \mathbf{k}=k\,\mathbf{n}_\mathrm{k}$$

You specify an incoming plane wave, $\phi_0 e^{-i(\mathbf{k}\cdot\mathbf{r})}$, by supplying its amplitude, ϕ_0, and propagation vector, $\mathbf{n}_\mathrm{k}$.

This boundary condition is most relevant for ports, because many wave-guide structures are only interesting in the plane-wave region.

NORMAL VELOCITY

In time-harmonic analysis you can specify the velocity component normal to the boundary:

$$\mathbf{n}\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^2}(i\omega\phi+\mathbf{V}\cdot\nabla\phi)\right)=\rho\left(v_n+\frac{1}{i\omega}\mathbf{V}\cdot\nabla v_n\right)$$

Here v_n denotes the outward normal velocity at the boundary surface, which you specify in the $\mathbf{v_n}$ edit field.

INTERFACE CONDITIONS ON INTERIOR BOUNDARIES

By default only exterior boundaries are active in the **Boundary Settings** dialog box. However, if you select the **Interior boundaries** check box you can specify the interface conditions for boundaries inside the modeling domain that serve as partitions between subdomains.

The following options are available in the Aeroacoustics application mode (subscripts 1 and 2 refer to the two sides of the boundary):

- **Continuity**:

$$\mathbf{n}\cdot\left[\left(\rho\left(\nabla\phi-\frac{\mathbf{V}}{c_\mathrm{s}^2}(i\omega+\mathbf{V}\cdot\nabla)\phi\right)\right)_1-\left(\rho\left(\nabla\phi-\frac{\mathbf{V}}{c_\mathrm{s}^2}(i\omega+\mathbf{V}\cdot\nabla)\phi\right)\right)_2\right]=0$$

  This condition, which expresses the continuity of the mass flow, is the default setting. It corresponds to a situation where the interior boundary has no direct effect on the acoustic velocity potential field.

- **Sound soft boundary**: $p = 0$
- **Pressure**: $p = p_0$

INTERFACE CONDITIONS ON PAIRS

If you have pairs connecting different parts of an assembly, you can set boundary conditions on the pairs. To do so, click on the **Pairs** tab on the **Boundary Settings** dialog box to activate the **Pairs selection** list and the associated **Boundary condition** list.

Because pairs can form interior boundaries, the boundary conditions available for interior boundaries listed in the previous paragraph are available also for pairs. In addition, pairs can have *slit boundary conditions*, that is, boundary conditions for which the dependent variable need not be continuous across the boundary.

The following slit boundary conditions are available in the Aeroacoustics application mode (subscripts 1 and 2 refer to the two sides of the boundary):

- **Sound hard boundary (wall)**: For transient analysis, this boundary condition reads:

$$\left[-\mathbf{n}\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{V}\cdot\nabla\phi\right)\right)\right]_i = 0 \qquad i = 1,2$$

  As usual, replace time-derivatives by the factor $i\omega$ to obtain the corresponding time-harmonic equations.

- **Impedance boundary condition**: This boundary condition, available for time-harmonic analysis, allows you to specify the impedance, Z, of an interior boundary. The equations defining this boundary conditions are:

$$\left[-\mathbf{n}\cdot\left(\rho\nabla\phi+\frac{\mathbf{V}}{c_s^2}p\right)\right]_1 = \frac{\rho}{i\omega Z}(i\omega+\mathbf{V}\cdot\nabla)(p_1-p_2)$$

$$\left[-\mathbf{n}\cdot\left(\rho\nabla\phi+\frac{\mathbf{V}}{c_s^2}p\right)\right]_2 = \frac{\rho}{i\omega Z}(i\omega+\mathbf{V}\cdot\nabla)(p_2-p_1)$$

$$p_i = [-\rho(i\omega+\mathbf{V}\cdot\nabla)\phi]_i \qquad i = 1,2$$

- **Vortex sheet**: To model a shear layer that separates a stream from the free velocity field use a vortex sheet boundary condition. Because the velocity potential is discontinuous over this boundary you must use an assembly and set the boundary condition on a pair.

  The equations defining the vortex sheet boundary conditions are

$$\left[\mathbf{n}\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^2}(i\omega\phi+\mathbf{V}\cdot\nabla\phi)\right)\right]_i = [\rho(i\omega+\mathbf{V}\cdot\nabla)w]_i \qquad i = 1,2$$

$$p_1 = p_2 \qquad w_1 = -w_2$$

where w denotes the outward normal displacement of the boundary surface.

This boundary condition is available for models using time-harmonic analysis.

The model "Jet Pipe" on page 34 of the *Acoustics Module Model Library* provides an example for how to apply the vortex-sheet boundary condition.

Point and Edge Conditions

Point and edge conditions are available to specify a mass flow rate.

$$\nabla\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^2}(i\omega\phi+\mathbf{V}\cdot\nabla\phi)\right) = m'$$

Application Mode Variables

Table 6-3 lists the application mode variables available for postprocessing purposes at the subdomain level for 3D geometries in the Aeroacoustics application mode.

TABLE 6-3: AEROACOUSTICS APPLICATION MODE 3D SUBDOMAIN VARIABLES

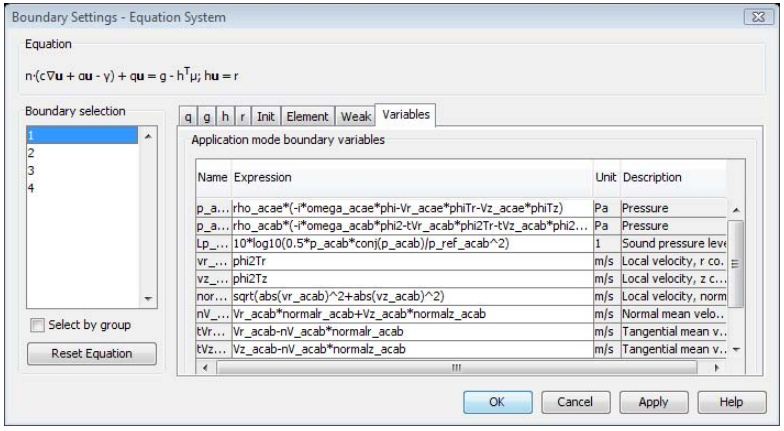
VARIABLE	SYMBOL	DESCRIPTION	EXPRESSION
omega	ω	Angular frequency	$2\pi f$
v*i*	v_i	Velocity in x_i direction	$\partial\phi/\partial x_i$
normv	$\|\mathbf{v}\|$	velocity, norm	$\sqrt{\mathbf{v}\cdot\mathbf{v}}$
p	p	Pressure	$-\rho(i\omega\phi+\mathbf{V}\cdot\nabla\phi)$
Lp	L_p	Sound pressure level	$10\log_{10}(pp^*/(2p_{\text{ref}}{}^2))$
I*i*	I_i	Intensity, x_i component	$\frac{1}{2}\text{Re}\left[\left(\frac{p}{\rho}+\mathbf{V}\cdot\mathbf{v}\right)\left(\rho v_{x_i}+\frac{p}{c_s^2}V_{x_i}\right)^*\right]$
normI	$\|\mathbf{I}\|$	Intensity, norm	$\sqrt{\mathbf{I}\cdot\mathbf{I}}$
R	R	Scaled radial coordinate, cylindrical PML	$R_0+(\delta_0-R_0)(1-i)L_r/d_r$ $\delta_0=[(x-x_0)^2+(y-y_0)^2]^{1/2}$

TABLE 6-3: AEROACOUSTICS APPLICATION MODE 3D SUBDOMAIN VARIABLES

VARIABLE	SYMBOL	DESCRIPTION	EXPRESSION
R	R	Scaled radial coordinate, spherical PML	$R_0+(\Delta_0-R_0)(1-i)L_r/d_r$ $\Delta_0=[(x-x_0)^2+(y-y_0)^2+(z-z_0)^2]^{1/2}$
PMLx*i*	$\mathrm{PML}x_i$	PML coordinate x_i	$(x_i-x_{0i})(1-i)L_i/d_i$
PMLx*i*	$\mathrm{PML}x_i$	PML coordinate x_i, cylindrical PML	$R(x_i-x_{0i})/\delta_0$ $\delta_0=[(x-x_0)^2+(y-y_0)^2]^{1/2}$
PMLx*i*	$\mathrm{PML}x_i$	PML coordinate x_i, spherical PML	$R(x_i-x_{0i})/\Delta_0$
PMLz	$\mathrm{PML}z$	PML coordinate z	$(z-z_0)(1-i)L_z/d_z$
Jxy	J_{xy}	PML transformation matrix, element xy	$\frac{\partial}{\partial x}\mathrm{PML}y$
invJxy		PML inverse transformation matrix, element xy	$(\mathbf{J}^{-1})_{xy}$
detJ	$\lvert\mathbf{J}\rvert$	Determinant of transform matrix	$\det(\mathbf{J})$
phiPMLx*i*		Pressure derivative in PMLx_i direction	$\nabla\phi\cdot((\mathbf{J}^{-1})_{xx_i},(\mathbf{J}^{-1})_{yx_i},(\mathbf{J}^{-1})_{zx_i})$

For each domain type in the current geometry, you can inspect the complete list of application mode variables defined at that level in the corresponding equation system dialog box. So, for example, to see what variables you can use for postprocessing at the boundary level, open the **Physics** menu and select **Equation System>Boundary Settings**. This action launches the **Boundary Settings - Equation System** dialog box, on which you find a tab labeled **Variables**. Clicking this tab, you find a list containing the application

mode variables defined on boundaries for all application modes included in your model.



Note: In some cases, the variables page is absent. There is, for example, no variables page at the edge level in 3D. However, you can still plot variables on edges as long as they are defined on the adjacent boundaries. In such situations, for each point on the edge, the software calculates the average value of the quantity you want to plot at the nearest elements on each side of the edge.

Aeroacoustics—Modal Analysis

The Acoustics Module provides the following types of analysis for calculating the aeroacoustical eigenmodes of ducts and waveguides:

- The Aeroacoustics, Boundary modal analysis application mode (acbm) for 3D and 2D axisymmetric geometries
- The modal analysis type of the Aeroacoustics application mode (acae) for 2D geometries

The presentation below first focuses on the case of 3D geometries, before remarking on the 2D axisymmetric and 2D cases.

For an example model involving an Aeroacoustics, Boundary modal analysis application mode, see "Flow Duct" on page 101.

Application Mode Properties

In addition to the standard properties **Default element type** and **Weak constraints**, you can specify the following property in the **Application Mode Properties** dialog box:

PROPERTY	VALUES	DESCRIPTION
Specify eigenvalues using	Eigenvalue | Propagation constant | Phase velocity	Specifies the quantity in which the solver parameters should be given

The default option is **Propagation constant**.

PDE Formulation

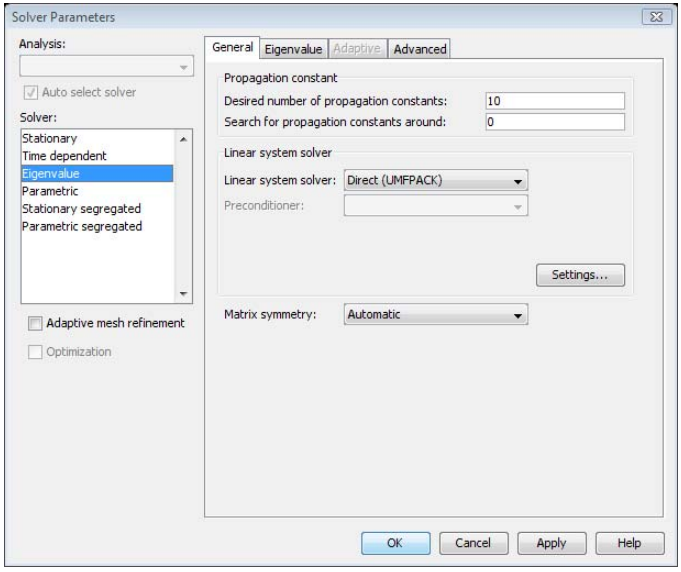
The boundary modal analysis type in 3D uses the eigenvalue solver to solve the equation

$$-i\omega\frac{\rho}{c_s^2}(i\omega\phi + (\mathbf{V}_t\cdot\nabla\phi) - ik_z V_n\phi) + \nabla\cdot\left(\rho\nabla\phi - \mathbf{V}_t\frac{\rho}{c_s^2}(i\omega\phi + (\mathbf{V}_t\cdot\nabla\phi) - ik_z V_n\phi)\right) + \rho k_z^2\phi + ik_z V_n\frac{\rho}{c_s^2}(i\omega\phi + (\mathbf{V}\cdot\nabla\phi) - ik_z V_n\phi) = 0 \quad (6\text{-}5)$$

for 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 is the density, c_s 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 mean velocity in the tangential plane while V_n is the mean-velocity component in the normal direction.

Note: Although the out-of-plane wave number is called k_z, the two-dimensional surface on which Equation 6-5 is defined does not necessarily have to be normal to the z-axis for 3D geometries.

In the **Solver Parameters** dialog box, you can specify the number of eigenmodes the solver should look for and the value around which it should start the search. Supply these values in the **Desired number of propagation constants** and **Search for propagation constants around** edit fields.



Application Scalar Variables

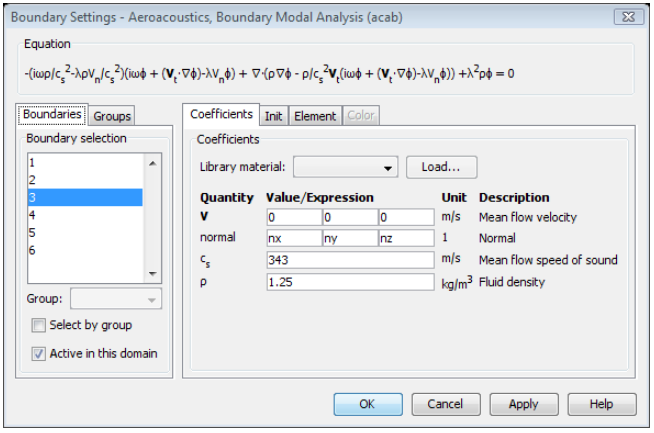
The application scalar variables are given in the table below.

NAME	DEFAULT	UNIT	DESCRIPTION
freq	100	Hz	Excitation frequency
pref	20e-6	Pa	Pressure reference for SPL
ikz	-lambda	1/m	Imaginary out-of-plane wave number

Boundary Conditions

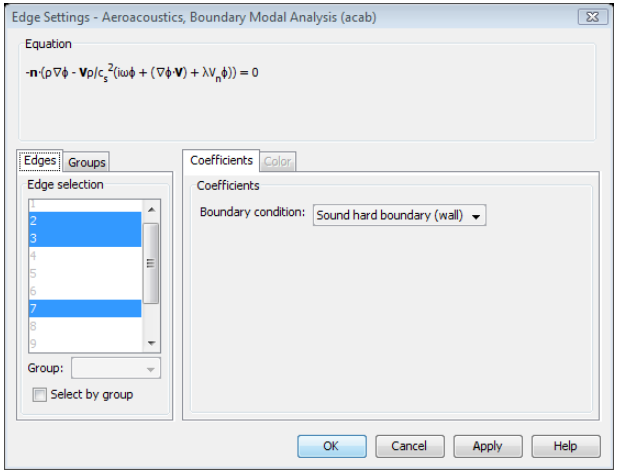
You can specify values for the following properties appearing in Equation 6-5 on the **Coefficients** page of the **Boundary Settings** dialog box:

- **Mean flow velocity**: Supply the x-, y-, and z-components of the mean-flow velocity, $\mathbf{V} = (V_x, V_y, V_z)$, in the respective boxes of the **V** edit field. By default, the mean-flow velocity is set to zero.
- **Normal**: The **normal** edit field takes the components of the outward unit normal to the boundary.
- **Mean flow speed of sound**: In the $\mathbf{c_s}$ edit field you can set the mean flow speed of sound. The default setting is 343 m/s, the speed of sound in air.
- **Fluid density**: In the ρ edit field you can set the fluid density to a value that matches the medium for the acoustic waves in your model. The default value, 1.25 kg/m^3, applies to air.



Note: In the 3D case, you must—in addition to specifying values for the above properties on the boundary surface where you want to solve Equation 6-5—inactivate the application mode on the remaining boundaries of your model geometry. Do this by selecting these boundaries in the **Boundary selection** list and then clearing the **Active in this domain** check box.

Edge Conditions



These are the conditions available on the **Coefficients** page of the **Edge Settings** dialog box for the edges of the two-dimensional boundary on which Equation 6-5 is to be solved:

- **Normal mass flow**: The natural edge condition for the total wave has the meaning of normal mass flow.

$$-\mathbf{n}\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^{\,2}}(i\omega\phi+(\nabla\phi\cdot\mathbf{V})+\lambda\phi V_{\mathrm{n}})\right)=m_{\mathrm{n}}$$

- **Sound hard boundary (wall)**: 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:

$$-\mathbf{n}\cdot\left(\rho\nabla\phi-\mathbf{V}\frac{\rho}{c_s^2}(i\omega\phi+(\nabla\phi\cdot\mathbf{V}))\right)=0$$

- **Sound soft boundary**: When the pressure is zero, the edge condition is called *sound soft*.
- **Velocity potential**: When coupling application modes together it can be necessary to set the velocity potential:

$$\phi=\phi_0$$

Application Mode Variables

VARIABLE	SYMBOL	DOMAIN	DESCRIPTION	EXPRESSION
omega	ω		Angular frequency	$2\pi f$
v*i*	v_i	B	Velocity, x_i component	$\partial\phi/\partial_{\mathrm{T}}x_i$
vnorm	$\|\mathbf{v}\|$	B	Velocity norm	$\sqrt{\mathbf{v}\cdot\mathbf{v}}$
p	p	B	Pressure	$-\rho(i\omega\phi+(\nabla_{\mathrm{T}}\phi\cdot\mathbf{V}_{\mathrm{T}}-ik_z\phi V_{\mathrm{n}}))$
Lp		B	Sound pressure level	$10\log_{10}(p\cdot p^*/(2p_{\mathrm{ref}}{}^2))$
nV	V_{n}	B	Normal mean velocity	$\mathbf{V}\cdot\mathbf{n}$
tV*i*	$(V_{\mathrm{T}})_i$	B	Tangential mean velocity, x_i component	$V_i-V_{\mathrm{n}}n_i$

2D Axisymmetric Geometries

BOUNDARY SETTINGS

In the **Boundary Settings** dialog box you specify the same properties as in the 3D case. In addition to the options available for that case, the boundary condition **Axial symmetry** applies to the symmetry axis $r = 0$. Note that the eigenvalue equation in the 2D axisymmetric case also involves the circumferential wave number, m, an integer entering the axisymmetric expression for the velocity potential:

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

You can specify the value of m in the **Application Scalar Variables** dialog box.

POINT SETTINGS

All options listed under "Edge Conditions" on page 127 apply as boundary conditions also for 2D axisymmetric geometries. In addition, you can select the option **Axial symmetry**. This is the correct setting for points on the symmetry axis $r = 0$.

2D Geometries

SUBDOMAIN SETTINGS

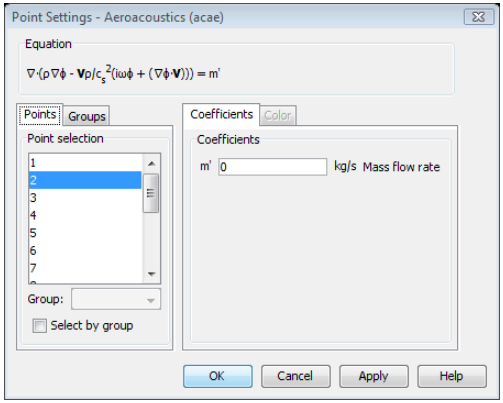
In the **Subdomain Settings** dialog box you can specify the same properties as in the **Boundary Settings** dialog box of the 3D case, except for the normal vector, $\mathbf{n}$, which in this case, by definition, is oriented out of the 2D geometry plane.

BOUNDARY SETTINGS

All options listed under "Edge Conditions" on page 127 apply. In addition, the option **Radiation condition** is available for 2D geometries.

Point Settings

In the **Point Settings** dialog box you can specify a mass flow rate m' (measured in kg/(m·s)). For time-harmonic analysis you supply the amplitude of the flow in the **m'** edit field. For transient analysis you can specify a time-dependent flow.



Compressible Potential Flow

This application mode calculates the mean flow assuming an ideal barotropic, irrotational fluid and constant entropy.

PDE Formulation

As previously noted, the equations to solve for the velocity potential, Φ, and the fluid density, ρ, are:

$$\frac{\partial \Phi}{\partial t} + \left(\frac{|\nabla \Phi|^2}{2} + \frac{\gamma}{\gamma - 1} \frac{\rho^{\gamma - 1} p_0}{\rho_0^{\gamma}} + \Psi \right) = \frac{v_0^2}{2} + \frac{\gamma p_0}{(\gamma - 1)\rho_0} + \Psi_0$$

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

Here 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 0 signifies reference quantities that apply at a specific point or surface. Thus, p_0 is a reference pressure, ρ_0 is a reference density, v_0 is a reference velocity, and Ψ_0 is a reference force potential.

Application Mode Properties

In addition to the standard properties for finite-element type and weak constraints, you can specify the analysis type in the **Application Mode Properties** dialog box.

PROPERTY	VALUE	DESCRIPTION
Analysis type	Static	Static analysis
	Transient	Transient analysis

Application Scalar Variables

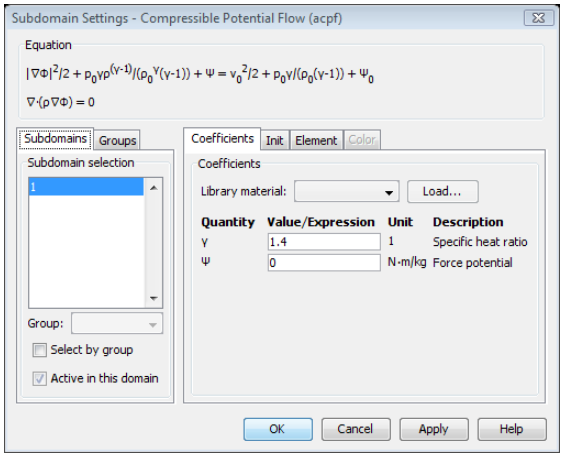
The application scalar variables are given in the following table.

NAME	SYMBOL	DEFAULT	UNIT	DESCRIPTION
p0	P_0	1e5	Pa	Reference pressure
rho0	ρ_0	1.2	kg/m^3	Reference density

NAME	SYMBOL	DEFAULT	UNIT	DESCRIPTION
v0	v_0	0	m/s	Reference velocity
psi0	Ψ_0	0	N	Reference energy level

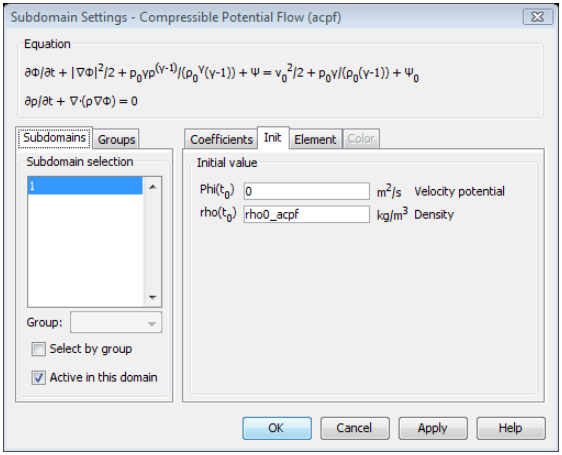
Subdomain Settings

On the **Coefficients** page of the **Subdomain Settings** dialog box you can supply values for the specific heat ratio, γ, and the force potential, Ψ. By default, $\gamma = 1.4$—the value appropriate for a diatomic gas—and $\Psi = 0$.



For transient analysis, the solver needs initial values at $t = t_0$ for the velocity potential, Φ, and the fluid density, ρ. These can be constants or functions of the spatial coordinates. The default settings are $\Phi(t_0) = 0$ and $\rho(t_0) = \rho_{0,\text{acpf}}$. You can specify different initial conditions on the **Init** page.

The **Init** page is available also for static analysis. In this case, you can supply values of Φ and ρ to serve as starting values for the solvers.



Boundary Settings

In the Compressible Potential Flow application mode, you have three boundary conditions to choose from in the **Boundary Settings** dialog box:

- **Slip/Symmetry**: The natural condition at a boundary impervious to the flow is that the velocity normal to the boundary is zero. By multiplying with the density, this condition can be alternatively be expressed as a vanishing mass flow through the boundary:

  $$\mathbf{n} \cdot \rho \nabla \Phi = 0$$

  Thus, you can also use this condition on boundaries where symmetry implies that the flow is tangential to the boundary.

- **Normal flow**: If the flow is normal to the boundary, 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:

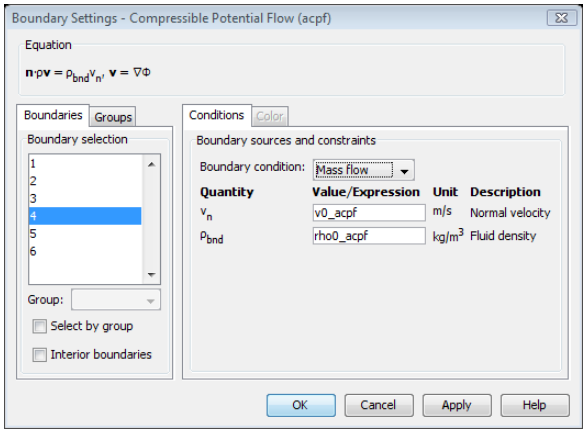
  $$\phi = 0$$

Note: Setting the **Normal flow** condition on two or more disjoint boundaries can give the wrong physics unless symmetry implies that the velocity potential is equal on the boundaries in question.

- **Mass flow**: The mass flow through a boundary is given by the product of two variables: the normal velocity, v_n, and the density at the boundary, ρ_{bnd}:

$$\mathbf{n} \cdot \rho \nabla \Phi = v_n \rho_{bnd}$$

You specify a boundary condition of this kind by supplying the values of v_n and ρ_{bnd} in their respective edit fields on the **Conditions** page. By default, they are set to the application scalar variables `v0_acpf` and `rho0_acpf`, respectively.



Application Mode Variables

The following variables are available for postprocessing and for use in equations and boundary conditions at the subdomain level:

NAME	SYMBOL	DESCRIPTION	EXPRESSION
vx*i*	v_i	Velocity in x_i direction	$\partial\phi/\partial x_i$
fx*i*	f_i	External force, x_i component	$\rho\partial\Psi/\partial x_i$
c0	c_0	Reference speed of sound	$\sqrt{\gamma p_0/\rho_0}$

NAME	SYMBOL	DESCRIPTION	EXPRESSION
cs	c_s	Speed of sound	$\sqrt{\gamma p/\rho}$
p	p	Pressure	$p_0(\rho/\rho_0)^\gamma$
normv	$\|\mathbf{v}\|$	velocity norm	$\sqrt{\mathbf{v}\cdot\mathbf{v}}$
normf	$\|\mathbf{f}\|$	External force, norm	$\sqrt{\mathbf{f}\cdot\mathbf{f}}$

Aeroacoustics with Flow

The application mode Aeroacoustics with Flow is simply a combination of the two application modes Aeroacoustics and Compressible Potential Flow with certain predefined couplings. Specifically, the background field velocity variables of the Aeroacoustics application mode are by default set equal to the flow velocity of the Compressible Potential Flow application mode.

An Example Model—Doppler Shift

Introduction

You can notice the Doppler effect when an ambulance or a fire engine passes by with its sirens blaring. The siren's pitch suddenly drops the moment the ambulance starts to move away from you. Another effect you can notice, is how the siren's sound suddenly becomes markedly quieter as soon as the ambulance passes.

In this example, the observer and air are at rest while the sound source, the ambulance, moves with the speed V. This gives the same effect as if the sound source were at rest and the observer and air were moving at the same speed but in the opposite direction.

Model Definition

This is an axisymmetric problem with a point source at rest at the origin, $(r, z) = (0, 0)$, emitting spherical sound waves with the frequency $f = 100$ Hz. The surrounding air moves at $V = 33$ m/s (roughly 120 km/h or 75 miles/hour) in the negative z direction. With this setup, the rz-plane is the horizontal plane at the level of the source and the observer, and the effects of reflection in the ground are neglected.

Assume, furthermore, that the observer stands 1 m from where the ambulance passes by. In the model geometry, this situation amounts to the observer moving with the flow along the line $r = 1$.

The boundary conditions are absorbing because there is no physical boundary around the source. Model this using perfectly matched layers.

Results and Discussion

The acoustic wavelength decreases for a wavefront moving in the opposite direction of the airflow. This situation corresponds to the approach stage of the ambulance, which, for the stationary observer on the ground, implies a perceived frequency that is higher than the nominal source frequency, f. Conversely, the wavelength increases and the perceived frequency decreases when the acoustic wave moves with the flow during the departing stage. These two stages correspond, respectively, to the left and right halves of Figure 6-1, in which the horizontal-axis coordinate is chosen to correspond to time.

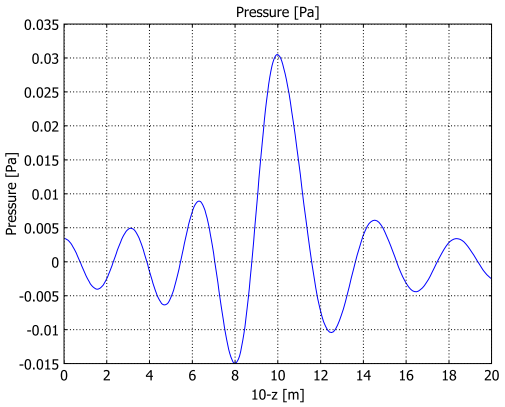


Figure 6-1: The pressure distribution along the observer's path.

A careful inspection of Figure 6-1 also shows that the amplitude drops off at a faster rate to the right of the center than to the left. This effect is more clearly visible in Figure 6-2 and Figure 6-3, which both show the sound pressure level.

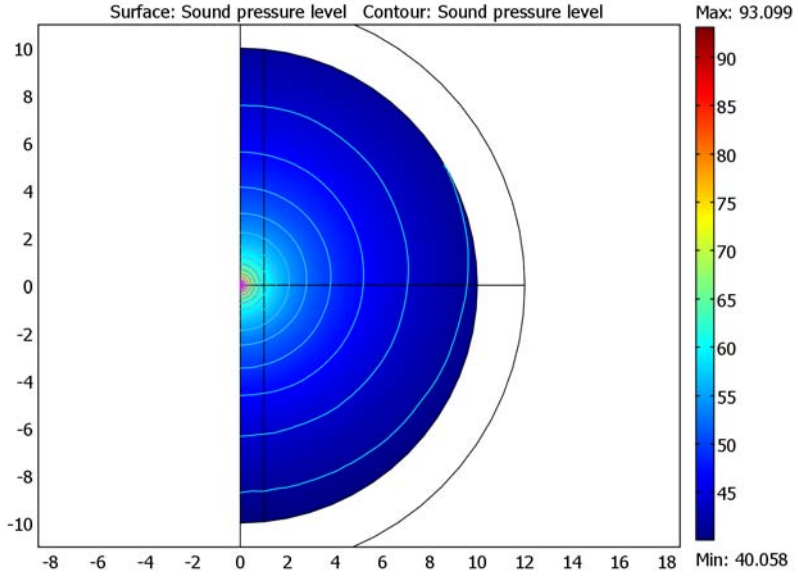


Figure 6-2: Sound pressure level around the point source.

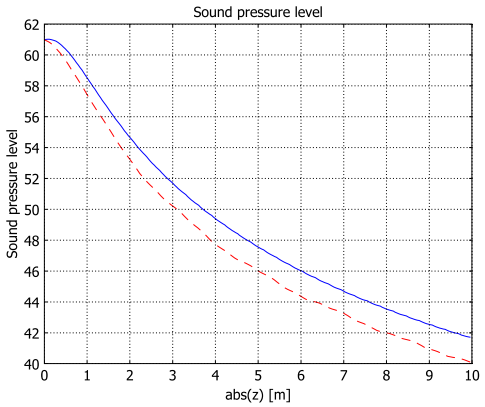


Figure 6-3: Sound pressure level at the observer's position during the ambulance's approach (solid, blue line) and departing (dashed, red line) vs. distance from the position $(r, z) = (1\text{ m}, 0)$ *(where the observer-ambulance distance is the smallest).*

Model Library path: `Acoustics Module/Tutorial Models/doppler_shift`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Start COMSOL Multiphysics.

2 In the **Model Navigator**, select **Axial symmetry (2D)** from the **Space dimension** list.

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

4 Click **OK**.

OPTIONS

Constants

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

2 Define a constant according to the following table (the description is optional):

NAME	EXPRESSION	DESCRIPTION
V	33[m/s]	Ambulance speed

3 Click **OK**.

GEOMETRY MODELING

1 Select **Draw>Specify Objects>Circle**. In the resulting dialog box, set the **Radius** to 10. When done, click **OK**.

2 Draw a new circle with a **Radius** of 12. When done, click **OK**

3 Select **Draw>Specify Objects>Rectangle**. In the dialog box that appears, specify the following values. When done, click **OK**

PROPERTY	VALUE
Width	12
Height	24
Position, z	-12

4 Copy the rectangle by pressing Ctrl+C.

5 Select the rectangle and one of the circles. Click the **Intersection** button on the Draw toolbar.

6 Paste the copy of the rectangle with no displacement.

7 Select the rectangle and the other circle. Click the **Intersection** button.

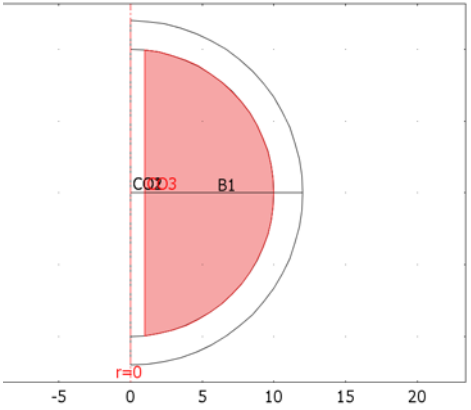
8 Click the **Line** button on the Draw toolbar. Draw a line from $r = 0, z = 0$ to $r = 12$, $z = 0$.

9 Select **Draw>Specify Objects>Circle**. Set the **Radius** to 10, then click **OK**.

10 Draw a rectangle with the following properties:

PROPERTY	VALUE
Width	9
Height	20
Position, r	1
Position, z	-10

11 Select the circle and the rectangle. Click the **Intersection** button on the Draw menu.



PHYSICS SETTINGS

Subdomain Settings

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

2 Select all the subdomains, then set **V** to `-V`.

3 Select Subdomains 1 and 4 only.

4 On the **PML** page, set the **PML type** to **Spherical**.

5 Select the **Absorbing in radial dir.** check box.

6 Click **OK** to accept the default PML parameter values and close the dialog box.

Boundary Conditions

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

2 Select Boundaries 1–3 and 5. Set the **Boundary condition** to **Axial symmetry**.

For the boundaries at $r = 12$ m, you can leave the default (sound-hard wall) boundary condition because the resulting reflected waves are effectively completely attenuated before reaching the physical modeling domain.

3 Click **OK**.

Point Settings

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

2 Click the **Point at r=0** option button.

3 Select Point 3 and set the **Mass flow rate** to `1e-4`. Click **OK**.

GENERATING THE MESH

To resolve the waves you need at least 10 measurement points per wavelength. Because this model uses 2nd-order elements, you need roughly 5 elements per wavelength. A wavelength of approximately 3.4 m requires mesh elements that are smaller than 0.6 m. Further, in the part of the geometry where the acoustic waves move against the flow, the mesh elements must be even smaller.

1. Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.
2. Click the **Subdomain** tab.
3. Select Subdomains 1, 2, and 5, then set the **Maximum element size** to `0.6`.
4. Select Subdomains 3, 4, and 6, then set the **Maximum element size** to `0.3`.
5. Click the **Remesh** button, then click **OK**.

COMPUTING THE SOLUTION

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

POSTPROCESSING

The solution in the PML domain is not of interest, so before plotting the results suppress these subdomains.

1. From the **Options** menu, select **Supress>Supress Subdomain**.
2. Select Subdomains 1 and 4, then click **OK**.
3. Click the **Zoom Extents** button on the Main toolbar.

First generate Figure 6-2 with the following steps.

1. Click the **Plot Parameters** button on the Main toolbar.
2. On the **Surface** page, from the **Predefined quantities** list on the **Surface Data** page, select **Sound pressure level**.
3. On the **Contour** page, select the **Contour plot** check box.
4. From the **Predefined quantities** list, select **Sound pressure level**.
5. Clear the **Color scale** check box.
6. Click **OK** to generate the plot and close the **Plot Parameters** dialog box.

Note how the contours preserve their shape when reaching the PML subdomain, showing that the PMLs provide an efficient nonreflecting boundary condition.

To generate Figure 6-1 and Figure 6-3 perform the following steps.

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

2 On the **Line/Extrusion** page, select **Pressure** from the **Predefined quantities** list in the **y-axis data** area.

3 From the **Boundary selection** list, select Boundaries 6 and 7.

4 In the **x-axis data** area, click the lower option button, then click the **Expression** button.

5 In the dialog box that appears type `10-z` in the **Expression** edit field. Click **OK**.

6 Click **Apply** to generate the plot in Figure 6-1.

7 Change the **Expression** in the **y-axis data** area to **Sound pressure level**.

8 From the **Boundary selection** list, select Boundary 7 only.

9 Click the **Expression** button in the **x-axis data** area. In the **Expression** edit field, type `abs(z)`, then click **OK** to close the **X-Axis Data** dialog box.

10 Click **Apply** to generate the solid line in Figure 6-3.

11 On the **General** page, select the **Keep current plot** check box, then return to the **Line/Extrusion** page.

12 From the **Boundary selection** list, select Boundary 6.

13 Click the **Line Settings** button. From the **Line color** list, select **Color**, and from the **Line style** list, select **Dashed line**. Click **OK** to close the **Line Settings** dialog box.

14 Click **OK** to complete Figure 6-3 and close the **Domain Plot Parameters** dialog box.

To create the figure that shows when you open the model, perform these steps:

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

2 Click the **Surface** tab.

3 On the **Surface Data** page, select **Pressure** from the **Predefined quantities** list.

4 Click the **Range** button. Clear the **Auto** check box, then set **Min** to `-0.03` and **Max** to `0.03`. Click **OK**. These settings gives a better use of the color scale in the main part of the model domain.

5 On the **Height Data** page, select the **Height data** check box and select **Pressure** from the **Predefined quantities** list.

6 On the **General** page, clear the **Contour** check box in the **Plot type** area.

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

8 Click the **Scene Light** button on the Camera toolbar and rotate the figure to any desired position.

7

Structural Mechanics Application Modes

This chapter describes the three structural-mechanics application modes that are included in the Acoustics Module in order to simplify modeling of acoustics-structure interaction:

- Solid, Stress-Strain (3D)
- Plane Strain
- Axial Symmetry, Stress-Strain

The analysis types available for these application modes are Static, Eigenfrequency, Damped eigenfrequency, Transient, and Frequency response.

The chapter begins with a brief theory review introducing the underlying concepts and equations.

Theory Background

Strain-Displacement Relationship

Under the assumption of small displacements, the normal strain components and the shear strain components are related to the deformation as follows:

$$\varepsilon_x = \frac{\partial u}{\partial x} \qquad \varepsilon_{xy} = \frac{\gamma_{xy}}{2} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)$$

$$\varepsilon_y = \frac{\partial v}{\partial y} \qquad \varepsilon_{yz} = \frac{\gamma_{yz}}{2} = \frac{1}{2}\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)$$

$$\varepsilon_z = \frac{\partial w}{\partial z} \qquad \varepsilon_{xz} = \frac{\gamma_{xz}}{2} = \frac{1}{2}\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right).$$

To express the shear strain, use either the tensor form, ε_{xy}, ε_{yz}, ε_{xz}, or the engineering form, γ_{xy}, γ_{yz}, γ_{xz}.

The symmetric strain tensor ε consists of both normal and shear strain components:

$$\varepsilon = \begin{bmatrix} \varepsilon_x & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{xy} & \varepsilon_y & \varepsilon_{yz} \\ \varepsilon_{xz} & \varepsilon_{yz} & \varepsilon_z \end{bmatrix}$$

Stress-Strain Relationship

The symmetric stress tensor σ describes stress in a material:

$$\sigma = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} \qquad \tau_{xy} = \tau_{yx} \qquad \tau_{xz} = \tau_{zx} \qquad \tau_{yz} = \tau_{zy}$$

This tensor consists of three normal stresses (σ_x, σ_y, σ_z) and six shear stresses—or, if symmetry is used, three: (τ_{xy}, τ_{yz}, τ_{xz}).

LINEAR ELASTIC MATERIAL

The stress-strain relationship—or the *constitutive equation*—for linear conditions reads:

$$\sigma = D\varepsilon$$

where D is the 6-by-6 elasticity matrix, and the stress and the strain are both given in column vector form:

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix}$$

Note: In the following descriptions, σ and ε denote either the stress and strain vectors or the corresponding tensors depending on the circumstances.

The elasticity matrix D—or its more basic inverse, the flexibility (compliance) matrix is defined by

$$D^{-1} = \frac{1}{E}\begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\ 0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix}$$

where E is the modulus of elasticity or *Young's modulus*, and ν is *Poisson's ratio*, which defines the contraction in the perpendicular direction. Inverting D^{-1} results in the elasticity matrix

$$D = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix}$$

Implementation

COMSOL Multiphysics' implementation of the equations in the application modes for structural analysis is based on the principle of virtual work expressed in stress and strain components.

The total stored energy, W, for a linear material from external and internal strains and loads equals

$$W = \int_V \left(\frac{1}{2}(-\varepsilon_x\sigma_x - \varepsilon_y\sigma_y - \varepsilon_z\sigma_z - 2\varepsilon_{xy}\tau_{xy} - 2\varepsilon_{yz}\tau_{yz} - 2\varepsilon_{xz}\tau_{xz}) + \mathbf{u}^t\mathbf{F}_V\right)dv + \int_S \mathbf{u}^t\mathbf{F}_S ds + \int_L \mathbf{u}^t\mathbf{F}_L dl + \sum_p \mathbf{U}^t\mathbf{F}_P$$

The principle of virtual works states that, for any virtual displacement, the total work from internal strains is the negative of the work from external loads, so that their sum equals zero:

$$\delta W = 0$$

To derive the expression for the variation of W, differentiate symbolically to obtain the expression

$$\delta W = \int_V (-\varepsilon_{x\text{test}}\sigma_x - \varepsilon_{y\text{test}}\sigma_y - \varepsilon_{z\text{test}}\sigma_z - 2\varepsilon_{xy\text{test}}\tau_{xy} - 2\varepsilon_{yz\text{test}}\tau_{yz} - 2\varepsilon_{xz\text{test}}\tau_{xz} + \mathbf{u}_{\text{test}}^t\mathbf{F}_V)dv + \int_S \mathbf{u}_{\text{test}}^t\mathbf{F}_S ds + \int_L \mathbf{u}_{\text{test}}^t\mathbf{F}_L dl + \mathbf{U}_{\text{test}}^t\mathbf{F}_P$$

SETTING UP EQUATIONS FOR DIFFERENT ANALYSES

All structural application modes in the Acoustics Module support static, eigenfrequency, damped eigenfrequency, transient, and frequency-response parametric analyses. Each type solves a different equation or employs a different solver. You control this choice with the **Analysis type** list that appear in the **Application Mode Properties** dialog box for the corresponding application mode.

Static Analysis

COMSOL Multiphysics' implementation is based on the stress and strain variables. The normal and shear strain variables depend on the displacement derivatives (described in general 3D terms in the section "Strain-Displacement Relationship" on page 144); the normal and shear stress variables depend on the strains (described in general 3D terms in the section"Stress-Strain Relationship" on page 144).

Using the shear and stress variables you can express the principle of virtual work as

$$\delta W = \int_V (-\varepsilon_{x\text{test}}\sigma_x - \varepsilon_{y\text{test}}\sigma_y - \varepsilon_{z\text{test}}\sigma_z - 2\varepsilon_{xy\text{test}}\tau_{xy} - 2\varepsilon_{yz\text{test}}\tau_{yz} - 2\varepsilon_{xz\text{test}}\tau_{xz} + \mathbf{u}_{\text{test}}^t \mathbf{F}_V) dv + \int_S \mathbf{u}_{\text{test}}^t \mathbf{F}_S ds + \int_L \mathbf{u}_{\text{test}}^t \mathbf{F}_L dl + \sum_p U_{\text{test}}^t \mathbf{F}_P = 0$$

Transient Analysis

For transient problems, Newton's second law

$$\rho \frac{\partial^2 \mathbf{u}}{\partial t^2} - \nabla \cdot c\nabla \mathbf{u} = \mathbf{F}$$

defines the equation of motion with no damping. To include viscous damping, COMSOL Multiphysics uses Rayleigh's model; see page 149 for further details.

Frequency Response Analysis

You specify harmonic loads with three components:

- the amplitude value, F_x
- the amplitude factor, $F_{x\text{Amp}}$ (a dimensionless factor; the default value is 1)
- the phase, $F_{x\text{Ph}}$

To derive the equations for the steady-state response from harmonic excitation loads

$$F_{x\text{freq}} = F_x F_{x\text{Amp}}(f) \cdot \cos\left(\omega t + F_{x\text{Ph}}(f)\frac{\pi}{180}\right)$$

$$\mathbf{F}_{\text{freq}} = \begin{bmatrix} F_{x\text{freq}} \\ F_{y\text{freq}} \\ F_{z\text{freq}} \end{bmatrix},$$

assume a harmonic response with the same angular frequency as the excitation load:

$$u = u_{\text{amp}} \cos(\omega t + \phi_u)$$

$$\mathbf{u} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}$$

You can also describe this relationship using complex notation

$$u = \text{Re}(u_{\text{amp}} e^{j\phi_u} e^{j\omega t}) = \text{Re}(\tilde{u} e^{j\omega t}) \text{ where } \tilde{u} = u_{\text{amp}} e^{j\phi_u}$$

$$\mathbf{u} = \text{Re}(\tilde{\mathbf{u}} e^{j\omega t})$$

$$F_{x\text{freq}} = \text{Re}\left(F_x F_{x\text{Amp}}(\omega) e^{jF_{x\text{Ph}}(f)\frac{\pi}{180}} e^{j\omega t}\right) = \text{Re}(\tilde{F}_x e^{j\omega t})$$

where

$$\tilde{F}_x = F_x F_{x\text{Amp}}(f) e^{jF_{x\text{Ph}}(f)\frac{\pi}{180}}$$

$$\tilde{\mathbf{F}} = \begin{bmatrix} \tilde{F}_x \\ \tilde{F}_y \\ \tilde{F}_z \end{bmatrix}$$

In addition to Rayleigh damping, COMSOL Multiphysics supports *loss factor damping* for frequency response analysis; see page 150 for further details.

Eigenfrequency Analysis

The eigenfrequency equations are derived by assuming a harmonic displacement field, similar as for the frequency-response formulation. The difference is that you introduce a new variable $j\omega$ explicitly expressed in the eigenvalue.

$$j\omega = -\lambda$$

The eigenfrequency, f is then derived from $j\omega$ as

$$f = \left|\frac{\mathrm{Im}(j\omega)}{2\pi}\right| .$$

In the eigenfrequency analysis no damping is added to the equations.

Damped Eigenfrequency Analysis

This analysis is similar to the eigenfrequency analysis except that viscous damping terms are added to the equation using the Rayleigh damping model. In addition to the eigenfrequency you can study the quality factor, Q, and the decay factor, δ, of a model:

$$Q = \frac{\mathrm{Im}(\lambda)}{2\mathrm{Re}(\lambda)} \qquad \delta = \mathrm{Re}(\lambda)$$

DAMPING MODELS

The structural mechanics application modes that come with the Acoustics Module offer two predefined damping models: Rayleigh damping and loss factor damping.

Rayleigh Damping

To model viscous damping, COMSOL Multiphysics uses *Rayleigh damping*, where you specify two damping coefficients. As an example, consider a system with a single degree of freedom. The equation of motion for such a system with viscous damping is

$$m\frac{d^2u}{dt^2} + \xi\frac{du}{dt} + ku = f(t) .$$

In the Rayleigh damping model you express the damping parameter ξ in terms of the mass m and the stiffness k as

$$\xi = \alpha_{dM}m + \beta_{dK}k .$$

The Rayleigh damping proportional to mass and stiffness is added to the static weak term.

A complication with the Rayleigh damping model is to obtain good values for the damping parameters. A more physical damping measure is the damping ratio, the ratio between actual and critical damping, often expressed as a damping factor in percentage of the critical damping. You can find commonly used values of damping factors in the literature.

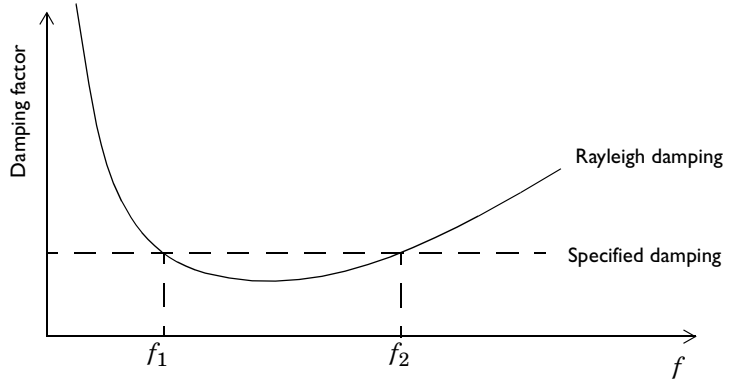
It is possible to transform damping factors to Rayleigh damping parameters. The damping factor, ξ, for a specified pairs of Rayleigh parameters, α_{dM} and β_{dK}, at a frequency, f, is

$$\xi = \frac{1}{2}\left(\frac{\alpha_{dM}}{2\pi f} + \beta_{dK} 2\pi f\right).$$

Using this relationship at two frequencies, f_1 and f_2, with different damping factors, ξ_1 and ξ_2, results in an equation system that can be solved for α_{dM} and β_{dK}:

$$\begin{bmatrix} \frac{1}{4\pi f_1} & \pi f_1 \\ \frac{1}{4\pi f_2} & \pi f_2 \end{bmatrix} \begin{bmatrix} \alpha_{dM} \\ \beta_{dK} \end{bmatrix} = \begin{bmatrix} \xi_1 \\ \xi_2 \end{bmatrix}.$$

Using the same damping factors, $\xi_1 = \xi_2$, does not result in a constant damping factor inside the interval $f_1 < f < f_2$. It can be shown that the damping factor is lower inside the interval, as the following figure shows.



Loss Factor Damping

Loss factor damping (sometimes referred to as material or structural damping) applies to viscoelastic materials modeled in the frequency domain. The complex modulus

$G^*(\omega)$ is the frequency-domain representation of the stress relaxation function of a viscoelastic material. It is defined as

$$G^* = G' + jG'' = (1 + j\eta)G'$$

where G' is the storage modulus, G'' is the loss modulus, and their ratio $\eta = G'/G''$ is the *loss factor*. The term G' defines the amount of stored energy for the applied strain, whereas G'' defines the amount of energy dissipated as heat. Both G' and G'' (and thus also η) can be frequency dependent.

In COMSOL Multiphysics, the loss information appears as a multiplier of the total strain in the stress-strain relationship:

$$\sigma = D((1 + j\eta)\varepsilon - \varepsilon_{\text{th}} - \varepsilon_0) + \sigma_0 .$$

For hyperelastic materials, the loss information appears as a multiplier in the first Piola-Kirchhoff stress, P:

$$P = (1 + j\eta)\frac{\partial W_{\text{hyp}}}{\partial \nabla \mathbf{u}}$$

Application Mode Overview

The following table lists the structural-mechanics application modes available in the Acoustics Module. For a detailed description of any of them, refer to the corresponding section on the page listed in the table.

The column for the dependent variables shows the field variables that formulate the PDEs or weak-form equations. Depending on the engineering assumptions, these variables might be a subset of the displacements u, v, and w. For axisymmetric simulations, COMSOL Multiphysics uses a variable transformation to avoid a singularity at the axis.

For each application mode the table indicates the availability of various analysis capabilities.

Finally the table lists the domains where you can specify application mode data such as material properties, loads, and constraints. Note that edges exist only in 3D geometries.

APPLICATION MODE	DEFAULT NAME	PAGE	DEPENDENT VARIABLES	ANALYSIS CAPABILITIES				DOMAINS			
		DESCRIPTION		STATIC	EIGENFREQUENCY	TRANSIENT	FREQUENCY RESPONSE	POINT	EDGE	BOUNDARY	SUBDOMAIN
Solid, Stress-Strain	acsld	174	u,v,w	√	√	√	√	√	√	√	√
Plane Strain	acpn	177	u,v	√	√	√	√	√		√	√
Axial Symmetry, Stress-Strain	acaxi	181	uor,v	√	√	√	√	√		√	√

Common Application Mode Features

This section describes the most important modeling steps and functionality common to all structural-mechanics application modes in the Acoustics Module.

It contains the following subsections:

- Coordinate Systems
- Application mode properties
- Scalar variables
- Material
- Constraints
- Loads
- Damping
- Perfectly matched layers (PMLs)

Coordinate Systems

Using different coordinate systems can be convenient when specifying loads, constraints, and anisotropic materials, and when postprocessing the results. The Acoustics Module provides the following coordinate systems:

- A global Cartesian coordinate system, where the geometry is created in 3D (x, y, z).
- A local geometrical coordinate system on 2D boundaries (t, n) and on 3D faces (t_1, t_2, n).
- User-defined coordinate systems.

To specify which coordinate system to use in a particular context, select it from the **Coordinate system** list on the **Constraint** or **Load** page.

THE GLOBAL COORDINATE SYSTEM

You can use the global coordinate system in all application modes to specify loads and constraints on all domain levels: points, edges, faces, and subdomains. It is the default

setting for loads and constraints in all application modes on all domain levels. The default names for the space coordinates are the following for the different geometries:

GEOMETRY	DEFAULT NAME OF SPACE COORDINATES
2D	$x\ y\ z$
3D	$x\ y\ z$
Axial symmetry (2D)	$r\ \varphi\ z$

It is possible to change the names of the space coordinates when creating a geometry from the **Model navigator**, see "Creating Cartesian and Cylindrical Coordinate Systems" on page 27 in *COMSOL Multiphysics User's Guide* for details.

LOCAL GEOMETRICAL COORDINATE SYSTEMS

Boundaries in 2D and 3D have geometric variables describing the parametrization of the geometry defined on them. These variables contain directions that define a local coordinate system that you can use when specifying loads and constraints.

In 2D, the local geometrical coordinate system (t, n) represents the directions tangential and normal to the boundary. For interior boundaries and free edges this coordinate system is right-oriented. For exterior boundaries the normal is always directed out from the domain.

In 3D, the local geometrical coordinate system (t_1, t_2, n) represents two tangential directions and one normal direction. t_1 and t_2 depend on the parametrization of the geometry. For interior boundaries and free faces this coordinate system is right-oriented but not always orthogonal. For exterior boundaries the normal is always directed out from the domain. Common applications for this coordinate system include specifying pressure or normal displacement on a surface.

Note: t_1 and t_2 depend on how the geometry was created and are usually perpendicular to each other.

Read more about this topic in "Geometric Variables" on page 165 in the *COMSOL Multiphysics User's Guide*.

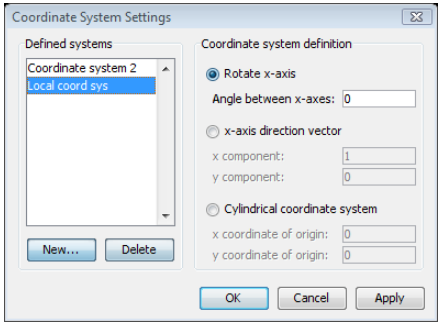
USER-DEFINED COORDINATE SYSTEMS

User-defined coordinate systems can be applied at all domain levels in all application modes. For the continuum application modes, they can define orthotropic and

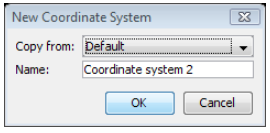
anisotropic material properties in a coordinate system other than the global Cartesian system.

Create a user-defined coordinate system by choosing **Options>Coordinate Systems**, thereby opening the **Coordinate Systems Settings** dialog box. Depending on the active geometry, the software creates a 2D or 3D coordinate system.

2D Geometry



The **New** button opens the **New Coordinate System** dialog box.

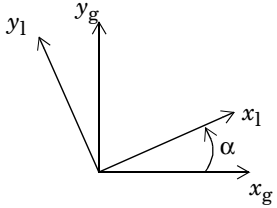


In the **Copy from** list you select from which existing coordinate system you want to copy the coordinate-system settings.

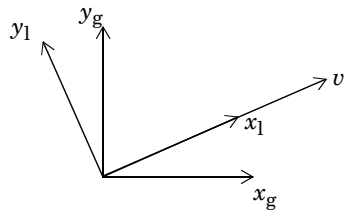
In the **Name** edit field you enter the name of the coordinate system, and it is the name that appears in all coordinate-system lists.

The software creates a coordinate system in one of three ways, which you control with option buttons:

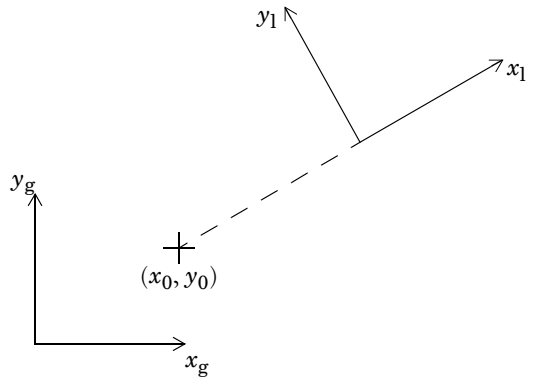
- **Rotate x-axis**: The local x_1-axis direction is specified by an angle (α) between the global and local x-axes.



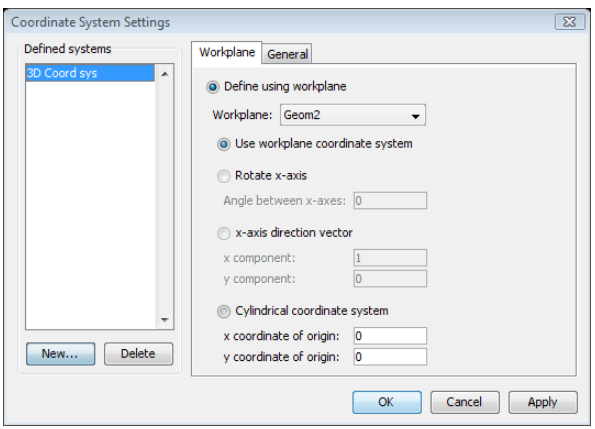
- **x-axis direction vector**: The local x_l-axis direction is specified by a direction vector v.



- **Cylindrical coordinate system**: A local cylindrical coordinate system (x_l, y_l) with origin at (x_0, y_0) is specified.



3D Geometry



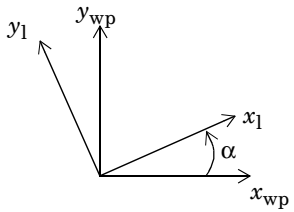
The **New** button works in the same way as for the 2D geometry case.

The software defines the coordinate system in one of two ways, which you control with the **Define using work plane** and **Define using global coordinates** option buttons.

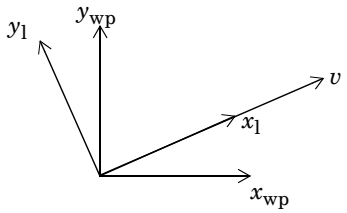
Define using work plane is enabled when a least one work plane/2D geometry exists. Select the work plane on which to base the local coordinate system from the **Work plane** list.

Four options are available, which you control with option buttons:

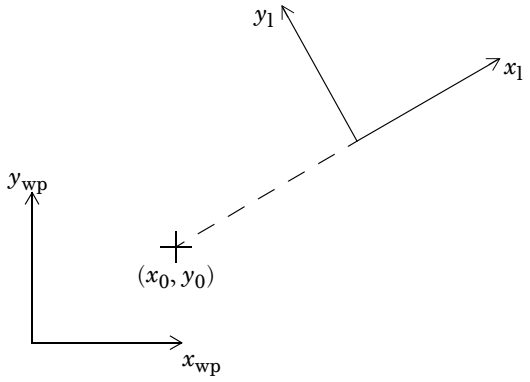
- **Use work plane coordinate system**: The local coordinate system is the same as the work plane. You control the definition of the work plane by going to the **Draw** menu and opening the **Work-Plane Settings** dialog box. Get details about the creation of work planes in "Creating and Using 2D Work Planes" on page 59 in the *COMSOL Multiphysics User's Guide*.
- **Rotate x-axis** The local x_1-axis direction is specified by an angle (α) between the work planes x_{wp}-axis and the local x_1-axis.



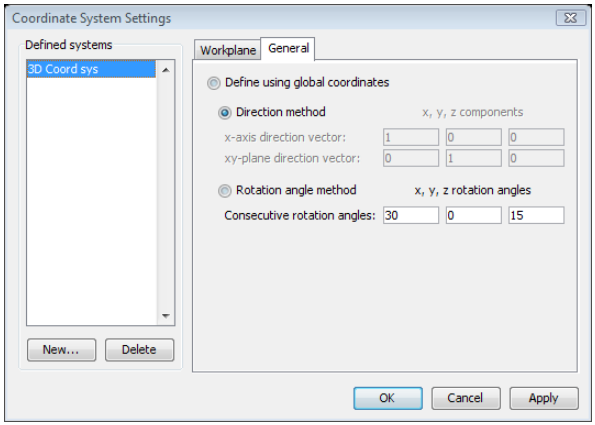
- **x-axis direction vector**: The local x_l-axis direction is specified by a direction vector v.



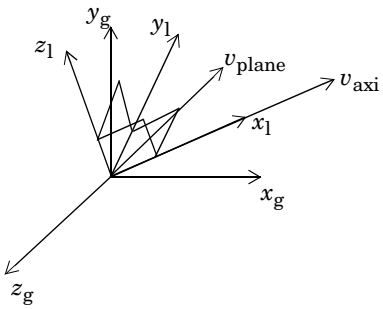
- **Cylindrical coordinate system**: A local cylindrical coordinate system (x_l, y_l) with origin at (x_0, y_0) in the work plane coordinates is specified.



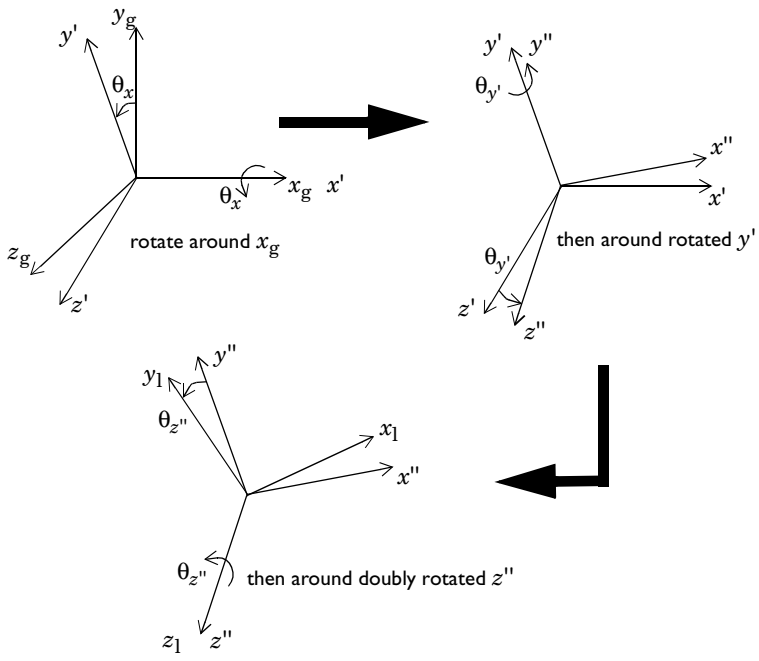
When you select **Define using global coordinates**, there are two different options available, which you control with option buttons:



- **Direction method**: The local x_1-axis direction is specified by a direction vector v_{axi}. The local x_1y_1-plane is specified by a direction vector v_{plane}, which is a vector in the local x_1y_1-plane.

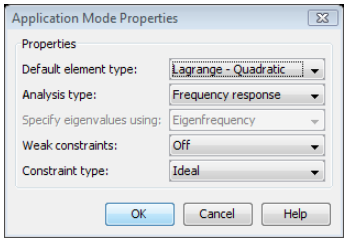


- **Rotation angle method**: The local coordinate system (x_1, y_1, z_1) is specified by three consecutive rotation angles θ_x, $\theta_{y'}$, and $\theta_{z''}$.


y_g
y'
θ_x
x_g x'
θ_x
z_g
z'
rotate around x_g
y' y''
$\theta_{y'}$
x''
x'
$\theta_{y'}$
then around rotated y'
z'
z''
y_l y''
$\theta_{z''}$
x_l
x''
$\theta_{z''}$
then around doubly rotated z''
z_l z''

Properties

To set or examine application mode properties go to the **Physics>Properties** menu to open the **Application Mode Properties** dialog box. Here you control various global settings for the model:



Application Mode Properties dialog box for the structural mechanics application modes.

- **Default element type**: The selected finite element type that makes up the discretized finite element model is the default on all new subdomains, and the choice does not affect subdomains already created. Available elements are:
  - **Lagrange - Linear**
  - **Lagrange - Quadratic**
  - **Lagrange - Cubic**
  - **Lagrange - Quartic**
  - **Lagrange - Quintic**
- **Analysis type**: This list shows the various analyses you can perform; the default is **Frequency response**. Your choice affects both the equations and which solver COMSOL Multiphysics uses when you set the **Auto select solver** option in the **Solver Parameters** dialog box.

ANALYSIS TYPE	COMSOL MULTIPHYSICS SOLVER
Static	Stationary
Eigenfrequency	Eigenvalue
Damped eigenfrequency	Eigenvalue
Time dependent (Transient)	Time dependent
Frequency response	Parametric

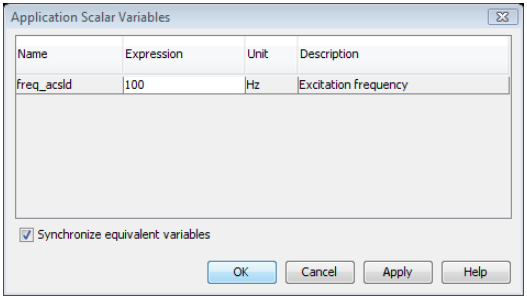
- **Specify eigenvalues using**: This list controls how to work with eigenmode analyses. Here you should specify **Eigenvalue** or **Eigenfrequency**; this property is enabled only for eigenfrequency analyses.

- **Eigenfrequency**; this property is enabled only for eigenfrequency and damped eigenfrequency analyses.
- **Weak constraints**: This list controls how to handle weak constraints; available options are **On** and **Off**. Select weak constraints for accurate computation of reaction forces. When you enable weak constraints, all constraints are of that type by default, but you can change this setting for individual domains.
- **Constraint type**: Constraints can be ideal or nonideal (see "Ideal vs. Non-Ideal Constraints" on page 301 in the *COMSOL Multiphysics Modeling Guide*).

Scalar Variables

There are two scalar variables:

- The excitation frequency, `freq`—applicable only to frequency-response analysis.
- The complex angular frequency, `jomega`—applicable only to eigenfrequency analysis.You normally do not need to edit the complex angular frequency.



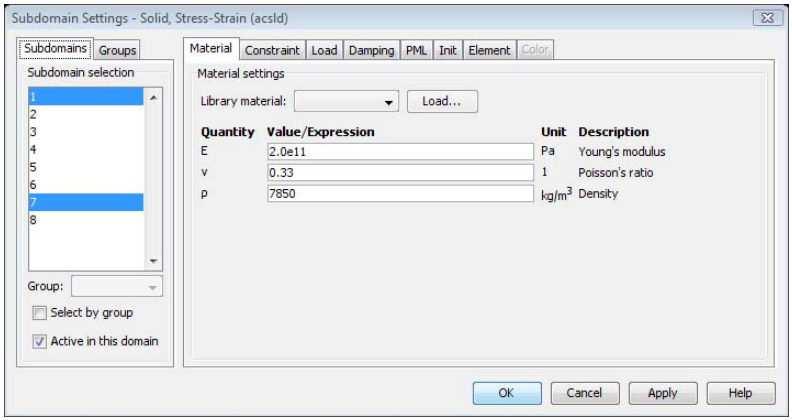
The Application Scalar Variables dialog box in a frequency-response analysis.

When you select **Frequency response** analysis, the parametric solver becomes the default solver, which makes it easy to perform a frequency sweep over several excitation frequencies in one analysis. In this case select the menu item **Solve>Solver Parameters**, and in the dialog box that appears go to the **General** page. In the **Parameter** area enter `freq_acsld` as the **Parameter name**. Any values you enter in the **Parameter values** field overrides the excitation frequency you might have entered in the **Application Scalar Variables** dialog box.

To access the excitation frequency, f, use the variable `freq`; to access the angular excitation frequency ω use `omega`.

Material

You define material properties on the **Material** page in the **Subdomain Settings** dialog box.



Material properties.

TABLE 7-1: MATERIAL PROPERTIES

PARAMETER	VARIABLE	DESCRIPTION
E	E	Young's modulus
ν	nu	Poisson's ratio
ρ	rho	Density

Now examine the various material properties in Table 7-1.

Young's modulus It defines a material's modulus of elasticity, E. For an isotropic material it is the spring stiffness in Hooke's law, which in 1D form is

$$\sigma = E\varepsilon$$

where σ is the stress and ε is the strain. An orthotropic material uses one value of Young's modulus for each direction, E_i as defined on page 144.

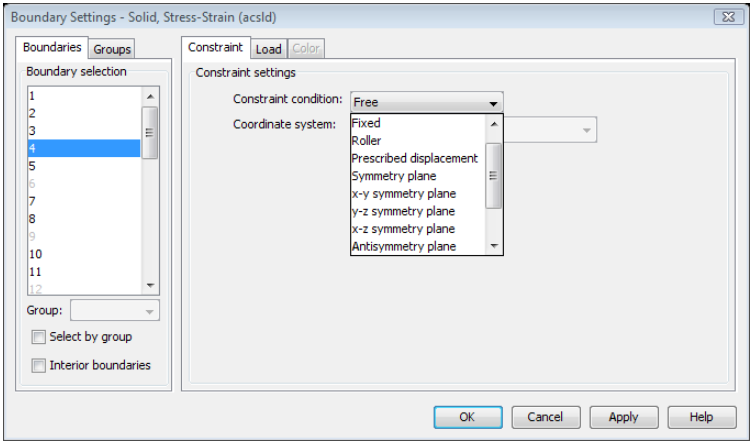
Poisson's ratio Denoted by ν, it defines the normal strain in the perpendicular direction, generated from a normal strain in the other direction and following the equation

$$\varepsilon_{\perp} = -\upsilon\varepsilon_{||}\,.$$

Density This entry specifies ρ, the material's density.

Constraints

A constraint specifies the displacement of certain parts of a structure. You can define constraints on all domain levels: points, edges, faces/boundaries, and subdomains in 3D; as well as points, boundaries, and subdomains in 2D. To control a constraint, go to the **Constraint** page of the **Subdomain Settings**, **Boundary Settings**, **Edge Settings**, and **Point Settings** dialog boxes that you open from the **Physics** menu. The following figure shows the **Boundary Settings** dialog box for the Solid, Stress-Strain application mode, but the **Constraints settings** area has the same appearance in all structural mechanics application modes.



An example of a Constraint page, taken here from the Solid, Stress-Strain application mode Boundary Settings dialog box.

Within the dialog box, the **Constraint condition** list lets you control which type of constraint you want to define. You can choose between the following options:

CONSTRAINT CONDITION	BOUNDARY	SUBDOMAIN	USE WHEN
Free	√	√	The domain has no constraint
Fixed	√	√	The displacement in the domain is fixed in all directions
Roller	√		The normal displacement is constrained
Prescribed displacement	√	√	The displacement in any direction need to be prescribed

CONSTRAINT CONDITION	BOUNDARY	SUBDOMAIN	USE WHEN
Symmetry plane	√		The boundary is a symmetry plane
xy symmetry plane	√		The selected coordinate system's xy-plane is a symmetry plane
yz symmetry plane	√		The selected coordinate system's yz-plane is a symmetry plane
xz symmetry plane	√		The selected coordinate system's xz-plane is a symmetry plane
Antisymmetry plane	√		The boundary is an antisymmetry plane
xy antisymmetry plane	√		The selected coordinate system's xy-plane is an antisymmetry plane
yz antisymmetry plane	√		The selected coordinate system's yz-plane is an antisymmetry plane
xz antisymmetry plane	√		The selected coordinate system's xz-plane is an antisymmetry plane
Prescribed velocity	√	√	The velocity in any direction need to be prescribed, available only for frequency-response analysis
Prescribed acceleration	√	√	The acceleration in any direction need to be prescribed, available only for frequency-response analysis

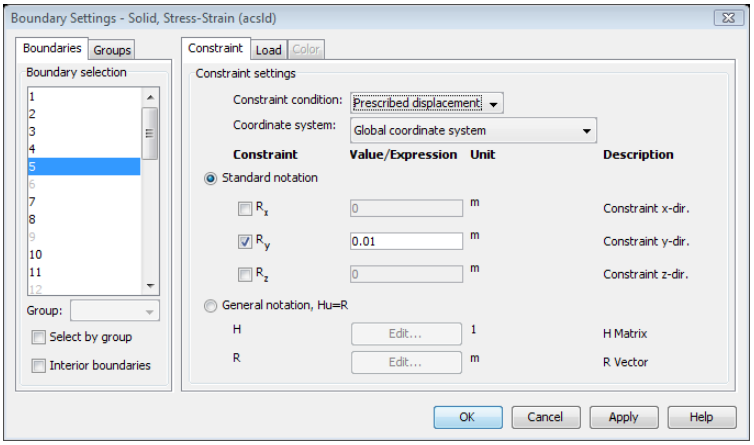
The symmetry or antisymmetry condition has the following interpretation.

CONDITION	X-DISPLACEMENT	Y-DISPLACEMENT	Z-DISPLACEMENT
xy symmetry plane			√
yz symmetry plane	√		
xz symmetry plane		√	
xy antisymmetry plane	√	√	
yz antisymmetry plane		√	√
xz antisymmetry plane	√		√

The **Coordinate system** list lets you control in which coordinate system you want the constraint defined. Available options are:

- Global coordinate system
- Tangent and normal coordinate system (available only on boundaries)
- User-defined coordinate systems if any local coordinate systems are defined.

When you select **Prescribed displacement**, a number of new options appear in the dialog box and the **Constraint** page takes on this appearance:



The Constraint page showing the Prescribed displacement options.

You can prescribe a constraint in two ways:

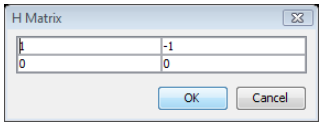
- In standard notation (select this option by clicking the **Standard notation** button) you constrain each displacement direction independently. The check boxes adjacent to the $\mathbf{R_x}$, $\mathbf{R_y}$, and $\mathbf{R_z}$ edit fields activate the constraint, whereupon you enter the value/expression of the displacement (the default value is `0`).
- General notation (select this option by clicking the **General notation, Hu=R** button) lets you specify constraints as any linear combination of displacements components. For instance, in the 2D case, use the relationship

$$H \begin{bmatrix} u \\ v \end{bmatrix} = R .$$

  Enter values for the H matrix and R vector in corresponding dialog boxes by clicking the respective **Edit** buttons. For example, to enforce the condition $u = v$, use the settings

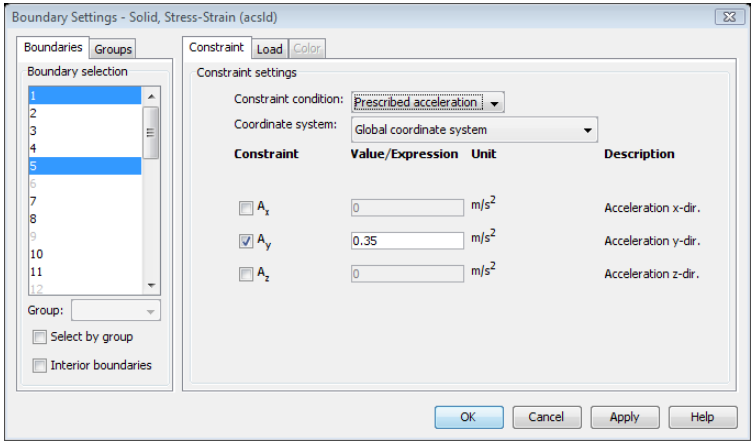
$$H = \begin{bmatrix} 1 & -1 \\ 0 & 0 \end{bmatrix}, \qquad R = \begin{bmatrix} 0 \\ 0 \end{bmatrix},$$

which force the domain to move only diagonally in the xy-plane.



The H Matrix dialog box for the example in the text.

In a frequency-response analysis you can specify not only a harmonic displacement but also a harmonic velocity or acceleration. You specify a prescribed velocity or acceleration in the same way as **Prescribed displacement** using **Standard notation** by first selecting **Prescribed velocity** or **Prescribed acceleration** in the **Constraint condition** list.

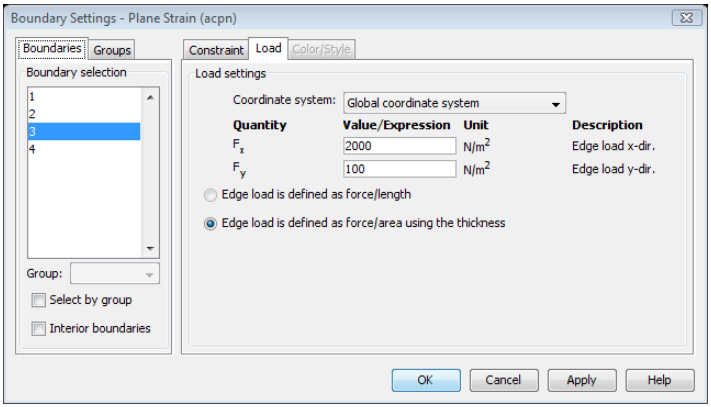


Constraint page showing the Prescribed acceleration settings.

Loads

"Load" is a general term for a force applied to a structure. In the structural application modes in the Acoustics Module you can specify loads on all domain types using the

Load page that appears in the **Subdomain Settings**, **Boundary Settings**, **Edge Settings**, and **Point Settings** dialog boxes under the **Physics** menu.



The Boundary Settings dialog box for the Plane Strain application mode shown here is representative of load pages for all domain levels in all structural mechanics application modes.

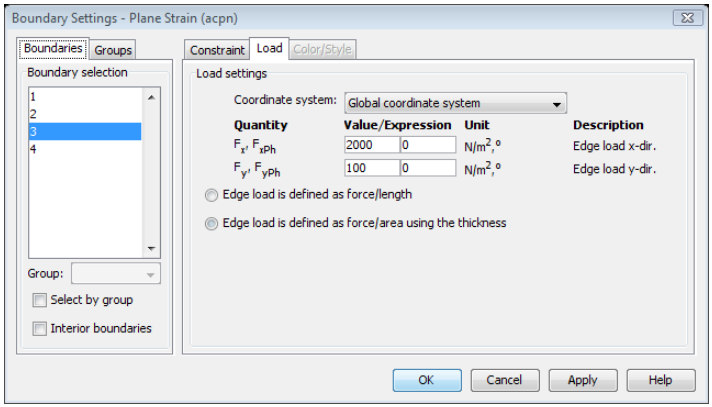
For plane strain, two option buttons allow you to choose how to specify the load using the thickness. The following table shows how to define the loads on different domains in different application modes; the entries give the SI unit in parenthesis.

APPLICATION MODE	POINT	EDGE	BOUNDARY	SUBDOMAIN
Plane Strain	force (N)		force/area (N/m^2) or force/length (N/m)	force/volume (N/m^3) or force/area (N/m^2)
Axisymmetry, Stress-Strain	total force along the circumferential (N)		force/area (N/m^2)	force/volume (N/m^3)
Solid, Stress-Strain	force (N)	force/length (N/m)	force/area (N/m^2)	force/volume (N/m^3)

Within the dialog box, the **Coordinate system** list lets you control in which coordinate system you want to define the load. Available options are:

- Global coordinate system
- Tangent and normal coordinate system (available only on boundaries)
- User-defined coordinate systems, if any local coordinate systems are defined

For a frequency-response analysis you have additional input data. To control the analysis type use the **Application Mode Properties** dialog box. When frequency response is the analysis type, the **Load** page takes on this appearance:



The Load page that appears for frequency-response analysis.

For frequency-response analysis the application mode splits the harmonic load into three parameters:

- the amplitude value, F
- the amplitude factor, F_{Amp} (a dimensionless factor; the default value is 1)
- the phase, F_{Ph}

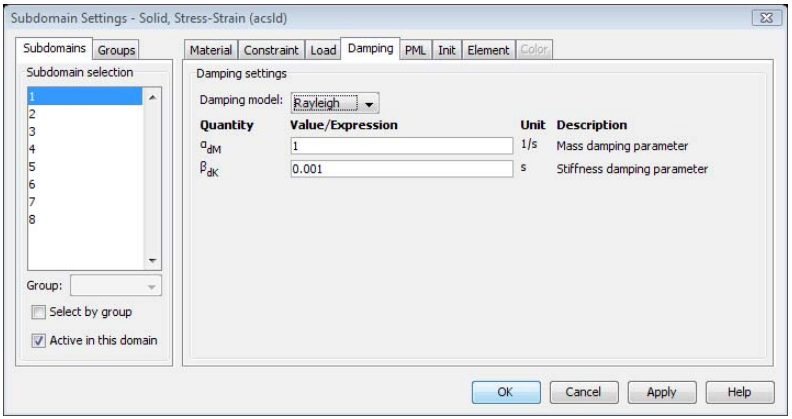
Together they define a harmonic load whose amplitude and phase shift can vary with the excitation frequency, f

$$F_{\mathrm{freq}} = F \cdot F_{\mathrm{Amp}}(f) \cdot \cos(2\pi f + F_{\mathrm{Ph}}(f)) .$$

Damping

In transient, eigenfrequency, and frequency-response analyses you can model undamped or damped problems. In the structural-mechanics application modes in the Acoustics Module you specify damping at the subdomain level using the **Damping** page that appears in the **Subdomain Settings** dialog box. From the **Damping model** list you can

select **No damping**, **Rayleigh**, or **Loss factor**, and the contents of the dialog box changes for each of these damping models.



The Damping page when Rayleigh damping is selected.

If you select **Rayleigh** as your damping model, you can specify the mass damping parameter, α_{dM}, and the stiffness damping parameter, β_{dK}. When selecting **Loss factor**, you supply the loss factor, η. For details about the underlying equations, see "Damping Models" on page 149.

The two available damping models apply to the following analysis types:

- Rayleigh damping: **Damped eigenfrequency**, **Time dependent**, and **Frequency response**
- Loss factor damping: **Frequency response**

Note: This means that if you choose, for example, time dependent analysis and loss factor damping, the software solves the model with no damping.

Table 7-2 and the following text describe the parameters that define damping:

TABLE 7-2: PARAMETERS FOR DAMPING MODELS

PARAMETER	VARIABLE	DESCRIPTION	DAMPING MODEL
α_{dM}	`alphadM`	Mass-damping parameter	Rayleigh
β_{dK}	`betadK`	Stiffness-damping parameter	Rayleigh
η	`eta`	Loss factor	Loss factor

Mass damping parameter Defines the Rayleigh damping model's mass damping, α_{dM}.

Stiffness damping parameter Defines the Rayleigh damping model's stiffness damping, β_{dK}.

Loss factor Defines the loss factor η for the loss factor damping model.

Perfectly Matched Layers (PMLs)

In frequency-response analysis of elastic waves, you can use perfectly matched layers to simulate absorbing boundary conditions. This section describes how to create Cartesian, cylindrical, and spherical PMLs for elastic waves. For an account of elastic waves in solids, see Chapters 4 and 5 of Ref. 1. For background information about PMLs in elastodynamics, see Ref. 2. The section "Perfectly Matched Layers (PMLs)" on page 37 describes PMLs for acoustic pressure waves.

PML IMPLEMENTATION

For a PML that absorbs waves in the coordinate direction ξ, the Structural Mechanics Module uses the following coordinate transformation inside the PML:

$$\xi' = \mathrm{sign}(\xi - \xi_0)|\xi - \xi_0|^n \frac{L}{\delta\xi^n}(1 - i) \tag{7-1}$$

The scaled PML width, L; the coordinate of the inner PML boundary, ξ_0; and the (actual) width of the PML, $\delta\xi$, are input parameters for each orthogonal absorbing coordinate direction.

The scaling exponent, n is an input parameter for each PML subdomain. The default value of n is 1, giving a linear scaling that works well in most cases, and the useful range is roughly between 1 and 2; increasing the exponent allows you to use fewer mesh elements to resolve wavelengths much smaller than the scaled PML width.

Usually, set L equal to one wavelength. The wavelength depends on the type of elastic wave you are considering. For example, for longitudinal (acoustic) waves, the wavelength is given by (Ref. 1)

$$\lambda = \frac{1}{f}\sqrt{\frac{(1-\nu)}{(1+\nu)(1-2\nu)}\frac{E}{\rho}}$$

where f is the frequency, E is Young's modulus, ν is Poisson's ratio and ρ is the density. If your analysis includes several wave types of different wavelengths, set L to the longest one. For this case, you can also try to set the scaling exponent, n, equal to 2.

The parameters ξ_0 and $\delta\xi$ get default settings that the software deduces from the drawn geometry and stores in so-called guess variables. You can inspect the values of the guess variables on the **Variables** page of the **Subdomain Settings - Equation System** dialog box or at the corresponding node of the **Model Tree**.

The default settings defined by the guess variables work nicely in most cases, but they might fail for PML subdomains of nonstandard shape.

SETTING UP A PML

To model an absorbing boundary using PMLs, you need an auxiliary subdomain outside the boundary. On the **PML** page in the **Subdomain Settings** dialog box, you can select different types of PMLs depending on the kind of wave you have:

PML TYPE	APPLICATION MODE	DESCRIPTION
None	all	No PML
Cartesian	Solid, Stress-Strain; Plane Strain	Absorbs waves in the specified Cartesian coordinate directions
Cylindrical	all	Absorbs cylindrical waves
Spherical	Solid, Stress-Strain; Axial Symmetry, Stress-Strain	Absorbs spherical waves
User defined	all	Define your own scaled space variables

The PML type **None** is the default. To add a PML, select one of the other types.

CARTESIAN PMLS

For Cartesian PMLs, you can choose whether use the default global coordinate system or a user-defined coordinate system to define the directions. If you want a curved coordinate system you must use the cylindrical or spherical PML type.

Select the check box for the directions in which you want the waves to be absorbed. For each of these directions, enter the scaled PML width, L in the associated edit field. Make sure all material properties are the same in the PML as in the adjacent subdomain.

CYLINDRICAL PMLS

In 2D, a cylindrical PML always absorbs waves in the radial direction. In the other dimensions, you can decide how the PML absorbs the wave: in the radial direction, the z direction, or both.

Select the directions in which you want the PML to absorb the waves and enter the scaled PML widths in those directions. To define a cylindrical PML you also need to

enter the center point of the cylindrical coordinates and, in 3D, the cylinder axis direction.

SPHERICAL PMLS

A spherical PML always absorbs waves in the radial direction. Enter the scaled PML width, L. Define the spherical coordinates by entering the center point.

USER-DEFINED PMLS

When using a PML, the algorithm scales the equation in this domain so that instead of the coordinates used in the rest of the model, the coordinates PMLx, PMLy, and PMLz appear in the equation. If you want to scale the equation in some other way than the automatic PML options provide, use a user-defined PML. In this case you enter your own **User-defined PML coordinates**.

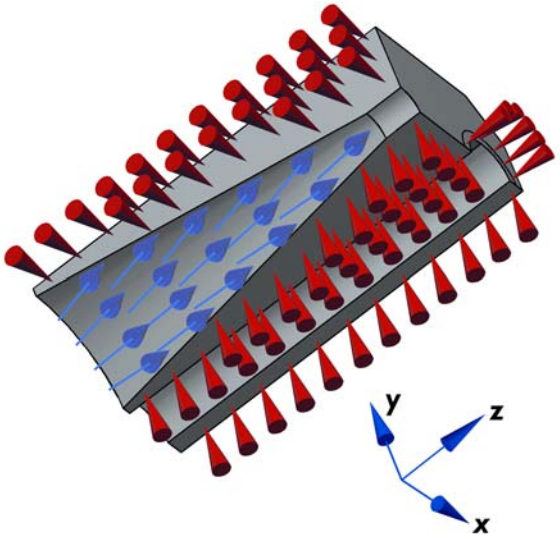
References

1. L.M. Brekhovskikh and V. Goncharov, *Mechanics of Continua and Wave Dynamics*, 2nd ed., Springer-Verlag, 1994.

2. W.C. Chew and Q.H. Liu, "Perfectly Matched Layers for Elastodynamics: A New Absorbing Boundary Condition," *J. Comp. Acoustics*, vol. 4, pp. 341–359, 1996.

Solid, Stress-Strain

This section explains how to use the Solid, Stress-Strain application mode in the Acoustics Module to analyze 3D solids.



Loads and constraints applied to a 3D solid using the Solid, Stress-Strain application mode.

Variables and Space Dimensions

The degrees of freedom (dependent variables) in this application mode are the global displacements u, v, and w in the global x, y, and z directions, respectively. If you select the mixed formulation, the application mode also includes the pressure p as a dependent variable.

Application Mode Parameters

The section "Common Application Mode Features" on page 153 defines the parameters you need in modeling loads, materials, and constraints.

Application Mode Variables

A large number of variables are available for use in expressions and postprocessing. In addition to the variables in Table 7-3, almost all application-mode parameters are available as variables. Some variables change their availability with the type of analysis, as noted in the Analysis column. For frequency-response analysis a number of additional variables are available. Furthermore, the amplitudes and phases of variables such as strains and stresses are available; to access them, append `_amp` or `_ph` to the variable name. For example:

- `sx_amp` is the amplitude of the normal stress in the x direction.
- `ex_ph` is the phase of the normal strain in the x direction.

The exception to this scheme consists of variables defined using a nonlinear operator such as `mises`, `disp`, `Tresca`, or `s1`.

Table 7-3 uses a convention where indices $i, j, \ldots$ (or `i`, `j`, ...) run over the geometry's Cartesian coordinate axes, x, y, and z. In particular, u_i (`ui`) refers to the global displacements (u, v, w). The Analysis column uses the following abbreviations:

ANALYSIS	ABBREVIATION
Static	S
Frequency response	F
Time dependent	T

TABLE 7-3: SOLID APPLICATION MODE VARIABLES

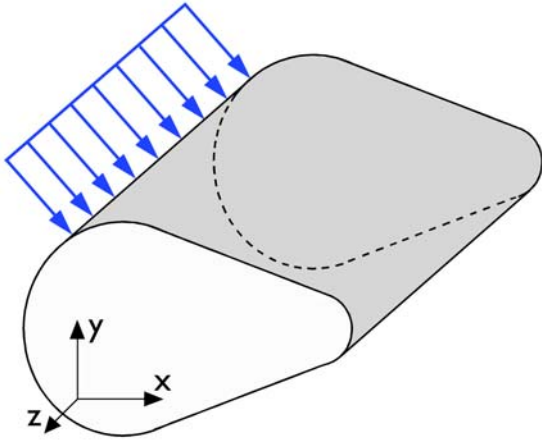
NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
u*i*	u_i	All	All	x_i displacement	u_i
u*i*t	u_{it}	T	All	x_i velocity	u_{it}
u*i*_amp	$u_{i\mathrm{amp}}$	F	All	x_i displacement amplitude	$\lvert u_i \rvert$
u*i*_ph	$u_{i\mathrm{ph}}$	F	All	x_i displacement phase	$(180°/\pi)\,\mathrm{mod}(\mathrm{angle}(u_i), 2\pi)$
u*i*_t	u_{it}	F	All	x_i velocity	$j\omega u_i$
u*i*_t_amp	$u_{it\mathrm{amp}}$	F	All	x_i velocity amplitude	$\omega u_{i\mathrm{amp}}$
u*i*_t_ph	$u_{it\mathrm{ph}}$	F	All	x_i velocity phase	$\mathrm{mod}(u_{i\mathrm{ph}} + 90°, 360°)$
u*i*_tt	u_{itt}	F	All	x_i acceleration	$-\omega^2 u_i$
u*i*_tt_amp	$u_{itt\mathrm{amp}}$	F	All	x_i acceleration amplitude	$\omega^2 \lvert u_i \rvert$

TABLE 7-3: SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
u*i*_tt_ph	$u_{itt\mathrm{ph}}$	F	All	x_i acceleration phase	$\mathrm{mod}(u_{i\mathrm{ph}} + 180°, 360°)$
disp	disp	All	All	Total displacement	$\sqrt{\sum_i (\mathrm{real}(u_i))^2}$
e*i*,e*ij*	ε_i, ε_{ij}	All	S	Strain, global coord. system	
e*i*_t, e*ij*_t	ε_{it}, ε_{ijt}	FT	S	Velocity strain, global coord system	
s*i*, s*ij*	σ_i, τ_{ij}	All	S	Cauchy stress, global coord. system	
s*i*_t, s*ij*_t	σ_{it}, τ_{ijt}	FT	S	Time derivative of Cauchy stress, global coord. system	
s*i*	σ_i	All	S	Principal stresses	
e*i*	ε_i	All	S	Principal strains	
s*i*x*j*	σ_{ixj}	All	S	Principal stress directions	
e*i*x*j*	ε_{ixj}	All	S	Principal strain directions	
tresca	σ_{tresca}	All	S	Tresca stress	$\max(\max(\lvert\sigma_1 - \sigma_2\rvert, \lvert\sigma_2 - \sigma_3\rvert), \lvert\sigma_1 - \sigma_3\rvert)$
mises	σ_{mises}	All	S	von Mises stress	
Ws	W_s	All	S	Strain energy density	Defined differently depending on material model and mixed or displacement formulation
Ta*i*	Ta_i	All	B	Surface traction (force/area) in x_i direction	Defined differently depending on large or small deformation
F*i*g	F_{ig}	All	All	Body load, face load, edge load, point load, in global x_i direction	Defined differently depending on force definition

Plane Strain

This section explains the Plane Strain application mode in the Acoustics Module; it solves for the global displacements (u, v) in the x and y directions and the pressure p (only if mixed formulation is used). The assumption that defines a state of plane strain is that the ε_z, ε_{yz}, and ε_{xz} components of the strain tensor are zero.



A geometry suitable for plane strain analysis.

Loads in the x and y directions are allowed. The loads are assumed to be constant throughout the thickness of the material, but the thickness can vary with x and y. The plane strain condition prevails in geometries whose extent is large in the z direction compared to the x and y directions, or when the z displacement is in some way restricted. One example is a long tunnel along the z-axis where it is sufficient to study a unit-depth slice in the xy-plane.

Material

An additional material parameter for plane strain is the thickness of the geometry.

PARAMETER	VARIABLE	DESCRIPTION	MATERIAL MODEL
th	thickness	The thickness of the geometry	All

Application Mode Parameters

The section "Common Application Mode Features" on page 153 defines the parameters you need in modeling including loads, the material and constraints.

Application Mode Variables

A large number of variables are available for use in expressions and for postprocessing. In addition to the variables in Table 7-4, almost all application mode parameters are available as variables. Some variables change their availability with the type of analysis, as noted in the Analysis column. For frequency-response analysis a number of additional variables are available. Furthermore, the amplitudes and phases of variables such as strains and stresses are available; to access them, append `_amp` or `_ph` to the variable name. For example:

- `sx_amp` is the amplitude of the normal stress in the x direction.
- `ex_ph` is the phase of the normal strain in the x direction.

The exception to this scheme consists of variables defined using a nonlinear operator such as `mises`, `disp`, `Tresca`, or `s1`.

Table 7-4 uses a convention where indices $i, j, \ldots$ (or `i`, `j`, ...) run over the geometry's Cartesian coordinate axes, x and y. In particular, u_i (`ui`) refers to the global displacements (u, v). The Analysis column uses the following abbreviations:

ANALYSIS	ABBREVIATION
Static	S
Frequency response	F
Time dependent	T

TABLE 7-4: PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
`ui`	u_i	All	All	x_i displacement	u_i
`uit`	u_{it}	T	All	x_i velocity	u_{it}
u_amp, v_amp	$u_{i\mathrm{amp}}$	F	All	x_i displacement amplitude	$\lvert u_i \rvert$
`ui_ph`	$u_{i\mathrm{ph}}$	F	All	x_i displacement phase	$(180°/\pi)\,\mathrm{mod}(\mathrm{angle}(u_i), 2\pi)$
`ui_t`	u_{it}	F	All	x_i velocity	$j\omega u_i$

TABLE 7-4: PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
u*i*_t_amp	$u_{it\mathrm{amp}}$	F	All	x_i velocity amplitude	$\omega u_{i\mathrm{amp}}$
u*i*_t_ph	$u_{it\mathrm{ph}}$	F	All	x_i velocity phase	$\mathrm{mod}(u_{i\mathrm{ph}} + 90°, 360°)$
u*i*_tt	u_{itt}	F	All	x_i acceleration	$-\omega^2 u_i$
u*i*_tt_amp	$u_{itt\mathrm{amp}}$	F	All	x_i acceleration amplitude	$\omega^2 \lvert u_i \rvert$
u*i*_tt_ph	$u_{itt\mathrm{ph}}$	F	All	x_i acceleration phase	$\mathrm{mod}(u_{i\mathrm{ph}} + 180°, 360°)$
disp	disp	All	All	Total displacement	$\sqrt{\sum_i (\mathrm{real}(u_i))^2}$
e*i*, exy	ε_i, ε_{xy}	All	S	Strain, global coord. system	
e*i*_t, exy_t	ε_{it}, ε_{xyt}	FT	S	Velocity strain, global coord. system	
e*i*l_t, exyl_t	ε_{ilt}, ε_{xylt}	FT	S	Velocity strain, user-defined coord. system	$T_{\mathrm{coord}}{}^T \varepsilon_t T_{\mathrm{coord}}$
s*i*, sz, sxy	σ_i, σ_z, τ_{xy}	All	S	Cauchy stress, global coord. system	
s*i*l, sxyl	$\sigma_{i\mathrm{l}}$, τ_{xyl}	All	S	Cauchy stress, user-defined coord. system	
s*i*_t, sz_t, sxy_t	σ_{it}, σ_z, τ_{xyt}	FT	S	Time derivative of Cauchy stress, global coord. system	
s*i*	σ_i	All	S	Principal stresses	
e*i*	ε_i	All	S	Principal strains	
s*i*x*j*	σ_{ixj}	All	S	Principal stress directions	

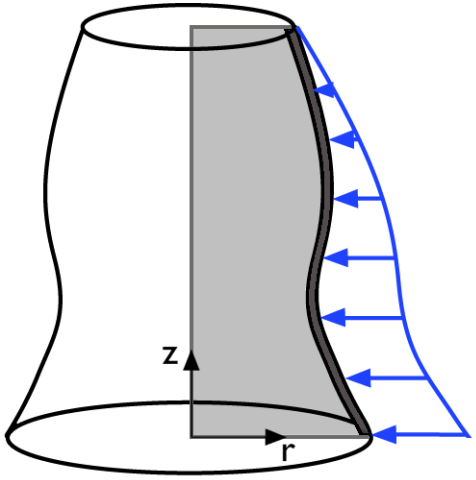
TABLE 7-4: PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
e*i*x*j*	ε_{ixj}	All	S	Principal strain directions	
tresca	σ_{tresca}	All	S	Tresca stress	$\max(\max(\lvert\sigma_1-\sigma_2\rvert, \lvert\sigma_2-\sigma_3\rvert), \lvert\sigma_1-\sigma_3\rvert)$
mises	σ_{mises}	All	S	von Mises stress	
Ws	W_s	All	S	Strain energy density	Defined differently depending on material model and if mixed or displacement formulation
Ta*i*	Ta_i	All	B	Surface traction (force/area) in x_i direction	Defined differently depending on the force definition
F*i*g	F_{ig}	All	S	Point, edge, body load in global x_i direction	Defined differently depending on the force definition

Axial Symmetry, Stress-Strain

The Axial Symmetry, Stress-Strain application mode uses cylindrical coordinates r, φ (`phi`), and z. It solves equations for the global displacement (u, w) in the r and z directions and the pressure p (only used for mixed formulation). It assumes that the displacement v in the φ direction together with the $\tau_{r\varphi}$, $\tau_{\varphi z}$, $\gamma_{r\varphi}$, and $\gamma_{\varphi z}$ components of the stresses and strains are zero. Loads are independent of φ, and this application mode allows loads only in the r and z directions.

You can view the domain where the application mode solves the equations as the intersection between the original axially symmetric 3D solid and the half plane $\varphi = 0$, $r \geq 0$. Therefore you draw the geometry only in the half plane $r \geq 0$ and recover the original 3D solid by rotating the 2D geometry about the z-axis.



Rotating a 2D geometry to recover a 3D solid.

The equilibrium equations in axial symmetry read

$$\frac{\partial \sigma_r}{\partial r} + \frac{\partial \tau_{rz}}{\partial z} + \frac{\sigma_r - \sigma_\theta}{r} + F_r = 0 \qquad (7\text{-}2)$$
$$\frac{\partial \tau_{rz}}{\partial r} + \frac{\partial \sigma_z}{\partial z} + \frac{\tau_{rz}}{r} + F_z = 0$$

The strain-displacement relationships for the axial symmetry case for small displacements are

$$\varepsilon_r = \frac{\partial u}{\partial r}, \qquad \varepsilon_\varphi = \frac{u}{r}, \qquad \varepsilon_z = \frac{\partial w}{\partial z}, \text{ and} \qquad \gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r}$$

To avoid division by r in the equilibrium equations, which causes problems on the axis where $r = 0$, you must transform the equations: multiply the first equation by r^2 and the second by r. This comes in naturally when the principle of virtual work is used, integrating over the volume by multiplying the integrand by $2\pi r$. In addition to this the application mode also introduces and solves for a new dependent variable

$$\text{uor} = \frac{u}{r},$$

which replaces the true radial displacement, u. The reason is to avoid division by r.

Note: $r = 0$ is the symmetry axis. In the Axisymmetry, Stress-Strain application mode $x \rightarrow r$ and $y \rightarrow z$.

PDE Formulation

The principle of virtual work for the axial symmetry case reads

$$dW = \int_A r(-\varepsilon_{r\text{test}}\sigma_r - \varepsilon_{\varphi\text{test}}\sigma_\varphi - \varepsilon_{z\text{test}}\sigma_z - 2\varepsilon_{rz\text{test}}\tau_{rz} + r \cdot \text{uor}_{\text{test}}F_r + w_{\text{test}}F_z)dA +$$

$$\int_S r(r \cdot \text{uor}_{\text{test}}F_r + w_{\text{test}}F_z)ds + (r \cdot \text{uor}_{\text{test}}F_r + w_{\text{test}}F_z)/2\pi$$

APPLICATION MODE PARAMETERS

The section “Common Application Mode Features” on page 153 defines the parameters you need in modeling including loads, the material and constraints.

Application Mode Variables

A large number of variables are available for use in expressions and postprocessing. In addition to the variables in Table 7-5, almost all application mode parameters are available as variables. Some variables change their availability with the type of analysis, as noted in the Analysis column. For frequency-response analysis a number of additional variables are available. Furthermore, the amplitudes and phases of variables such as strains and stresses are available; to access them, append `_amp` or `_ph` to the variable name. For example:

- `sx_amp` is the amplitude of the normal stress in the x direction.
- `ex_ph` is the phase of the normal strain in the x direction.

The exception to this scheme consists of variables defined using a nonlinear operator such as `mises`, `disp`, `Tresca`, or `s1`.

The Analysis column uses the following abbreviations:

ANALYSIS	ABBREVIATION
Static	S
Frequency response	F
Time dependent	T

TABLE 7-5: AXIAL SYMMETRY STRESS-STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
uor	uor	All	All	r displacement divided by r	uor
uaxi	uaxi	All	All	r displacement	$\text{uor}\cdot r$
w	w	All	All	z displacement	w
uort	uor_t	T	All	r velocity divided by r	uor_t
uaxi_t	uaxi_t	T	All	r velocity	$\text{uor}_t\cdot r$
w_t	w_t	T	All	z velocity	w_t
uaxi_amp	uaxi_{amp}	F	All	r displacement amplitude	$\lvert\text{uaxi}\rvert$
w_amp	w_{amp}	F	All	z displacement amplitude	$\lvert w\rvert$
uaxi_ph	uaxi_{ph}	F	All	r displacement phase	$(180°/\pi)\,\text{mod}(\text{angle}(\text{uaxi}),2\pi)$

TABLE 7-5: AXIAL SYMMETRY STRESS-STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
w_ph	w_{ph}	F	All	z displacement phase	$(180°/\pi)\,\mathrm{mod}(\mathrm{angle}(w), 2\pi)$
uaxi_t	uaxi_t	F	All	r velocity	$j\omega\,\mathrm{uaxi}$
w_t	w_t	F	All	z velocity	$j\omega w$
uaxi_t_amp	$\mathrm{uaxi}_{t\mathrm{amp}}$	F	All	r velocity amplitude	$\omega\,\mathrm{uaxi}_{\mathrm{amp}}$
w_t_amp	$w_{t\mathrm{amp}}$	F	All	z velocity amplitude	$\omega\, w_{\mathrm{amp}}$
uaxi_t_ph	$\mathrm{uaxi}_{t\mathrm{ph}}$	F	All	r velocity phase	$\mathrm{mod}(\mathrm{uaxi}_{\mathrm{ph}} + 90°, 360°)$
w_t_ph	$w_{t\mathrm{ph}}$	F	All	z velocity phase	$\mathrm{mod}(w_{\mathrm{ph}} + 90°, 360°)$
uaxi_tt	uaxi_{tt}	F	All	r acceleration	$-\omega^2\,\mathrm{uaxi}$
w_tt	w_{tt}	F	All	z acceleration	$-\omega^2 w$
uaxi_tt_amp	$\mathrm{uaxi}_{t\mathrm{amp}}$	F	All	r acceleration amplitude	$\omega^2\,\mathrm{uaxi}_{\mathrm{amp}}$
w_tt_amp	$w_{t\mathrm{amp}}$	F	All	z acceleration amplitude	$\omega^2 w_{\mathrm{amp}}$
uaxi_tt_ph	$\mathrm{uaxi}_{t\mathrm{ph}}$	F	All	r acceleration phase	$\mathrm{mod}(\mathrm{uaxi}_{\mathrm{ph}} + 180°, 360°)$
w_tt_ph	$w_{t\mathrm{ph}}$	F	All	z acceleration phase	$\mathrm{mod}(w_{\mathrm{ph}} + 180°, 360°)$
disp	disp	All	All	Total displacement	$\sqrt{\mathrm{uaxi}^2 + w^2}$
er, ez, ephi, erz	ε_r, ε_z, ε_φ, ε_{rz}	All	S	Strain, global coord. system	
epr, epz, epphi, eprz	ε_{pr}, ε_{pz}, $\varepsilon_{p\varphi}$, ε_{prz}	ST	S	Plastic strain, global coord. system	
er_t, ez_t, ephi_t, erz_t	ε_{rt}, ε_{zt}, $\varepsilon_{\varphi t}$, ε_{rzt}	FT	S	Velocity strain, global coord. system	Defined differently depending on small or large displacement
sr, sphi, sz, srz	σ_r, σ_φ, σ_z, τ_{rz}	All	S	Cauchy stress, global coord. system	

TABLE 7-5: AXIAL SYMMETRY STRESS-STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
sr_t, sphi_t, sz_t, srz_t	σ_{rt}, $\sigma_{\varphi t}$, σ_{zt}, σ_{rzt}	FT	S	Time der. of Cauchy stress, global coord. system	
s*i*	σ_i	All	S	Principal stresses	
e*i*	ε_i	All	S	Principal strains	
s*i*x*j*	σ_{ixj}	All	S	Principal stress directions	
e*i*x*j*	ε_{ixj}	All	S	Principal strain directions	
tresca	σ_{tresca}	All	S	Tresca stress	$\max(\max(\lvert\sigma_1-\sigma_2\rvert, \lvert\sigma_2-\sigma_3\rvert), \lvert\sigma_1-\sigma_3\rvert)$
mises	σ_{mises}	All	S	von Mises stress	
Ws	W_s	All	S	Strain energy density	Defined differently depending on material model and if mixed or displacement formulation
Tar, Taz	Ta_r, Ta_z	All	B	Surface traction (force/area) in r and z directions	Defined differently depending on small or large deformation
Frg, Fzg	F_{rg}, F_{zg}	All	All	Body, edge, point load in global r and z directions	Defined differently depending on force definition

8

Piezoelectric Application Modes

This chapter describes the application modes for modeling piezoelectric effects in the Acoustics Module.

Theory Background

The Piezoelectric Effect

The piezoelectric effect manifests itself as a transfer of electric to mechanical energy and vice-versa. It is observable in many crystalline materials, while some materials such as quartz, Rochelle salt, and lead titanate zirconate ceramics display the phenomenon strongly enough for the phenomenon to be of practical use.

The *direct* piezoelectric effect consists of an electric polarization in a fixed direction when the piezoelectric crystal is deformed. The polarization is proportional to the deformation and causes an electric potential difference over the crystal.

The *inverse* piezoelectric effect, on the other hand, constitutes the opposite of the *direct* effect. This means that an applied potential difference induces a deformation of the crystal.

PIEZOELECTRIC CONVENTIONS

The documentation and the user interface use piezoelectric conventions as far as possible. These conventions differ from those used in other structural mechanics application modes. For instance, the numbering of the shear components in the stress-strain relation differs, as the following section describes. However, the names of the stress and strain components remain the same as in the other structural mechanics application modes.

Piezoelectric Constitutive Relations

It is possible to express the relation between the stress, strain, electric field, and electric displacement field in either a stress-charge or strain-charge form:

STRESS-CHARGE

$$\mathbf{T} = c_E\mathbf{S} - e^T\mathbf{E}$$
$$\mathbf{D} = e\mathbf{S} + \varepsilon_S\mathbf{E}$$

STRAIN-CHARGE

$$\mathbf{S} = s_E\mathbf{T} + d^T\mathbf{E}$$
$$\mathbf{D} = d\mathbf{T} + \varepsilon_T\mathbf{E}$$

The naming convention differs in piezoelectric theory compared to structural mechanics theory, but the piezoelectric application modes use the structural mechanics nomenclature. The strain is named ε instead of $\mathbf{S}$, and the stress is named σ instead of $\mathbf{T}$. This makes the names consistent with those used in the other structural mechanics application modes.

The numbering of the strain and stress components is also different in piezo and structural mechanics theory, and it is quite important to keep track of this aspect in order to give the correct material data. In structural mechanics the following is the most common numbering convention, and it is also the one used in the other structural mechanics application modes:

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ 2\varepsilon_{xy} \\ 2\varepsilon_{yz} \\ 2\varepsilon_{xz} \end{bmatrix}$$

In contrast, textbooks on piezoelectric effects and the IEEE standard on piezoelectric effects use the following numbering convention:

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{yz} \\ \tau_{xz} \\ \tau_{xy} \end{bmatrix} \qquad \varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{yz} \\ \gamma_{xz} \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ 2\varepsilon_{yz} \\ 2\varepsilon_{xz} \\ 2\varepsilon_{xy} \end{bmatrix}$$

The piezoelectric application modes employ the immediately preceding piezo numbering convention to make it easier to work with materials data and avoid mistakes.

The constitutive relation using COMSOL Multiphysics symbols for the different constitutive forms are thus:

STRESS-CHARGE

$$\sigma = c_E \varepsilon - e^T \mathbf{E}$$
$$\mathbf{D} = e\varepsilon + \varepsilon_0 \varepsilon_{rS} \mathbf{E}$$

STRAIN-CHARGE

$$\varepsilon = s_E \sigma + d^T \mathbf{E}$$
$$\mathbf{D} = d\sigma + \varepsilon_0 \varepsilon_{rT} \mathbf{E}$$

Most material data appears in the strain-charge form, and you can easily transform into the stress-charge form. COMSOL Multiphysics allows you to use both constitutive forms; simply select one, and COMSOL Multiphysics makes any necessary transformations. The following equations transform strain-charge material data to stress-charge data

$$c_E = s_E^{-1}$$
$$e = d \; s_E^{-1}$$
$$\varepsilon_S = \varepsilon_0 \varepsilon_{rT} - d \; s_E^{-1} \; d^T$$

Material Models

In addition to modeling piezoelectric materials, the Piezoelectric application mode provides different material models for easier modeling of piezo components. This means, that in the subdomain settings of the application mode, you can define the material of each domain as:

- Piezoelectric
- Decoupled, isotropic
- Decoupled, anisotropic

The Piezoelectric material operates as described in the chapter above, whereas using the two other material models, you can model structural and electrical problems or either of them independently.

The structural part of the *Decoupled, isotropic* and *Decoupled, anisotropic* material operates as the linear elastic material with small deformations as described in "Structural Mechanics Application Modes" on page 59"Structural Mechanics

Application Modes" on page 143. However, the initial stress and strain and thermal expansion are not supported within the Piezoelectric application mode.

For the *Decoupled, isotropic* material you define the material using the Young's modulus, E, and the Poisson ratio, ν. For the *Decoupled, anisotropic* material you define the full 6-by-6 elasticity matrix D. Note here, that you define D using the standard structural mechanics ordering. Thus the ordering of the D is different from the ordering of the piezoelectric c_E matrix.

Depending on the value of the *Electrostatics formulation* property (See "Electrical Formulations" below), the electrical part of the *Decoupled, isotropic* and *Decoupled, anisotropic* material solves either the electrostatics equation:

$$-\nabla \cdot (\varepsilon_0 \varepsilon_r \nabla V) = \rho_v$$

where ε_0 is the electrical permittivity of free space, ε_r is the relative electrical permittivity, and ρ_v is the volume charge density, or the quasi-static electric currents equation:

$$-\nabla \cdot ((\sigma_e + j\omega\varepsilon_0\varepsilon_r)\nabla V) = 0$$

where σ_e is the electrical conductivity of the material (note that σ is used also for the structural stress vector).

In frequency response analysis the conductivity appears also into the electrostatics equation:

$$-\nabla \cdot \left(\left(\frac{\sigma_e}{j\omega} + \varepsilon_0\varepsilon_r\right)\nabla V\right) = \rho_v$$

and thus you can define and use conductivity of the material independently of the Electrostatics formulation property.

For a *Decoupled, isotropic* material you define ε_r and σ_e as scalars, but for a *Decoupled, anisotropic* material you define them as 3-by-3 matrices.

Electrical Formulations

The default formulation of the equations in the Piezoelectric application modes is such that the resulting equation system with piezoelectric material is symmetric. This allows reduced memory requirements with solvers that utilize symmetry information.

The drawback of this design is that by default the Piezoelectric application modes are not electrically compatible with the Electrostatics application mode found in the AC/DC Module and the MEMS Module, nor is it compatible with the Quasi-Statics -Electric, Electric currents application modes in the AC/DC Module.

The Piezoelectric application modes support an application mode property, Electrostatics formulation, which makes them compatible with the electrostatic or quasi-static application modes so that it is possible to couple them in a model. The Electrostatics formulation property has the following choices:

- Symmetric, Electrostatics: The default implementation creates a symmetric equation system, but the application mode is not compatible with the other application modes.
- Unsymmetric, Electrostatics: This implementation creates an unsymmetric equation system which is compatible with the Electrostatics application modes.
- Unsymmetric, Electric currents: This implementation creates an unsymmetric equation system which is compatible with the Quasistatics - Electric, Electric currents application modes.

At the equation level the difference between these formulation is the following. The default formulation is that the variational electrical energy is written using a positive sign:

$$\delta We = \int (\mathbf{D} \cdot \hat{\mathbf{E}}) d\Omega$$

Here $\mathbf{D}$ is the electric displacement vector, and $\hat{\mathbf{E}}$ is the test function for the Electric field. Ω is the integration domain.

On the other hand, the formulation compatible with the Electrostatics application mode uses variational electrical energy with the negative sign:

$$\delta We = -\int (\mathbf{D} \cdot \hat{\mathbf{E}}) d\Omega$$

Finally, the electric currents formulation uses the following variational electrical energy:

$$\delta We = \int (\mathbf{J} \cdot \nabla \hat{V}) d\Omega$$

where $\mathbf{J}$ is the electric current density vector, and $\nabla \hat{V}$ is the test function for the potential gradient.

The use of the Unsymmetric, electric currents formulation sets certain limitations: you cannot model any charges, and any boundary conditions that use charges or electric displacement are written in terms of electric current. Also, this formulation only appears in the frequency response analysis.

The Piezoelectric Application Modes

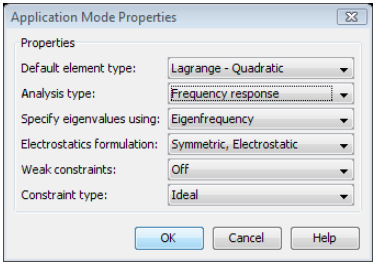
This section describes the interface for defining a model using the piezoelectric application modes:

- Piezo Solid (in 3D)
- Piezo Plane Strain (in 2D)
- Piezo Axial Symmetry (in axisymmetric 2D)

It consists of the following sections:

Application Mode Properties

To set or examine material properties, go to the **Physics** menu and open the **Application Mode Properties** dialog box.



Here you control various global settings for the model, which include:

- **Default element type**: A list of elements, where the selection becomes the default on all new subdomains. The default is to use second-order Lagrange elements.
- **Analysis type**: A list of analyses to perform. It affects both the equations and which solver to use with the **Auto select solver** option in the **Solver Parameters** dialog box.

The default is static analysis. You can also select transient, eigenfrequency, damped eigenfrequency, and frequency response analysis types.

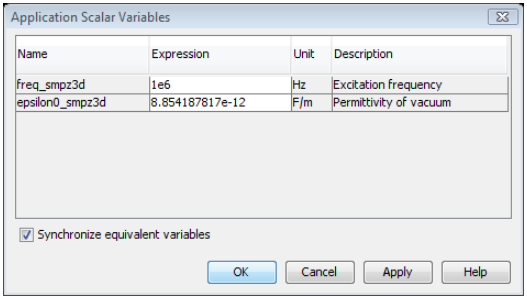
- **Specify eigenvalues using**: A list controlling whether the application mode works with eigenvalues or eigenfrequencies.
- **Electrostatics formulation**: Select the electrical formulation to use:
  - **Symmetric, Electrostatic**: the default setting.
  - **Unsymmetric, Electrostatic**: for compatibility with the Electrostatics application mode.
  - **Unsymmetric, Electric currents**: for compatibility with the application modes for electric currents in the AC/DC Module (Electric Currents in 3D, In-Plane Electric Currents in 2D, and Meridional Electric Currents in 2D axial symmetry). Available for frequency response analysis.
- **Weak constraints**: Controls whether or not weak constraints are active Use weak constraints for accurate reaction-force computation. When weak constraints are enabled, all constraints are weak by default, but it is possible to change this setting for individual domains.
- **Constraint type**: Constraints can be ideal or nonideal (see "Ideal vs. Non-Ideal Constraints" on page 301 in the *COMSOL Multiphysics Modeling Guide*).

Scalar Variables

The piezoelectric application modes have the following scalar variables:.

PROPERTY	VARIABLE	DEFAULT	SI UNIT	DESCRIPTION
ε_0	`epsilon0`	`8.854187817e-12`	F/m	Permittivity of vacuum
f	`freq`	`1e6`	Hz	Excitation frequency
$j\omega$	`jomega`	`-lambda`	rad/s	Complex angular frequency

You control the scalar variables by going to the **Physics** menu and opening the **Application Scalar Variables** dialog box.



The excitation frequency (the frequency of the harmonic forces, potential, and displacement) is available only for frequency response analysis. The equations and documentation describing frequency response use the angular excitation frequency, $\omega = 2\pi f$, which is available as the variable `omega`. The complex angular frequency is available for eigenfrequency analysis and damped eigenfrequency analysis.

When you select **Frequency response** as the analysis type, the default solver is the parametric solver. This default makes it easy to perform a frequency sweep over several excitation frequencies in one analysis. In this case enter `freq` as the **Parameter name** on the **General** page in the **Solver Parameters** dialog box. The values you enter in the **Parameter values** edit field override the excitation frequency you might have entered in the **Application Scalar Variables** dialog box.

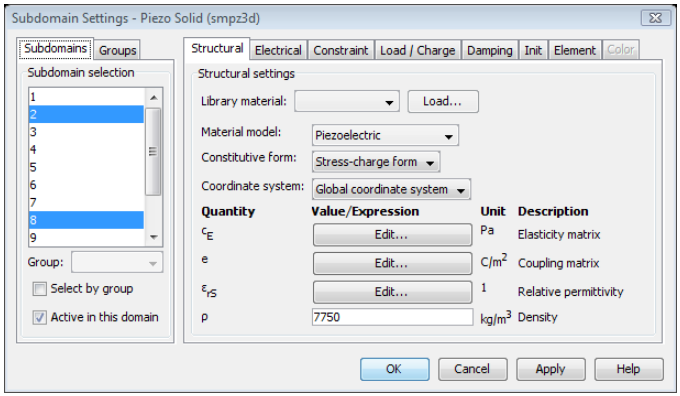
Material Properties

The **Subdomain Settings** window has two pages where you define the material properties: the **Structural** page and the **Electrical** page. On top of both pages you find the **Library material** list and the **Load** button for importing and selecting data from the material libraries and the **Material model** list for selecting the material model for each domain. These settings are shared between the pages, and if you change the **Structural** page, the settings change also on the **Electrical** page. Note that loading a material from a material library does not change the material model, so you need to change it manually in the **Material model** list to match the type of material.

Everything else you see and define on the pages depends on the material model you select. Setting for different material model are described in the following chapters.

SUBDOMAIN SETTINGS FOR PIEZOELECTRIC MATERIAL

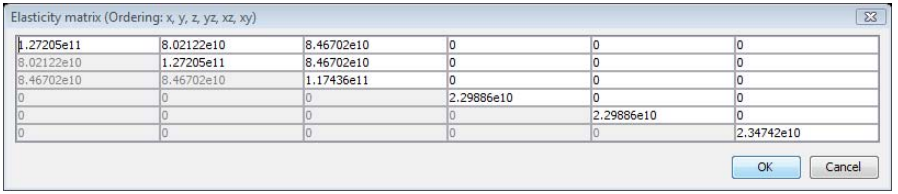
The piezoelectric material is a complete structural-electrical material, and thus you define all piezoelectric material properties on the **Structural** page.



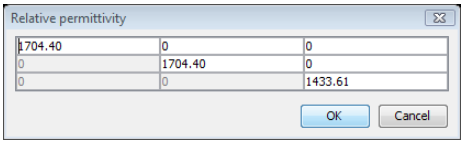
The **Structural** page has two lists in 3D, three lists in 2D, and three lists in axial symmetry:

- **Constitutive form**: Select the constitutive form from those in the following list. Depending on the selection, different material properties are shown in the dialog box.
  - **Stress-charge form**: Define the constitutive relation of the material on the stress-charge form through the c_E, e, and ε_{rS} matrices. The previous figure shows

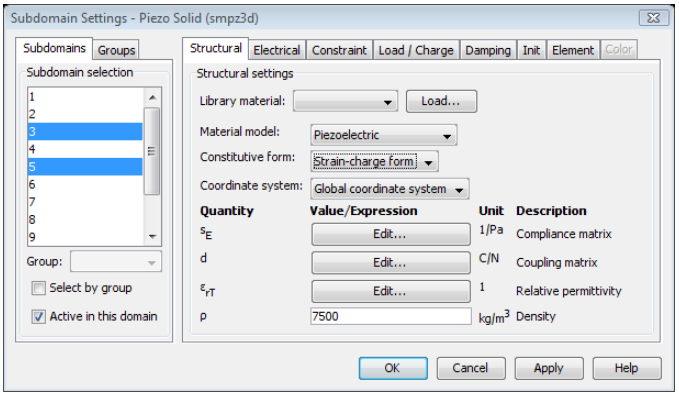
the **Material** page for stress-charge, while the following figure shows the **Elasticity matrix** dialog box for entering the c_E matrix.



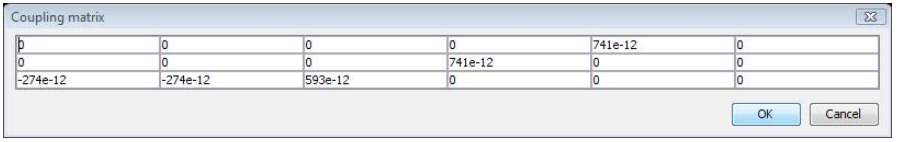
The figure below shows the **Relative permittivity** dialog box for entering the ε_{rS} matrix components.



- **Strain-charge form**: You define the constitutive relation of the material on the strain-charge form through the s_E, d, and ε_{rT} matrices (see page 190 for details). The following figure shows the **Material** page for strain-charge.



The next graphic shows the **Coupling matrix, strain-charge form** dialog box for entering the d matrix components.



- **Material orientation** (2D and axisymmetry only): Here you select how the 3D

material properties are oriented relative the 2D/axial symmetric analysis plane. There are six options: *xy*, *yz*, *zx*, *yx*, *zy*, and the default *xz*-plane. The plane represents how the 3D material is oriented relative the 2D/axial symmetric analysis plane: The first letter indicates which 3D direction coincides with the *x* direction in 2D or the *r* direction for axisymmetry; the second letter indicates which 3D direction coincides with the *y* direction in 2D or the *z* direction for axisymmetry. The material coordinates names are fixed and do not depend of the names of the space coordinates (independent variables), which have different defaults in 2D and axial symmetry.

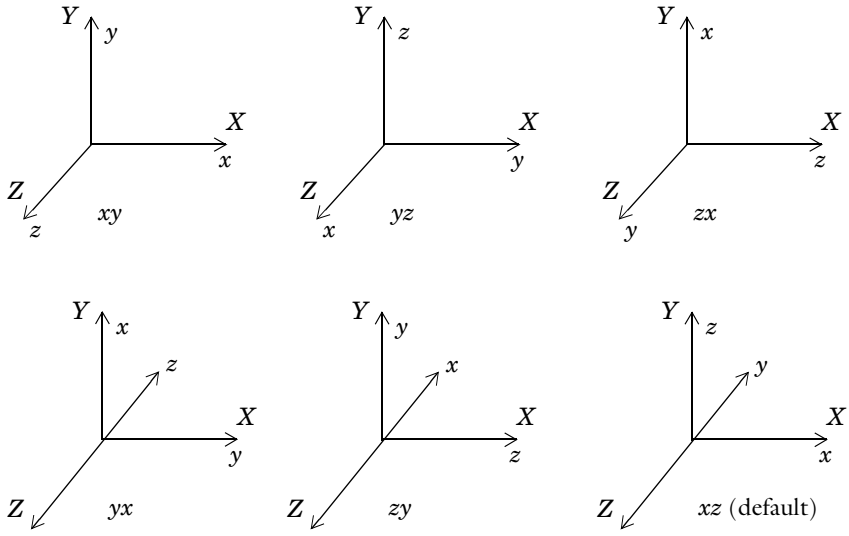


Figure 8-1:Orientation of 3D material xyz relative the 2D analysis coordinate system XYZ.

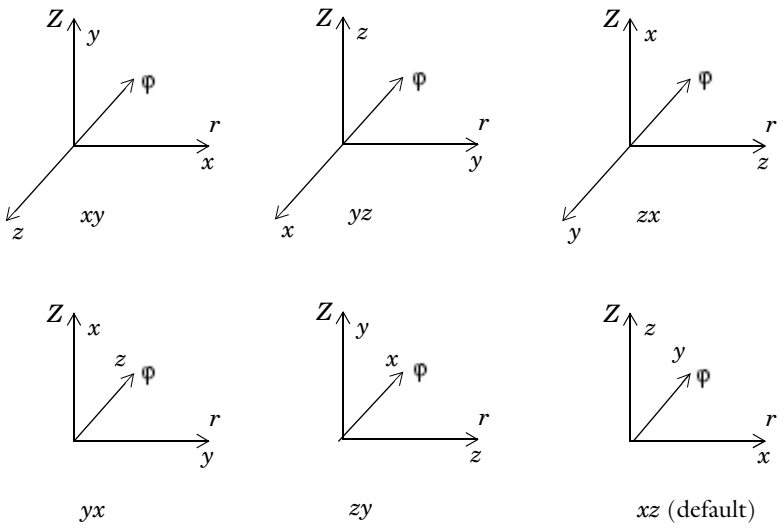


Figure 8-2: Orientation of 3D material xyz relative the axisymmetric analysis coordinate system r φ Z.

- **Coordinate system**: Select the coordinate system where the material properties are defined. This choice is useful if you want to define the material in a coordinate system other than the global system, or if you need results in a local coordinate system for postprocessing. The **Coordinate system** list contains only the global coordinate system unless you have made available a user-defined coordinate system. You find the **Coordinate System Settings** dialog box on the **Options** menu.

The following table shows the material properties for the union of all constitutive forms and all piezoelectric application modes.

PARAMETER	VARIABLE	DESCRIPTION	CONSTITUTIVE FORM
c_E	`cElk`	Elasticity matrix	Stress-charge
s_E	`sElk`	Compliance matrix	Strain-charge
e	`eik`	Coupling matrix, stress-charge form	Stress-charge
d		Coupling matrix, strain-charge form	Strain-charge
ε_{rS}		Relative permittivity matrix, stress-charge form	Stress-charge

PARAMETER	VARIABLE	DESCRIPTION	CONSTITUTIVE FORM
ε_{rT}		Relative permittivity matrix, strain-charge form	Strain-charge
ρ	`rho`	Density	All
th	`thickness`	Thickness of the geometry (2D only)	All

Elasticity matrix defines the stress-strain relation matrix c_E

$$\sigma = c_E \varepsilon$$

where σ is the stress, and ε is the strain.

Coupling matrix defines the piezo coupling matrix e used in the stress-charge form of the constitutive equation

$$\sigma = c_E \varepsilon - e^T \mathbf{E}$$

where σ is the stress, ε is the strain, and $\mathbf{E}$ is the electric field.

Compliance matrix defines the strain-stress relation matrix s_E

$$\varepsilon = s_E \sigma$$

where σ is the stress, and ε is the strain.

Coupling matrix defines the piezo coupling matrix d used in the strain-charge form of the constitutive equation

$$\varepsilon = s_E \sigma + d^T \mathbf{E}$$

where σ is the stress, ε is the strain, and $\mathbf{E}$ is the electric field.

Relative permittivity the relative permittivity, ε_{rS} and ε_{rT}, appears in the constitutive relation on stress-charge and strain-charge forms, respectively.

$$\mathbf{D} = e\varepsilon + \varepsilon_0 \varepsilon_{rS} \mathbf{E}$$

$$\mathbf{D} = d\sigma + \varepsilon_0 \varepsilon_{rT} \mathbf{E}$$

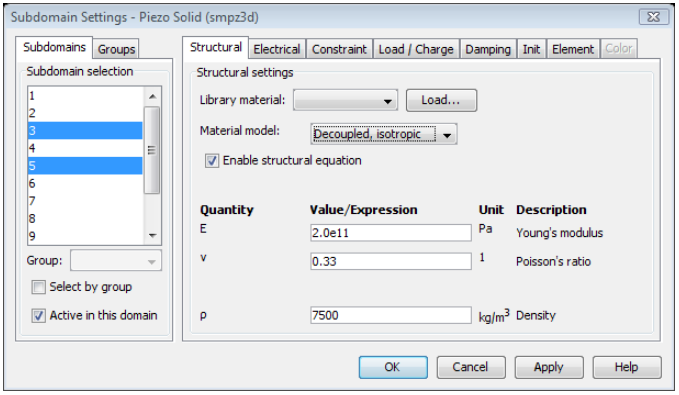
Density this material property, ρ, specifies the material's density.

Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

SUBDOMAIN SETTINGS FOR DECOUPLED, ISOTROPIC MATERIAL

With this material model you specify material properties on the **Structural** page and the **Electrical** page.

You define the structural material properties on the **Structural** page:



On the first row after the **Material Model** list you find the **Enable structural equation** check box. Use this check box to activate the structural equation or inactivate it to model only electrical problems. By default the **Enable structural equation** check box is selected. If this setting is selected you can define the following structural material properties:

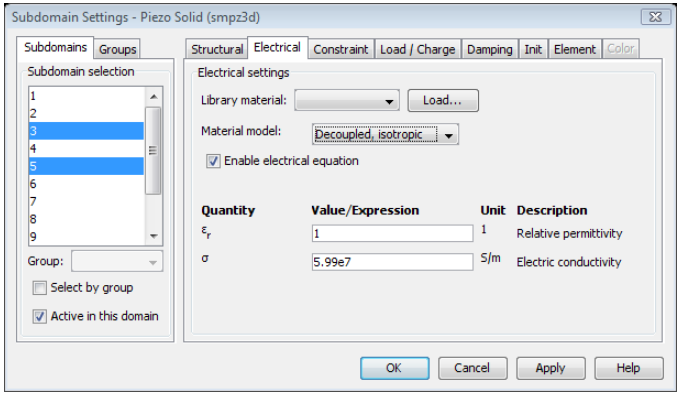
Young's modulus This material property, E, is the modulus of elasticity of the material. It is used to form the elasticity matrix D for the stress strain relationship as described in the chapter "Material Models" on page 190.

Poisson's ratio This material property, ν, defines the contraction of the structure in the perpendicular direction. It is used to form the elasticity matrix D for the stress strain relationship as described in the chapter "Material Models" on page 190.

Density this material property, ρ, specifies the material's density.

Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

You define the electrical material properties on the **Electrical** page:



On the first row after the **Material Model** list you find the **Enable electrical equation** check box. Use this check box to activate the electrical equation or inactivate it to model only structural problems. If you select it and clear the **Enable structural equation** check box, only the electrical equation is active. By default the **Enable electrical equation** check box is selected. If this setting is selected you can define the following electrical material properties:

Relative permittivity This material property, ε_r, defines the isotropic relative electrical permittivity of the material.

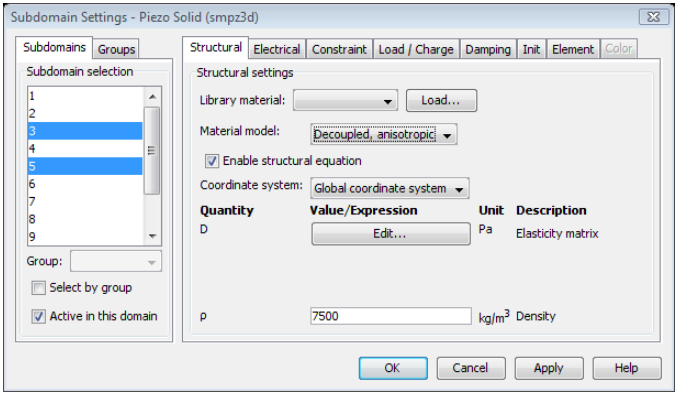
Electric conductivity This material property, σ, defines the isotropic electrical conductivity of the material. This setting only appears for frequency response analysis.

Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

SUBDOMAIN SETTINGS FOR DECOUPLED, ANISOTROPIC MATERIAL

With this material model you specify material properties on the **Structural** page and the **Electrical** page.

You define the structural material properties on the **Structural** page:

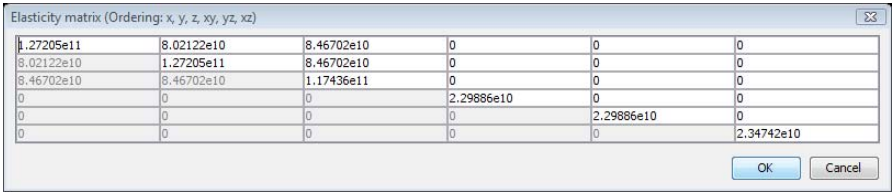


On the first row after the **Material Model** list you find the **Enable structural equation** check box. Use this check box to activate the structural equation or inactivate it to model only electrical problems. By default, **Enable structural equation** is selected. If this setting is selected you can define the following structural material properties:

Material orientation (2D and axisymmetry only): Here you select how the 3D material properties are oriented relative the 2D/axial symmetric analysis plane. There are six options: *xy*, *yz*, *zx*, *yx*, *zy*, and the default *xz*. This setting works the way same as for the piezoelectric material (See description on page 198).

Coordinate system Select the coordinate system where the material properties are defined. This setting works the way same as for the piezoelectric material (See description on page 200).

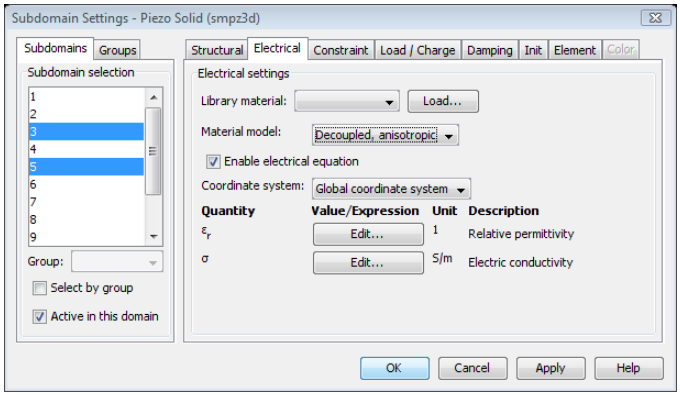
Elasticity matrix This material property, D, defines the elasticity matrix of the anisotropic material (See "Material Models" on page 190.). You define D as a symmetric 6-by-6 matrix:



Density this material property, ρ, specifies the material's density.

Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

You define the electrical material properties on the **Electrical** page:

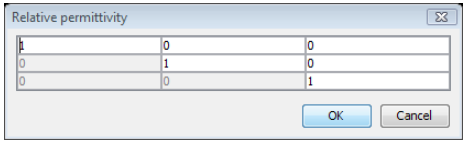


On the first row after the **Material Model** list you find the **Enable electrical equation** check box. Use this check box to activate the electrical equation or inactivate it to model only structural problems. By default **Enable electrical equation** is selected. If this setting is selected you can define the following electrical material properties:

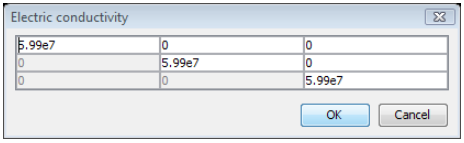
Material orientation (2D and axisymmetry only) This is the same setting as the **Material orientation** in the **Structural** page.

Coordinate system This is the same setting as the **Coordinate system** on the **Structural** page.

Relative permittivity This material property, ε_r, defines the anisotropic relative electrical permittivity of the material. You define ε_r using a symmetric 3-by-3 matrix:



Electric conductivity This material property, σ, defines the anisotropic electrical conductivity of the material. This setting only appears for frequency response analysis. You define σ using a symmetric 3-by-3 matrix:



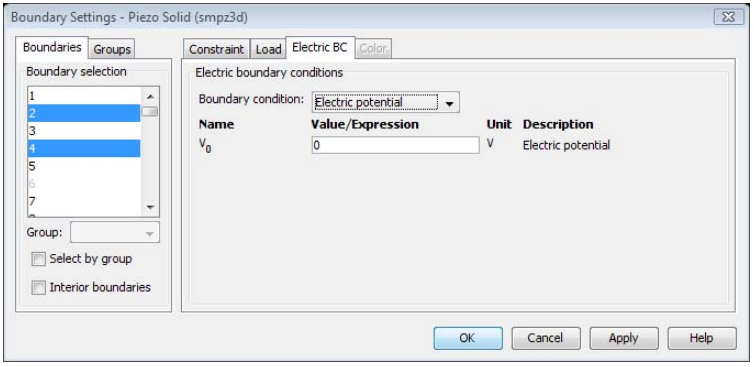
Thickness this material property, thickness, specifies the material's thickness and appears in 2D only.

PIEZOELECTRIC MATERIALS PROPERTIES LIBRARY

A library of about 25 common piezoelectric materials is available through the **Materials/ Coefficients Library** dialog box."Piezoelectric Material Properties Library" on page 47

Electric Boundary Conditions

The electric boundary conditions in the piezoelectric application modes depend on the setting of the **Electrostatics formulation** property in the **Application Mode Properties** dialog box. You specify the electric boundary conditions on the **Electric BC** page in the **Boundary Settings** dialog box.



The **Electric BC** page also has a **Boundary condition** list where you select the type of electric boundary condition; the software enables different edit fields depending on the selected type.

BOUNDARY CONDITIONS FOR ELECTROSTATICS

For the Unsymmetric, Electrostatic and Symmetric, Electrostatic formulations, the boundary conditions include:

Electric Displacement

$$\mathbf{n} \cdot \mathbf{D} = \mathbf{n} \cdot \mathbf{D}_0$$

This boundary condition specifies the normal component of the electric displacement at a boundary. Enter the components of the electric displacement $\mathbf{D}_0$.

Surface Charge

$$-\mathbf{n} \cdot \mathbf{D} = \rho_s, \qquad \mathbf{n} \cdot (\mathbf{D}_1 - \mathbf{D}_2) = \rho_s$$

This boundary condition specifies the surface charge density ρ_s at an exterior boundary (left equation) or at the interior boundary between two media with electric displacement $\mathbf{D}_1$ and $\mathbf{D}_2$, respectively.

Zero Charge/Symmetry

$$\mathbf{n} \cdot \mathbf{D} = 0$$

This boundary condition specifies that the normal component of the electric displacement is zero. The Zero charge/Symmetry boundary condition is also useful at symmetry boundaries where the potential is symmetric with respect to the boundary.

Electric Potential

$$V = V_0$$

This boundary condition specifies the voltage V_0 at the boundary. Because the application mode computes the electric potential, you must define its value at some boundary in the geometry to be fully determined.

Ground

$$V = 0$$

This boundary condition is a special case of the previous one specifying zero potential. The Ground boundary condition is also be useful at symmetry boundaries, where the potential is antisymmetric with respect to the boundary.

Continuity

$$\mathbf{n} \cdot (\mathbf{D}_1 - \mathbf{D}_2) = 0$$

This boundary condition specifies that the normal component of the electric displacement is continuous across an interior boundary or across a boundary between a piezoelectric and an electrostatic domain if you use the Unsymmetric, Electrostatic formulation. Using the Symmetric, Electrostatic formulation the Continuity condition is only available for interior boundaries, where it is the default.

Floating Potential

This condition the potential on the boundary to a spatially constant value such that the total charge on the boundary equals the user defined total charge Q_0:

$$\int_{\partial\Omega} \rho_s = Q_0$$

You also define the group index, which defines how the boundaries are grouped in to a set of electrodes.

Axial Symmetry

$$E_r = 0$$
$$\frac{\partial E_z}{\partial r} = 0$$

This boundary condition is the natural Neumann boundary condition, which you use on the z-axis ($r = 0$) to maintain the symmetry conditions. The Axial Symmetry boundary condition is available only in the Piezo Axial Symmetry application mode.

BOUNDARY CONDITIONS FOR ELECTRIC CURRENTS

For the Unsymmetric, Electric currents formulations, the boundary conditions include:

Ground

$$V = 0$$

This boundary condition is a special case of the previous one specifying zero potential. The Ground boundary condition is also be useful at symmetry boundaries, where the potential is antisymmetric with respect to the boundary.

Electric Potential

$$V = V_0$$

This boundary condition specifies the voltage V_0 at the boundary. Because the application mode computes the electric potential, you must define its value at some boundary in the geometry to be fully determined.

Current Flow

$$\mathbf{n} \cdot \mathbf{J} = \mathbf{n} \cdot \mathbf{J}_0$$

This boundary condition specifies the current flow. Enter the components of the current density $\mathbf{J}_0$.

Inward Current Flow

$$-\mathbf{n} \cdot \mathbf{J} = J_n$$

This boundary condition specifies the normal current density J_n at an exterior boundary.

Electric Insulation

$$\mathbf{n} \cdot \mathbf{J} = 0$$

This boundary condition specifies that the normal component of the electric current is zero; that is, the boundary is electrically insulated.

Current Source

The current source boundary condition

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = J_n$$

is applicable to interior boundaries that represent either a source or a sink of current.

Continuity

$$\mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) = 0$$

This boundary condition specifies that the normal component of the electric current is continuous across the interior boundary (where it is the default setting) or across a boundary between a piezoelectric and an domain with electric currents.

Floating Potential

This condition the potential on the boundary to a spatially constant value such that the total current through the boundary equals the user defined total current I_0:

$$\int_{\partial\Omega} -\mathbf{n} \cdot \mathbf{J} = I_0$$

You also define the group index, which defines how the boundaries are grouped in to a set of electrodes.

Axial Symmetry

This boundary condition is the natural Neumann boundary condition, which you use on the z-axis ($r = 0$) to maintain the symmetry conditions. The Axial Symmetry boundary condition is available only in the Piezo Axial Symmetry application mode.

CONVERSION OF ELECTRIC BOUNDARY CONDITIONS

Some boundary conditions are applicable only for the formulations for electrostatics, whereas others apply only to the formulation for electric currents. Table 8-1 contains the boundary conditions that the software converts when changing from one formulation to the other:

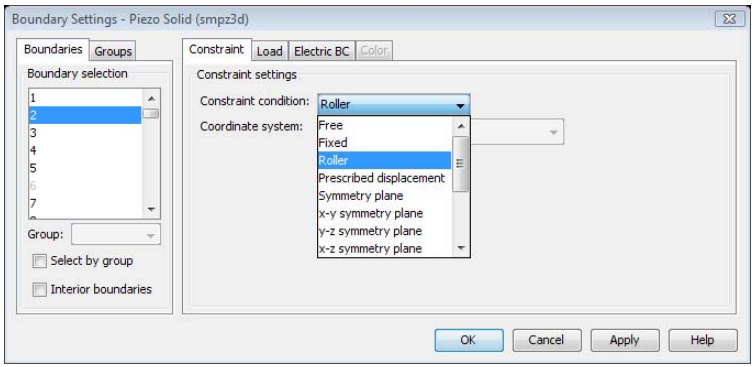
TABLE 8-1: BOUNDARY CONDITION CONVERSIONS

BOUNDARY CONDITION FOR ELECTROSTATICS	BOUNDARY CONDITION FOR ELECTRIC CURRENTS
Electric displacement	Current flow
Zero charge/Symmetry	Electric insulation
Surface charge (exterior boundaries)	Inward current flow
Surface charge (interior boundaries)	Current source

Constraints

A constraint specifies the displacement or potential of certain parts of a structure. You can define constraints for the displacements on all domain levels including points, edges, faces/boundaries, and subdomains (in 3D), and points, boundaries, and subdomains (in 2D). In addition, you can define constraints for the potential on points and edges in 3D, and for points in 2D. To control them, go to the **Constraint** page in the **Subdomain/Boundary/Edge/Point Settings** dialog boxes, and set constraints on boundaries from the **Electric BC** page. The following figure shows the **Boundary Settings**

dialog box for the Piezo Solid application mode, but the page has the same appearance in all piezoelectric application modes.

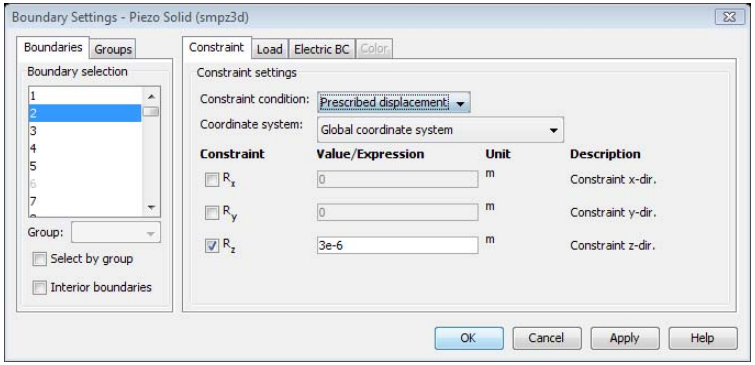


Use the **Constraint condition** list in this dialog box to select the type of constraint that you want to define.

The **Coordinate system** list lets you control in which coordinate system you want the constraint defined. Available options are:

- Global coordinate system
- Tangent and normal coordinate system, available only on boundaries
- User-defined coordinate systems, if any local coordinate systems are defined.

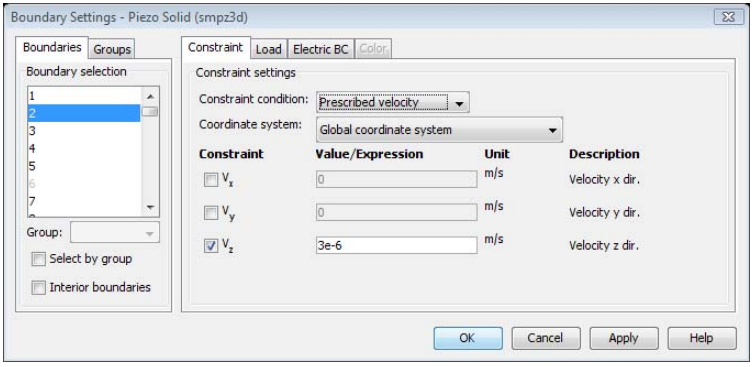
When you select **Prescribed displacement** a number of new options appears in the dialog box and the **Constraint** page takes on this appearance:



The Constraint page showing the prescribed displacement options.

The check boxes adjacent to the $\mathbf{R_x}$, $\mathbf{R_y}$, and $\mathbf{R_z}$ edit fields activate the constraint, whereupon you enter the value/expression of the displacement (the default value is `0`).

In a frequency response analysis you have the possibility to specify not only a harmonic displacement but also a harmonic velocity or acceleration. You specify the **Prescribed velocity** and **Prescribed acceleration** in the same way as **Prescribed displacement**.

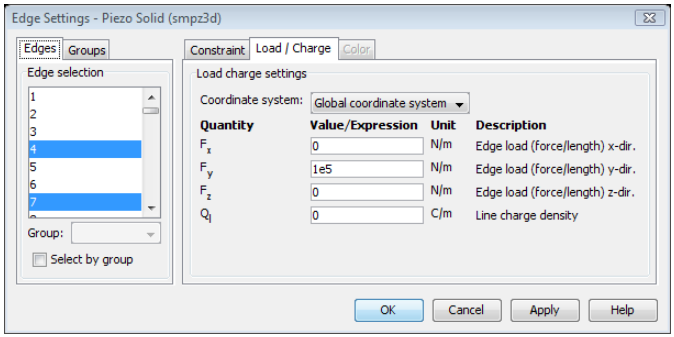


Constraint page showing the prescribed velocity settings.

Loads and Charges

Load is a general name for forces applied to a structure. You can specify loads on all domain types. To do so, click the **Load** tab in the **Boundary Settings** dialog boxes or the **Load/Charge** tab in the **Subdomain Settings**, **Edge Settings**, and **Point Settings** dialog boxes, where you can also specify a charge density. The formulation for electric currents does not include charges, so in that case, the name of the tab is **Load** also in the **Subdomain Settings**, **Edge Settings**, and **Point Settings** dialog boxes. The following

image shows the **Edge Settings** dialog box for the Piezo Solid application mode, but the tab looks similar on all domain levels in all piezoelectric application modes.



SPECIFYING LOADS

For plane stress and plane strain, option buttons allow you to specify the load in different ways using the thickness. The following table summarizes the options for defining loads on different domains in different application modes; the SI unit appears in parenthesis.

APPLICATION MODE	POINT	EDGE	BOUNDARY	SUBDOMAIN
Plane Stress, Plane Strain	force (N)		force/area (N/m^2) or force/length (N/m)	force/volume (N/m^3) or force/area (N/m^2)
Axial symmetry	total force along the circumferential (N)		force/area (N/m^2)	force/volume (N/m^3)
Solid	force (N)	force/length (N/m)	force/area (N/m^2)	force/volume (N/m^3)

With the **Coordinate system** list you control in which coordinate system the load is defined. Available options are:

- Global coordinate system
- Tangent and normal coordinate system, only available on boundaries
- User-defined coordinate systems, if there are any local coordinate systems defined.

SPECIFYING CHARGES

You can specify a charge on the **Edge**/**Point** level when you use a formulation for electrostatics. For plane stress and plane strain, option buttons allow you to specify the charge in different ways using the thickness. The following table summarizes the

options for defining charge on different domains in different application modes; the SI units appears in parenthesis.

APPLICATION MODE	POINT	EDGE	SUBDOMAIN
Plane Stress, Plane Strain	charge (C)		charge/volume (C/m^3) or charge/area (C/m^2)
Axial symmetry	total charge along the circumferential (C)		charge density (C/m^3)
Solid	force (C)	charge/length (C/m)	charge density (C/m^3)

To specify charge density on boundaries, click the **Electric BC** tab.

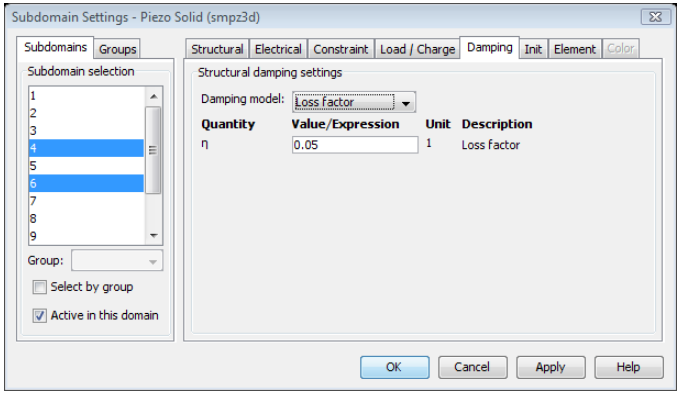
Structural Damping

For time-dependent analysis, you can specify viscous damping (structural damping) using Rayleigh damping, where the damping matrix is specified to be proportional to the mass and stiffness matrix:

$$C = \alpha_{dM} M + \beta_{dK} K$$

For frequency response analysis you can specify viscous damping using either Rayleigh damping, loss factor damping, or equivalent viscous damping.

To specify structural damping parameters, go to the **Damping** page in the **Subdomain Settings** dialog box, and choose the type of damping model from the **Damping model** list. The layout of the dialog box changes for each damping model.



The Damping page when loss factor damping is selected.

Note: Loss factor damping and equivalent viscous damping are valid only for frequency response analysis. If you choose a transient analysis and either of these damping types, COMSOL Multiphysics solves the model with no damping.

Table 8-2 and the following text describe the parameters that define damping:

TABLE 8-2: PARAMETERS FOR DAMPING MODELS

PARAMETER	VARIABLE	DESCRIPTION	DAMPING MODEL
α_{dM}	`alphadM`	Mass-damping parameter	Rayleigh
β_{dK}	`betadK`	Stiffness-damping parameter	Rayleigh
η	`eta`	Loss factor	Loss factor, Equivalent viscous

Mass damping parameter Defines the Rayleigh damping model's mass damping, α_{dM}.

Stiffness damping parameter Defines the Rayleigh damping model's stiffness damping, β_{dK}.

Loss factor Defines the loss factor η for the loss factor damping and equivalent viscous damping models.

The Piezo Solid Application Mode

Use the Piezo Solid application mode for analysis of 3D structures that exhibit piezoelectric effects.

VARIABLES AND SPACE DIMENSIONS

The degrees of freedom (dependent variables) are the global displacements u, v, and w in the global x, y, and z directions, and the electric potential, V.

PDE FORMULATION

The implementation of this application mode uses the principle of virtual work, described in general terms in the section "Implementation" on page 146.

APPLICATION MODE VARIABLES

For information about available application mode variables, see Table 8-3 on page 203.

The Piezo Plane Strain Application Mode

Use the Piezo Plane Strain application mode to compute the global displacements (u, v) in the x and y directions and the electric potential for a piezoelectric structure in a state of plane strain. The plane strain condition assumes that the ε_z, ε_{yz}, and ε_{xz} components of the strain tensor are zero.

VARIABLES AND SPACE DIMENSIONS

The degrees of freedom (dependent variables) are the global displacements u and v in the global x and y directions, and the electric potential V.

PDE FORMULATION

The implementation of this application mode uses the principle of virtual work, described in general terms in the section "Implementation" on page 146.

Application Mode Parameters

For details about the application mode parameters that define the loads, charges, material properties, constraints, and electric boundary conditions, see the sections earlier in this chapter.

APPLICATION MODE VARIABLES

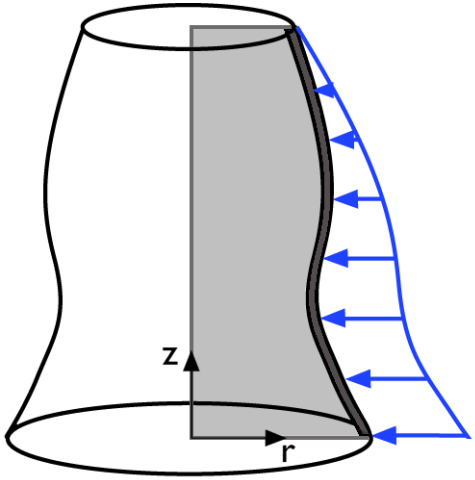
For information about available application mode variables, see Table 8-4 on page 209.

The Piezo Axial Symmetry Application Mode

Use the Piezo Axial Symmetry application mode to analyze axisymmetric models of materials showing piezoelectric effects.

This application mode uses cylindrical the coordinates r, φ (`phi`), and z. It solves the equations for the global displacement (u, w) in the r and z directions. It assumes that the displacement v in the φ direction together with the $\tau_{r\varphi}$, $\tau_{\varphi z}$, $\gamma_{r\varphi}$, and $\gamma_{\varphi z}$ components of the stresses and strains are zero. Loads are independent of φ, and it allows loads only in the r and z directions.

You can consider the domain where the software solves the equations as the intersection between the original axially symmetric 3D solid and the half plane $\varphi = 0$, $r \geq 0$. Therefore it is necessary to draw the geometry only in the half plane $r \geq 0$. The software recovers the original 3D solid by rotating the 2D geometry about the z-axis as seen in the following figure:



The strain-displacement relations for the axial symmetry case for small displacements are:

$$\varepsilon_r = \frac{\partial u}{\partial r} \qquad \varepsilon_\varphi = \frac{u}{r} \qquad \varepsilon_z = \frac{\partial w}{\partial z} \qquad \gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r}$$

To avoid division by r (which causes problems on the axis, where $r = 0$), the program automatically transforms the equations by multiplying by r. When using the principle of virtual work, you normally do not think of this multiplication as a transformation

but merely as an integration around the circumference. Integrating over the volume, you must multiply the integrand by $2\pi r$. The application mode introduces and solves for a new dependent variable

$$\mathrm{uor} = \frac{u}{r}$$

instead of the true radial displacement, u.

Note: $r = 0$ is the symmetry axis. x -> r and y -> z in the Piezo Axial Symmetry, application mode.

VARIABLES AND SPACE DIMENSIONS

The degrees of freedom (dependent variables) are uor the radial displacement divided by r and w the global displacement in the z direction and the electric potential V.

PDE FORMULATION

The implementation of this application mode uses the principle of virtual work, described in general terms in the section "Implementation" on page 146.

Application Mode Parameters

For details about the application mode parameters that define the loads, charges, material properties, constraints, and electric boundary conditions, see the sections earlier in this chapter.

APPLICATION MODE VARIABLES

For information about available application mode variables, see Table 8-5 on page 216.

Application Mode Variables

A large number of variables are available for use in expressions and for postprocessing purposes. In addition to the variables listed below, almost all application mode parameters are available as variables. Some variables change their availability with the type of analysis, as noted in the Analysis column. For frequency-response analysis a number of additional variables are available. Furthermore, the amplitudes and phases

of variables such as strains and stresses are available; to access them, append `_amp` or `_ph` to the variable name. For example:

- `sx_amp` represents the amplitude of the normal stress in the x direction
- `ex_ph` represents the phase of the normal strain in the x direction

The exception to this scheme consists of variables defined using a nonlinear operator such as `mises`, `disp`, `Tresca`, or `s1`.

The tables use a convention where indices $i, j, \ldots$ (or `i`, `j`, ...) run over the geometry's Cartesian coordinate axes, x, y, and z. In particular, u_i (`ui`) refers to the global displacements (u, v, w). The Analysis column uses the following abbreviations:

ANALYSIS	ABBREVIATION
Static	S
Frequency response	F
Time dependent	T

TABLE 8-3: PIEZO SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
`ui`	u_i	All	All	x_i displacement	u_i
`V`	V	All	All	Electric potential	V
`uit`	u_{it}	T	All	x_i velocity	u_{it}
`ui_amp`	$u_{i\mathrm{amp}}$	F	All	x_i displacement amplitude	$\lvert u_i \rvert$
`ui_ph`	$u_{i\mathrm{ph}}$	F	All	x_i displacement phase	$\frac{180^\circ}{\pi}\mathrm{mod}(\mathrm{angle}(u_i), 2\pi)$
`V_amp`	V_{amp}	F	All	Electric potential amplitude	$\lvert V \rvert$
`V_ph`	V_{ph}	F	All	Electric potential phase	$\frac{180^\circ}{\pi}\mathrm{mod}(\mathrm{angle}(V), 2\pi)$
`ui_t`	u_{it}	F	All	x_i velocity	$j\omega u_i$
`ui_t_amp`	$u_{it\mathrm{amp}}$	F	All	x_i velocity amplitude	$\omega u_{i\mathrm{amp}}$

TABLE 8-3: PIEZO SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
u*i*_t_ph	u_{itph}	F	All	x_i velocity phase	$\mathrm{mod}(u_{iph}+90°, 360°)$
u*i*_tt	u_{itt}	F	All	x_i acceleration	$-\omega^2 u_i$
u*i*_tt_amp	u_{ittamp}	F	All	x_i acceleration amplitude	$\omega^2 u_{iamp}$
u*i*_tt_ph	u_{ittph}	F	All	x_i acceleration phase	$\mathrm{mod}(u_{iph}+180°, 360°)$
disp	disp	All	All	Total displacement	$\sqrt{\sum_i (\mathrm{real}(u_i))^2}$
e*i*	ε_i	All	S	ε_i normal strain global coord. system	$\frac{\partial u_i}{\partial x_i}$
e*ij*	ε_{ij}	All	S	ε_{ij} shear strain global coord. system	$\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$
E*i*	E_i	All	S	Electric field	$-\left(\frac{\partial V}{\partial x_i}\right)$
normE	E_i	All	S	Electric field	$\sqrt{\mathbf{E}\cdot\mathbf{E}}$
e*i*l	ε_{il}	All	S	ε_{il} normal strain, user-defined coord. system	$T_{coord}{}^T \varepsilon T_{coord}$
e*ij*l	ε_{ijl}	All	S	ε_{ijl} shear strain, user-defined coord. system	$T_{coord}{}^T \varepsilon T_{coord}$
E*i*l	E_{il}	All	S	Electric field, user-defined coord. system	$T_{coord}{}^T \mathbf{E}$
V*i*l	V_{il}	All	S	Electric potential gradient, user-defined coord. system	$T_{coord}{}^T \nabla V$

TABLE 8-3: PIEZO SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
e*i*_t	ε_{it}	T	S	ε_{it} normal velocity strain, global system	$\frac{\partial u_{it}}{\partial x_i}$
e*i*_t	ε_{it}	F	S	ε_{it} normal velocity strain, global system	$\frac{\partial u_i}{\partial x_i} j\omega$
e*ij*_t	ε_{ijt}	T	S	ε_{ijt} shear velocity strain, global coord. system	$\frac{1}{2}\left(\frac{\partial u_{it}}{\partial x_j} + \frac{\partial u_{jt}}{\partial x_i}\right)$
e*ij*_t	ε_{ijt}	F	S	ε_{ijt} shear velocity strain, global coord. system	$\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) j\omega$
e*i*l_t	ε_{ilt}	F T	S	ε_{ilt} normal velocity strain, user-defined coord. system	$T_{\text{coord}}{}^T \varepsilon_t T_{\text{coord}}$
e*ij*l_t	ε_{ijlt}	F T	S	ε_{ijlt} shear velocity strain, user-defined coord. system	$T_{\text{coord}}{}^T \varepsilon_t T_{\text{coord}}$
cE	c_E	All	S	Stiffness matrix components	s_E^{-1}, if material is specified on strain-charge form, calculated by a special inverting-matrices element.
e	e	All	S	Piezoelectric coupling matrix, if material is specified on strain-charge form	ds_E^{-1}
epsilonT	ε_T	All	S	Electric permittivity with stress field constant	$\varepsilon_0 \varepsilon_{rT}$

TABLE 8-3: PIEZO SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
epsilonS	ε_S	All	S	Electric permittivity with strain field constant	If material defined on stress-charge from $\varepsilon_0\varepsilon_{rS}$ If material defined on strain-charge from $\varepsilon_0\varepsilon_{rT} - d \cdot s_E^{-1} \cdot d^t$
D	D	All	S	Stiffness matrix components	For isotropic and anisotropic material
epsilon	ε_e	All	S	Electric permittivity matrix components	$\varepsilon_0\varepsilon_r$, for isotropic and anisotropic material
sigma	σ_e	freq	S	Electric conductivity matrix components	For isotropic and anisotropic material
s*i*	σ_i	All	S	σ_i normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_1{T_{\text{coord}}}^T$
D*i*	D_i	All	S	Electric displacement, x_i component	If material defined in global coord. sys. $e\varepsilon + \varepsilon_S\mathbf{E}$ or $\varepsilon_e\mathbf{E}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{D}_1$
J*i*	J_i	T F	S	Total current density, x_i component	$J_{d,i} + J_{p,i}$ or $J_{d,i}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{J}_1$
Jd*i*	$J_{d,i}$	T	S	Displacement current density, x_i component	$\frac{\partial D_i}{\partial t}$
Jd*i*	$J_{d,i}$	F	S	Displacement current density, x_i component	$j\omega D_i$

TABLE 8-3: PIEZO SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
Jp*i*	$J_{p,i}$	T F	S	Potential current density, x_i component	$\sigma_e\mathbf{E}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{J}_{\text{l}}$
s*ij*	τ_{ij}	All	S	τ_{ij} shear stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_{\text{l}}{T_{\text{coord}}}^T$
s*i*l	σ_i	All	S	σ_i normal stress, user-defined local coord. system	$c_E\varepsilon_{\text{l}} - e^t\mathbf{E}_{\text{l}}$ or $D\varepsilon_{\text{l}}$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon_{\text{l}} - e^t\mathbf{E}_{\text{l}}$ or $(1+j\eta)D\varepsilon_{\text{l}}$
D*i*l	$D_{i\text{l}}$	All	S	Electric displacement, x_i component, local coord. sys.	$e\varepsilon_{\text{l}} + \varepsilon_S\mathbf{E}_{\text{l}}$ or $\varepsilon_e\mathbf{E}_{\text{l}}$
J*i*l	$J_{i\text{l}}$	T F	S	Total current density, x_i component, local coord. sys.	$J_{d,i\text{l}} + J_{p,i\text{l}}$ or $J_{d,i\text{l}}$
Jd*i*l	$J_{d,i\text{l}}$	T	S	Displacement current density, x_i component, local coord. sys.	$\dfrac{\partial D_{i\text{l}}}{\partial t}$
Jd*i*l	$J_{d,i\text{l}}$	F	S	Displacement current density, x_i component, local coord. sys.	$j\omega D_{i\text{l}}$
Jp*i*l	$J_{p,i\text{l}}$	F	S	Potential current density, x_i component, local coord. sys.	$\sigma_e\mathbf{E}_{\text{l}}$

TABLE 8-3: PIEZO SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
sijl	τ_{ij}	All	S	τ_{ij} shear stress, user-defined local coord. system	$c_E\varepsilon_l - e^t\mathbf{E}_l$ or $D\varepsilon_l$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon_l - e^t\mathbf{E}_l$ or $(1+j\eta)D\varepsilon_l$
si_t	σ_{it}	F T	S	σ_{it} time derivative of normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon_t$ or $D\varepsilon_t$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_t$ or $(1+j\eta)j\omega D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_{lt}{T_{\text{coord}}}^T$
sij_t	τ_{ijt}	T	S	τ_{ijt} time derivative of shear stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon_t$ or $D\varepsilon_t$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon$ or $(1+j\eta)j\omega D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_{lt}{T_{\text{coord}}}^T$
sil_t	σ_{ilt}	F T	S	σ_{ilt} time derivative of normal stress, user-defined local coord. system	$c_E\varepsilon_{lt}$ or $D\varepsilon_{lt}$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_l$ or $(1+j\eta)j\omega D\varepsilon_l$
sijl_t	τ_{ijlt}	F T	S	τ_{ijlt} time derivative of shear stress, user-defined local coord. system	$c_E\varepsilon_{lt}$ or $D\varepsilon_{lt}$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_l$ or $(1+j\eta)j\omega D\varepsilon_l$
si	σ_i	All	S	Principal stresses, i=1,2,3	Defined by elpric element
ei	ε_i	All	S	Principal strains, i=1,2,3	Defined by elpric element

TABLE 8-3: PIEZO SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
sixj	σ_{ixj}	All	S	Principal stress directions, i,j=1,2,3	Defined by elpric element
eixj	ε_{ixj}	All	S	Principal strain directions, i,j=1,2,3	Defined by elpric element
tresca	σ_{tresca}	All	S	Tresca stress	$\max(\max(\lvert\sigma_1-\sigma_2\rvert, \lvert\sigma_2-\sigma_3\rvert)), \lvert\sigma_1-\sigma_3\rvert))$
mises	σ_{mises}	All	S	von Mises stress	
normD	$\lvert\mathbf{D}\rvert$	All	S	Electric displacement, norm	$\sqrt{\mathbf{D}\cdot\mathbf{D}}$
Ws	W_{s}	All	S	Strain energy density	If material properties defined in global coord. sys. $0.5\sigma\cdot\varepsilon$ $\frac{\sigma\cdot\varepsilon}{2}, \frac{1}{2}\text{real}(\sigma\cdot\text{conj}(\varepsilon))$ in frequency response analyses If material properties defined in local user-defined coord. sys. $\frac{\sigma_l\cdot\varepsilon_l}{2}, \frac{1}{2}\text{real}(\sigma_l\cdot\text{conj}(\varepsilon_l))$ in freq. resp.
We	W_{e}	All	S	Electric energy density	If material properties defined in global coord. sys. $\mathbf{E}\cdot\mathbf{D}/2$, $\text{real}(\text{conj}(\mathbf{E})\cdot\mathbf{D})/2$ in freq. resp. If material properties defined in local user-defined coord. sys. $\mathbf{E}_l\cdot\mathbf{D}_l/2$, $\text{real}(\text{conj}(\mathbf{E}_l)\cdot\mathbf{D}_l)/2$ in freq. resp.
Tai	Ta_i	All	B	Surface traction (force/area) in x_i direction	$\begin{bmatrix}\text{Ta}_x\\ \text{Ta}_y\\ \text{Ta}_z\end{bmatrix} = \begin{bmatrix}\sigma_x & \tau_{xy} & \tau_{xz}\\ \tau_{xy} & \sigma_y & \tau_{yz}\\ \tau_{xz} & \tau_{yz} & \sigma_z\end{bmatrix}\begin{bmatrix}n_x\\ n_y\\ n_z\end{bmatrix}$

TABLE 8-3: PIEZO SOLID APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
nD	nD	All	B	Surface charge density	$\mathbf{n}_{\text{up}} \cdot (\mathbf{D}_{\text{down}} - \mathbf{D}_{\text{up}})$
nJ	nJ	F T	B	Current density outflow	$\mathbf{n} \cdot \mathbf{J}$
nJs	nJs	F	B	Source current density	Only for unsymmetric electric currents. $\mathbf{n}_{\text{up}} \cdot (\mathbf{J}_{\text{down}} - \mathbf{J}_{\text{up}})$ or, with weak constraints, the Lagrange multiplier for V.
F*i*g	F_{ig}	All	All	Body load, face load, edge load, point load, in global x_i direction	If global coordinate system $\begin{bmatrix} F_{x\text{g}} \\ F_{y\text{g}} \\ F_{z\text{g}} \end{bmatrix} = \begin{bmatrix} F_x \\ F_y \\ F_z \end{bmatrix}$ If other coordinate system $\begin{bmatrix} F_{x\text{g}} \\ F_{y\text{g}} \\ F_{z\text{g}} \end{bmatrix} = T_{\text{coord}} \begin{bmatrix} F_x \\ F_y \\ F_z \end{bmatrix}$
smon	smon	All	S	Structural equation available	1 or 0
eson	eson	All	S	Electrical equation available	1 or 0

PIEZO PLANE STRAIN

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
u*i*	u_i	All	All	x_i displacement	u_i
V	V	All	All	Electric potential	V
u*i*t	u_{it}	T	All	x_i velocity	u_{it}
u*i*_amp	$u_{i\text{amp}}$	F	All	x_i displacement amplitude	$\lvert u_i \rvert$

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
u*i*_ph	$u_{i\mathrm{ph}}$	F	All	x_i displacement phase	$\frac{180^\circ}{\pi}\mathrm{mod}(\mathrm{angle}(u_i), 2\pi)$
V_amp	V_{amp}	F	All	Electric potential amplitude	$\lvert V\rvert$
V_ph	V_{ph}	F	All	Electric potential phase	$\frac{180^\circ}{\pi}\mathrm{mod}(\mathrm{angle}(V), 2\pi)$
u*i*_t	u_{it}	F	All	x_i velocity	$j\omega u_i$
u*i*_t_amp	$u_{it\mathrm{amp}}$	F	All	x_i velocity amplitude	$\omega u_{i\mathrm{amp}}$
u*i*_t_ph	$u_{it\mathrm{ph}}$	F	All	x_i velocity phase	$\mathrm{mod}(u_{i\mathrm{ph}} + 90^\circ, 360^\circ)$
u*i*_tt	u_{itt}	F	All	x_i acceleration	$-\omega^2 u_i$
u*i*_tt_amp	$u_{itt\mathrm{amp}}$	F	All	x_i acceleration amplitude	$\omega^2 u_{i\mathrm{amp}}$
u*i*_tt_ph	$u_{itt\mathrm{ph}}$	F	All	x_i acceleration phase	$\mathrm{mod}(u_{i\mathrm{ph}} + 180^\circ, 360^\circ)$
disp	disp	All	All	Total displacement	$\sqrt{\sum_i (\mathrm{real}(u_i))^2}$
e*i*	ε_i	All	S	ε_i normal strain, global coord. system	$\frac{\partial u_i}{\partial x_i}$
exy	ε_{xy}	All	S	ε_{xy} shear strain, global coord. system	$\frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)$
E*i*	E_i	All	S	Electric field	$-\left(\frac{\partial V}{\partial x_i}\right)$
normE	E_i	All	S	Electric field	$\sqrt{\mathbf{E}\cdot\mathbf{E}}$
e*i*l	$\varepsilon_{i\mathrm{l}}$	All	S	ε_{il} normal strain, user-defined coord. system	$T_{\mathrm{coord}}{}^T \varepsilon T_{\mathrm{coord}}$

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
eijl	ε_{ijl}	All	S	ε_{ijl} shear strain, user-defined coord. system	$T_{\text{coord}}{}^{T}\varepsilon T_{\text{coord}}$
Eil	E_{il}	All	S	Electric field, user-defined coord. system	$T_{\text{coord}}{}^{T}\mathbf{E}$
Vil	V_{il}	All	S	Electric potential gradient, user-defined coord. system	$T_{\text{coord}}{}^{T}\nabla V$
ei_t	ε_{it}	T	S	ε_{it} normal velocity strain, global system	$\frac{\partial u_{it}}{\partial x_i}$
ei_t	ε_{it}	F	S	ε_{it} normal velocity strain, global system	$\frac{\partial u_i}{\partial x_i} j\omega$
exy_t	ε_{xyt}	T	S	ε_{xyt} shear velocity strain, global coord. system	$\frac{1}{2}\left(\frac{\partial u_t}{\partial y}+\frac{\partial v_t}{\partial x}\right)$
exy_t	ε_{xyt}	F	S	ε_{xyt} shear velocity strain, global coord. system	$\frac{1}{2}\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right) j\omega$
eil_t	ε_{ilt}	F T	S	ε_{ilt} normal velocity strain, user-defined coord. system	$T_{\text{coord}}{}^{T}\varepsilon_t T_{\text{coord}}$
exyl_t	ε_{xylt}	F T	S	ε_{xylt} shear velocity strain, user-defined coord. system	$T_{\text{coord}}{}^{T}\varepsilon_t T_{\text{coord}}$
cE	c_E	All	S	Stiffness matrix components	s_E^{-1}, if material is specified on strain-charge form, calculated by a special inverting-matrices element.

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
e	e	All	S	Piezoelectric coupling matrix if material is specified on strain-charge form	ds_E^{-1}
epsilonT	ε_T	All	S	Electric permittivity with stress field constant	$\varepsilon_0\varepsilon_{rT}$
epsilonS	ε_S	All	S	Electric permittivity with strain field constant	If material defined on stress-charge from $\varepsilon_0\varepsilon_{rS}$ If material defined on strain-charge from $\varepsilon_0\varepsilon_{rT} - d \cdot s_E^{-1} \cdot d^t$
D	D	All	S	Stiffness matrix components	For isotropic and anisotropic material
epsilon	ε_e	All	S	Electric permittivity matrix components	$\varepsilon_0\varepsilon_r$, for isotropic and anisotropic material
sigma	σ_e	freq	S	Electric conductivity matrix components	For isotropic and anisotropic material
s*i*	σ_i	All	S	σ_i normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_1{T_{\text{coord}}}^T$
D*i*	D_i	All	S	Electric displacement, x_i component	If material defined in global coord. sys. $e\varepsilon + \varepsilon_S\mathbf{E}$ or $\varepsilon_e\mathbf{E}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{D}_1$

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
`Ji`	J_i	T F	S	Total current density, x_i component	$J_{d,i} + J_{p,i}$ or $J_{d,i}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{J}_\text{l}$
`Jdi`	$J_{d,i}$	T	S	Displacement current density, x_i component	$\frac{\partial D_i}{\partial t}$
`Jdi`	$J_{d,i}$	F	S	Displacement current density, x_i component	$j\omega D_i$
`Jpi`	$J_{p,i}$	T F	S	Potential current density, x_i component	$\sigma_e\mathbf{E}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{J}_\text{l}$
`sij`	τ_{ij}	All	S	τ_{ij} shear stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_\text{l}{T_{\text{coord}}}^T$
`sil`	σ_i	All	S	σ_i normal stress, user-defined local coord. system	$c_E\varepsilon_\text{l} - e^t\mathbf{E}_\text{l}$ or $D\varepsilon_\text{l}$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon_\text{l} - e^t\mathbf{E}_\text{l}$ or $(1+j\eta)D\varepsilon_\text{l}$
`Dil`	$D_{i\text{l}}$	All	S	Electric displacement, x_i component, local coord. sys.	$e\varepsilon_\text{l} + \varepsilon_S\mathbf{E}_\text{l}$ or $\varepsilon_e\mathbf{E}_\text{l}$
`Jil`	$J_{i\text{l}}$	T F	S	Total current density, x_i component, local coord. sys.	$J_{d,i\text{l}} + J_{p,i\text{l}}$ or $J_{d,i\text{l}}$
`Jdil`	$J_{d,i\text{l}}$	T	S	Displacement current density, x_i component, local coord. sys.	$\frac{\partial D_{i\text{l}}}{\partial t}$

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
Jd*i*l	$J_{d,i\mathrm{l}}$	F	S	Displacement current density, x_i component, local coord. sys.	$j\omega D_{i\mathrm{l}}$
Jp*i*l	$J_{p,i\mathrm{l}}$	F	S	Potential current density, x_i component, local coord. sys.	$\sigma_e \mathbf{E}_\mathrm{l}$
s*ij*l	τ_{ij}	All	S	τ_{ij} shear stress, user-defined local coord. system	$c_E\varepsilon_\mathrm{l} - e^t\mathbf{E}_\mathrm{l}$ or $D\varepsilon_\mathrm{l}$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon_\mathrm{l} - e^t\mathbf{E}_\mathrm{l}$ or $(1+j\eta)D\varepsilon_\mathrm{l}$
s*i*_t	σ_{it}	F T	S	σ_{it} time derivative of normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon_t$ or $D\varepsilon_t$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_t$ or $(1+j\eta)j\omega D\varepsilon$ If material defined in user-def. coord. sys. $T_\mathrm{coord}\sigma_{\mathrm{l}t}{T_\mathrm{coord}}^T$
s*ij*_t	τ_{ijt}	T	S	τ_{ijt} time derivative of shear stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon_t$ or $D\varepsilon_t$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon$ or $(1+j\eta)j\omega D\varepsilon$ If material defined in user-def. coord. sys. $T_\mathrm{coord}\sigma_{\mathrm{l}t}{T_\mathrm{coord}}^T$
s*i*l_t	$\sigma_{i\mathrm{l}t}$	F T	S	$\sigma_{i\mathrm{l}t}$ time derivative of normal stress, user-defined local coord. system	$c_E\varepsilon_{\mathrm{l}t}$ or $D\varepsilon_{\mathrm{l}t}$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_\mathrm{l}$ or $(1+j\eta)j\omega D\varepsilon_\mathrm{l}$

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
sijl_t	τ_{ijlt}	F T	S	τ_{ijlt} time derivative of shear stress, user-defined local coord. system	$c_E\varepsilon_{1t}$ or $D\varepsilon_{1t}$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_1$ or $(1+j\eta)j\omega D\varepsilon_1$
sz	σ_z	All	S	σ_z normal stress	If material defined in global coord. sys. $\sum_k (c_E)_{3k}\varepsilon_k - \sum_j e_{j3}E_j$, or $\sum_k (D)_{3k}\varepsilon_k$ With loss factor damping in frequency response analysis $\sum_k (1+j\eta)(c_E)_{3k}\varepsilon_k - \sum_j e_{j3}E_j$, or $\sum_k (1+j\eta)(D)_{3k}\varepsilon_k$ If material defined in user-def. coord. sys. $\sum_k (c_E)_{3k}(\varepsilon_1)_k - \sum_j e_{j3}(E_1)_j$, or $\sum_k (D)_{3k}(\varepsilon_1)_k$
sz_t	σ_{zt}	All	S	σ_{zt} time derivative of normal stress	If material defined in global coord. sys. $\sum_k (D)_{3k}(\varepsilon_t)_k$ (M is c_E or D) With loss factor damping in frequency response analysis $\sum_k (1+j\eta)(M)_{3k}j\omega\varepsilon_k$ (M is c_E or D) If material defined in user-def. coord. sys. $\sum_k (M)_{3k}(\varepsilon_{1t})_k$ (M is c_E or D)

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
s*i*	σ_i	All	S	Principal stresses, $i=1,2,3$	Defined by elpric element
e*i*	ε_i	All	S	Principal strains, $i=1,2,3$	Defined by elpric element
s*i*x*j*	σ_{ixj}	All	S	Principal stress directions, $i,j=1,2,3$	Defined by elpric element
e*i*x*j*	ε_{ixj}	All	S	Principal strain directions, $i,j=1,2,3$	Defined by elpric element
tresca	σ_{tresca}	All	S	Tresca stress	$\max(\max(\lvert\sigma_1-\sigma_2\rvert, \lvert\sigma_2-\sigma_3\rvert)), \lvert\sigma_1-\sigma_3\rvert))$
mises	σ_{mises}	All	S	von Mises stress	
normD	normD	All	S	Electric displacement, norm	$\sqrt{\mathbf{D}\cdot\mathbf{D}}$
Ws	W_s	All	S	Strain energy density	If material properties defined in global coord. sys. $\frac{\sigma\cdot\varepsilon}{2}\text{th}$, $\frac{1}{2}\text{real}(\sigma\cdot\text{conj}(\varepsilon))\text{th}$ in frequency response analyses. If material properties defined in local user-defined coord. sys. $\frac{\sigma_l\cdot\varepsilon_l}{2}\text{th}$, $\frac{1}{2}\text{real}(\sigma_l\cdot\text{conj}(\varepsilon_l))\text{th}$ in freq. resp.

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
We	W_e	All	S	Electric energy density	If material properties defined in global coord. sys. $\frac{\mathbf{E}\cdot\mathbf{D}}{2}\mathrm{th}$, $\frac{1}{2}\mathrm{real}(\mathrm{conj}(\mathbf{E})\cdot\mathbf{D})\mathrm{th}$ in frequency response analyses. If material properties defined in local user-defined coord. sys. $\frac{\mathbf{E}_\mathrm{l}\cdot\mathbf{D}_\mathrm{l}}{2}\mathrm{th}$, $\frac{1}{2}\mathrm{real}(\mathrm{conj}(\mathbf{E}_\mathrm{l})\cdot\mathbf{D}_\mathrm{l})\mathrm{th}$ in frequency response analyses.
Tai	Ta_i	All	B	Surface traction (force/area) in x_i direction	$\begin{bmatrix}\mathrm{Ta}_x\\ \mathrm{Ta}_y\end{bmatrix} = \begin{bmatrix}\sigma_x & \tau_{xy}\\ \tau_{xy} & \sigma_y\end{bmatrix}\begin{bmatrix}n_x\\ n_y\end{bmatrix}$
nD	nD	All	B	Surface charge density	$\mathbf{n}_\mathrm{up}\cdot(\mathbf{D}_\mathrm{down}-\mathbf{D}_\mathrm{up})$
nJ	nJ	F T	B	Current density outflow	$\mathbf{n}\cdot\mathbf{J}$
nJs	nJs	F	B	Source current density	Only for unsymmetric electric currents. $\mathbf{n}_\mathrm{up}\cdot(\mathbf{J}_\mathrm{down}-\mathbf{J}_\mathrm{up})$ or, with weak constraints, the Lagrange multiplier for V.
Fig	F_{ig}	All	All	Body load, edge load, point load, in global x_i direction	If global coordinate system $\begin{bmatrix}F_{x\mathrm{g}}\\ F_{y\mathrm{g}}\end{bmatrix} = \begin{bmatrix}F_x\\ F_y\end{bmatrix}$ If other coordinate system $\begin{bmatrix}F_{x\mathrm{g}}\\ F_{y\mathrm{g}}\end{bmatrix} = T_\mathrm{coord}\begin{bmatrix}F_x\\ F_y\end{bmatrix}$

TABLE 8-4: PIEZO PLANE STRAIN APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
smon	smon	All	S	Structural equation available	1 or 0
eson	eson	All	S	Electrical equation available	1 or 0

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
uor	uor	All	All	r displacement divided by r	uor
uaxi	uaxi	All	All	r displacement	$\text{uor}\cdot r$
w	w	All	All	z displacement	w
V	V	All	All	Electric potential	V
uort	uor_t	T	All	r velocity divided by r	uor_t
uaxi_t	uaxi_t	T	All	r velocity	$\text{uor}_t\cdot r$
w_t	w_t	T	All	z velocity	w_t
uaxi_amp	uaxi_{amp}	F	All	r displacement amplitude	$\lvert\text{uaxi}\rvert$
w_amp	w_{amp}	F	All	z displacement amplitude	$\lvert w\rvert$
uaxi_ph	uaxi_{ph}	F	All	r displacement phase	$\frac{180^\circ}{\pi}\text{mod}(\text{angle}(\text{uaxi}), 2\pi)$
w_ph	w_{ph}	F	All	z displacement phase	$\frac{180^\circ}{\pi}\text{mod}(\text{angle}(w), 2\pi)$
V_amp	V_{amp}	F	All	Electric potential amplitude	$\lvert V\rvert$
V_ph	V_{ph}	F	All	Electric potential phase	$\frac{180^\circ}{\pi}\text{mod}(\text{angle}(V), 2\pi)$
uaxi_t	uaxi_t	F	All	r velocity	$j\omega\,\text{uaxi}$
w_t	w_t	F	All	z velocity	$j\omega w$

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
uaxi_t_amp	$\mathrm{uaxi}_{t\mathrm{amp}}$	F	All	r velocity amplitude	$\omega\,\mathrm{uaxi}_{\mathrm{amp}}$
w_t_amp	$w_{t\mathrm{amp}}$	F	All	z velocity amplitude	ωw_{amp}
uaxi_t_ph	$\mathrm{uaxi}_{t\mathrm{ph}}$	F	All	r velocity phase	$\mathrm{mod}(\mathrm{uaxi}_{\mathrm{ph}} + 90°, 360°)$
w_t_ph	$w_{t\mathrm{ph}}$	F	All	z velocity phase	$\mathrm{mod}(w_{\mathrm{ph}} + 90°, 360°)$
uaxi_tt	uaxi_{tt}	F	All	r acceleration	$-\omega^2\,\mathrm{uaxi}$
w_tt	w_{tt}	F	All	z acceleration	$-\omega^2 w$
uaxi_tt_amp	$\mathrm{uaxi}_{t\mathrm{amp}}$	F	All	r acceleration amplitude	$\omega^2\,\mathrm{uaxi}_{\mathrm{amp}}$
w_tt_amp	$w_{t\mathrm{amp}}$	F	All	z acceleration amplitude	$\omega^2 w_{\mathrm{amp}}$
uaxi_tt_ph	$\mathrm{uaxi}_{t\mathrm{ph}}$	F	All	r acceleration phase	$\mathrm{mod}(\mathrm{uaxi}_{\mathrm{ph}} + 180°, 360°)$
w_tt_ph	$w_{t\mathrm{ph}}$	F	All	z acceleration phase	$\mathrm{mod}(w_{\mathrm{ph}} + 180°, 360°)$
disp	disp	All	All	Total displacement	$\sqrt{\mathrm{uaxi}^2 + w^2}$
er	ε_r	All	S	ε_r normal strain, global system	$\mathrm{uor} + \frac{\partial}{\partial r}(\mathrm{uor}) \cdot r$
ez	ε_z	All	S	ε_z normal strain, global system	$\frac{\partial w}{\partial z}$
ephi	ε_φ	All	S	ε_φ normal strain	uor
erz	ε_{rz}	All	S	ε_{rz} shear strain, global coord. system	$\frac{1}{2}\left(\frac{\partial}{\partial z}(\mathrm{uor}) \cdot r + \frac{\partial w}{\partial r}\right)$
exl, eyl	ε_{xl}, ε_{yl}	All	S	ε_{xl}, ε_{yl} normal strains, user-defined coord. system	$T_{\mathrm{coord}}{}^T \varepsilon T_{\mathrm{coord}}$
exyl	ε_{xyl}	All	S	ε_{xy} shear strain, user-defined coord. system	$T_{\mathrm{coord}}{}^T \varepsilon T_{\mathrm{coord}}$

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
er_t	ε_{rt}	T	S	ε_{rt} velocity normal strain, global system	$\mathrm{uor}_t + \frac{\partial}{\partial r}(\mathrm{uor}_t) \cdot r$
er_t	ε_{rt}	F	S	ε_{rt} velocity normal strain, global system	$j\omega\left(\mathrm{uor} + \frac{\partial}{\partial r}\mathrm{uor} \cdot r\right)$
ez_t	ε_{zt}	T	S	ε_{zt} velocity normal strain, global system	$\frac{\partial w_t}{\partial z}$
ez_t	ε_{zt}	F	S	ε_{zt} velocity normal strain, global system	$j\omega\left(\frac{\partial w}{\partial z}\right)$
ephi_t	$\varepsilon_{\varphi t}$	T	S	$\varepsilon_{\varphi t}$ velocity normal strain	uor_t
ephi_t	$\varepsilon_{\varphi t}$	F	S	$\varepsilon_{\varphi t}$ velocity normal strain	$j\omega\,\mathrm{uor}$
erz_t	ε_{rzt}	T	S	ε_{rzt} shear strain, global coord. system	$\frac{1}{2}\left(\frac{\partial}{\partial z}(\mathrm{uor}_t) \cdot r + \frac{\partial w_t}{\partial r}\right)$
erz_t	ε_{rzt}	F	S	ε_{rzt} shear strain, global coord. system	$\frac{1}{2}\left(\frac{\partial}{\partial z}(\mathrm{uor}) \cdot r + \frac{\partial w}{\partial r}\right)j\omega$
exl_t, eyl_t	ε_{xlt}, ε_{ylt}	F T	S	ε_{xlt}, ε_{ylt} velocity normal strain, user-defined coord. system	$T_{\mathrm{coord}}{}^{T}\varepsilon_t T_{\mathrm{coord}}$
exyl_t	ε_{xylt}	F T	S	ε_{xylt} velocity shear strain, user-defined coord. system	$T_{\mathrm{coord}}{}^{T}\varepsilon_t T_{\mathrm{coord}}$
disp	disp	All	All	Total displacement	$\sqrt{\sum_i (\mathrm{real}(u_i))^2}$
Ei	E_i	All	S	Electric field	$-\left(\frac{\partial V}{\partial x_i}\right)$
normE	$\lvert\mathbf{E}\rvert$	All	S	Electric field	$\sqrt{\mathbf{E} \cdot \mathbf{E}}$

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
Eil	E_{il}	All	S	Electric field, user-defined coord. system	${T_{\text{coord}}}^T \mathbf{E}$
Vil	V_{il}	All	S	Electric potential gradient, user-defined coord. system	${T_{\text{coord}}}^T \nabla V$
cE	c_E	All	S	Stiffness matrix components	s_E^{-1} , if material is specified on strain-charge form, calculated by a special inverting-matrices element.
e	e	All	S	Piezoelectric coupling matrix if material is specified on strain-charge form	ds_E^{-1}
epsilonT	ε_T	All	S	Electric permittivity with stress field constant	$\varepsilon_0\varepsilon_{rT}$
epsilonS	ε_S	All	S	Electric permittivity with strain field constant	If material defined on stress-charge from $\varepsilon_0\varepsilon_{rS}$ If material defined on strain-charge from $\varepsilon_0\varepsilon_{rT} - d \cdot s_E^{-1} \cdot d^t$
D	D	All	S	Stiffness matrix components	For isotropic and anisotropic material
epsilon	ε_e	All	S	Electric permittivity matrix components	$\varepsilon_0\varepsilon_r$, for isotropic and anisotropic material
sigma	σ_e	freq	S	Electric conductivity matrix components	For isotropic and anisotropic material

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
sr, sz	σ_r, σ_z	All	S	$\sigma_{r,z}$ normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_{\text{l}}{T_{\text{coord}}}^T$
sphi	σ_φ	All	S	σ_φ normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $c_E\varepsilon_{\text{l}} - e^t\mathbf{E}_{\text{l}}$
srz	τ_{rz}	All	S	τ_{rz} shear stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_{\text{l}t}{T_{\text{coord}}}^T$

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
s*i*	σ_i	All	S	σ_i normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_1 {T_{\text{coord}}}^T$
D*i*	D_i	All	S	Electric displacement, x_i component	If material defined in global coord. sys. $e\varepsilon + \varepsilon_S\mathbf{E}$ or $\varepsilon_e\mathbf{E}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{D}_1$
J*i*	J_i	T F	S	Total current density, x_i component	$J_{d,i} + J_{p,i}$ or $J_{d,i}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{J}_1$
Jd*i*	$J_{d,i}$	T	S	Displacement current density, x_i component	$\frac{\partial D_i}{\partial t}$
Jd*i*	$J_{d,i}$	F	S	Displacement current density, x_i component	$j\omega D_i$
Jp*i*	$J_{p,i}$	T F	S	Potential current density, x_i component	$\sigma_e\mathbf{E}$ If material defined in user-def. coord. sys. $T_{\text{coord}}\mathbf{J}_1$

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
sij	τ_{ij}	All	S	τ_{ij} shear stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon - e^t\mathbf{E}$ or $D\varepsilon$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon - e^t\mathbf{E}$ or $(1+j\eta)D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_1{T_{\text{coord}}}^T$
sil	σ_i	All	S	σ_i normal stress, user-defined local coord. system	$c_E\varepsilon_1 - e^t\mathbf{E}_1$ or $D\varepsilon_1$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon_1 - e^t\mathbf{E}_1$ or $(1+j\eta)D\varepsilon_1$
Dil	D_{i1}	All	S	Electric displacement, x_i component, local coord. sys.	$e\varepsilon_1 + \varepsilon_S\mathbf{E}_1$ or $\varepsilon_e\mathbf{E}_1$
Jil	J_{i1}	T F	S	Total current density, x_i component, local coord. sys.	$J_{d,i1} + J_{p,i1}$ or $J_{d,i1}$
Jdil	$J_{d,i1}$	T	S	Displacement current density, x_i component, local coord. sys.	$\frac{\partial D_{i1}}{\partial t}$
Jdil	$J_{d,i1}$	F	S	Displacement current density, x_i component, local coord. sys.	$j\omega D_{i1}$
Jpil	$J_{p,i1}$	F	S	Potential current density, x_i component, local coord. sys.	$\sigma_e\mathbf{E}_1$

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
s*ij*l	τ_{ij}	All	S	τ_{ij} shear stress, user-defined local coord. system	$c_E\varepsilon_l - e^t\mathbf{E}_l$ or $D\varepsilon_l$ With loss factor damping in frequency response analysis $(1+j\eta)c_E\varepsilon_l - e^t\mathbf{E}_l$ or $(1+j\eta)D\varepsilon_l$
s*i*_t	σ_{it}	F T	S	σ_{it} time derivative of normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon_t$ or $D\varepsilon_t$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_t$ or $(1+j\eta)j\omega D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_{lt}{T_{\text{coord}}}^T$
s*ij*_t	τ_{ijt}	T	S	τ_{ijt} time derivative of shear stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon_t$ or $D\varepsilon_t$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon$ or $(1+j\eta)j\omega D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_{lt}{T_{\text{coord}}}^T$
s*i*l_t	σ_{ilt}	F T	S	σ_{ilt} time derivative of normal stress, user-defined local coord. system	$c_E\varepsilon_{lt}$ or $D\varepsilon_{lt}$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_l$ or $(1+j\eta)j\omega D\varepsilon_l$
s*ij*l_t	τ_{ijlt}	F T	S	τ_{ijlt} time derivative of shear stress, user-defined local coord. system	$c_E\varepsilon_{lt}$ or $D\varepsilon_{lt}$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_l$ or $(1+j\eta)j\omega D\varepsilon_l$

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
sr_t, sz_t	σ_{rt},σ_{zt}	F T	S	σ_{rt}, σ_{zt} time derivative of normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon_t$ or $D\varepsilon_t$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_t$ or $(1+j\eta)j\omega D\varepsilon$ If material defined in user-def. coord. sys. $T_{\text{coord}}\sigma_{1t}T_{\text{coord}}{}^T$
sphi_t	$\sigma_{\varphi t}$	F T	S	$\sigma_{\varphi t}$ time derivative of normal stress, global coord. system	If material defined in global coord. sys. $c_E\varepsilon_t$ or $D\varepsilon_t$ With loss factor damping in frequency response analysis $(1+j\eta)j\omega c_E\varepsilon_t$ or $(1+j\eta)j\omega D\varepsilon$ If material defined in user-def. coord. sys. $c_E\varepsilon_{1t}$
s*i*	σ_i	All	S	Principal stresses, i = 1,2,3	Defined by elpric element
e*i*	ε_i	All	S	Principal strains, i = 1,2,3	Defined by elpric element
s*i*x*j*	σ_{ixj}	All	S	Principal stress directions, i,j = 1,2,3	Defined by elpric element
e*i*x*j*	ε_{ixj}	All	S	Principal strain directions, i,j = 1,2,3	Defined by elpric element
tresca	σ_{tresca}	All	S	Tresca stress	$\max(\max(\lvert\sigma_1-\sigma_2\rvert, \lvert\sigma_2-\sigma_3\rvert)), \lvert\sigma_1-\sigma_3\rvert))$
mises	σ_{mises}	All	S	von Mises stress	

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
normD	normD	All	S	Electric displacement, norm	$\sqrt{\mathbf{D}\cdot\mathbf{D}}$
Ws	W_s	All	S	Strain energy density	If material properties defined in global coord. sys. $\frac{\sigma\cdot\varepsilon}{2}$, $\frac{1}{2}\mathrm{real}(\sigma\cdot\mathrm{conj}(\varepsilon))$ in frequency response analyses. If material properties defined in local user-defined coord. sys. $\frac{\sigma_l\cdot\varepsilon_l}{2}$, $\frac{\mathrm{real}(\sigma_l\cdot\mathrm{conj}(\varepsilon_l))}{2}+\frac{\mathrm{real}(\sigma_\varphi\cdot\mathrm{conj}(\varepsilon_\varphi))}{2}$ in freq. resp.
We	W_e	All	S	Electric energy density	If material properties defined in global coord. sys. $\mathbf{E}\cdot\mathbf{D}/2$, $\mathrm{real}(\mathrm{conj}(\mathbf{E})\cdot\mathbf{D})/2$ in freq. resp. If material properties defined in local user-defined coord. sys. $\mathbf{E}_l\cdot\mathbf{D}_l/2$, $\mathrm{real}(\mathrm{conj}(\mathbf{E}_l)\cdot\mathbf{D}_l)/2$ in freq. resp.
Tai	Ta_i	All	B	Surface traction (force/area) in x_i direction	$\begin{bmatrix}\mathrm{Ta}_r\\ \mathrm{Ta}_z\end{bmatrix}=\begin{bmatrix}\sigma_r & \tau_{rz}\\ \tau_{rz} & \sigma_z\end{bmatrix}\begin{bmatrix}n_r\\ n_z\end{bmatrix}$
nD	nD	All	B	Surface charge density	$\mathbf{n}_{\mathrm{up}}\cdot(\mathbf{D}_{\mathrm{down}}-\mathbf{D}_{\mathrm{up}})$
smon	smon	All	S	Structural equation available	1 or 0
eson	eson	All	S	Electrical equation available	1 or 0
nJ	nJ	F T	B	Current density outflow	$\mathbf{n}\cdot\mathbf{J}^d$

TABLE 8-5: PIEZO AXIAL SYMMETRY APPLICATION MODE VARIABLES

NAME	SYMBOL	ANALYSIS	DOMAIN	DESCRIPTION	EXPRESSION
nJs	nJs	F	B	Source current density	Only for unsymmetric electric currents. $\mathbf{n}_{\mathrm{up}} \cdot (\mathbf{J}_{\mathrm{down}} - \mathbf{J}_{\mathrm{up}})$ or, with weak constraints, the Lagrange multiplier for V.
Fig	F_{ig}	All	All	Body load, edge load, point load, in global x_i direction	If global coordinate system $\begin{bmatrix} F_{rg} \\ F_{zg} \end{bmatrix} = \begin{bmatrix} F_r \\ F_z \end{bmatrix}$ If other coordinate system $\begin{bmatrix} F_{xg} \\ F_{zg} \end{bmatrix} = T_{\mathrm{coord}} \begin{bmatrix} F_r \\ F_z \end{bmatrix}$

9

Glossary

This glossary 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, please see the glossary in the *COMSOL Multiphysics User's Guide*. 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: 1 Pa/(m/s).

acoustic reactance The imaginary part of the acoustic impedance.

acoustic resistance The real part of the acoustic impedance.

admittance The reciprocal of impedance.

anisotropy Variation of material properties with direction.

characteristic impedance The product of the equilibrium density and the speed of sound in a medium. SI unit: 1 Pa/(m/s).

compliance Reciprocal of *stiffness*.

damping Dissipation of energy with time or distance.

decibel (dB) Unit of level when the base of the logarithm used in defining the level is the tenth root of ten and the quantities concerned are proportional to power.

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: 1 Pa = 1 N/m^2.

eigenmode A possible propagating mode of an acoustic wave.

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: 1 Pa = 1 N/m^2.

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 nonreflecting boundary condition independently of the shape and frequency of the incident wave front.

reference sound pressure See definition in the entry for *sound pressure level*.

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 $p_{\mathrm{RMS}} = p_0/\sqrt{2}$.

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: 1 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: 1 $\mathrm{W/m^2}$.

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: 1 $\mathrm{W/m^2}$.

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, $p_{\mathrm{ref}} = 20\ \mu\mathrm{Pa}$, for other media (unless otherwise specified) $p_{\mathrm{ref}} = 1\ \mu\mathrm{Pa}$. Unit: 1 dB (*decibel*).

sound source strength Maximum instantaneous rate of volume displacement produced by a source when emitting a harmonic sound wave. SI unit: 1 $\mathrm{m^3/s}$.

specific acoustic impedance At a point in a sound field, the quotient of sound pressure by particle velocity. SI unit: 1 Pa/(m/s).

speed of sound The rate of change of particle displacement with distance for a sound wave. SI unit: 1 m/s.

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.

INDEX