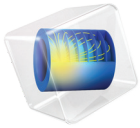


Acoustics Module

User's Guide

6.3

COMSOL

Acoustics Module User's Guide

© 1998–2024 COMSOL

Protected by patents listed on www.comsol.com/patents, or see Help > About COMSOL Multiphysics on the File menu in the COMSOL Desktop for less detailed lists of U.S. Patents that may apply. Patents pending.

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

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

Version: COMSOL 6.3

Contact Information

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

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

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase
- Learning Center: https://www.comsol.com/support/learning-center

Part number: CM020201

Contents

Chapter 1: Introduction

Acoustics Module Capabilities **31**
What Can the Acoustics Module Do? 31
What Are the Application Areas? 34
Which Problems Can You Solve?. 36

Fundamental of Acoustics **38**
Acoustics Explained. 38
Mathematical Models for Acoustic Analysis 39
Damping 41
Artificial Boundaries. 43

Acoustics Module Physics Interface Guide **45**
Common Physics Interface and Feature Settings and Nodes 52
Where Do I Access the Documentation and Application Libraries? 52

Overview of the User's Guide **56**

Chapter 2: Pressure Acoustics Interfaces

The Pressure Acoustics, Frequency Domain Interface **65**
Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface 70
Pressure Acoustics 73
Poroacoustics 80
Narrow Region Acoustics 87
Anisotropic Acoustics 91
Anisotropic Poroacoustics 92
Background Pressure Field 94
Initial Values 97
Monopole Domain Source 97

Dipole Domain Source 98
Heat Source 98
Aeroacoustic Flow Source 99
Flame Model 99
Sound Hard Boundary (Wall) 101
Axial Symmetry 102
Normal Acceleration 102
Normal Velocity 102
Normal Displacement 103
Sound Soft Boundary 104
Pressure 104
Impedance 104
Symmetry 110
Periodic Condition 110
Matched Boundary 113
Exterior Field Calculation 114
Port. 119
Port Reference Axis 127
Lumped Port 127
Lumped Speaker Boundary 131
Thermoviscous Boundary Layer Impedance 133
Transfer Matrix Coupling. 136
Plane Wave Radiation 138
Spherical Wave Radiation 139
Cylindrical Wave Radiation 139
Incident Pressure Field. 140
Perfectly Matched Boundary 142
Interior Sound Hard Boundary (Wall) 144
Interior Normal Acceleration 144
Interior Normal Velocity 145
Interior Normal Displacement. 145
Interior Impedance/Pair Impedance 146
Interior Perforated Plate/Pair Perforated Plate. 147
Interior Lumped Speaker Boundary 149
Continuity 151
Line Source 151
Line Source on Axis. 153
Monopole Point Source 154

Dipole Point Source. 156
Quadrupole Point Source 158
Point Sources (for 2D Components) 160
Circular Source (for 2D Axisymmetric Components) 162
Pressure (Point Condition) 163
AWE Expression 164

The Pressure Acoustics, Transient Interface 165
Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface 168
Transient Pressure Acoustics Model 168
Nonlinear Acoustics (Westervelt) Contributions. 170
Background Pressure Field (for Transient Models) 173
Impedance 174
Incident Pressure Field (for Transient Models). 175
The Gaussian Pulse Source Type Settings. 176
Normal Acceleration 177
Normal Velocity 177
Normal Displacement 178
Exterior Field Calculation (for Transient Models) 178
Lumped Speaker Boundary 179
Interior Normal Acceleration 181
Interior Normal Velocity 182
Interior Normal Displacement. 182
Interior Impedance/Pair Impedance 183
Interior Lumped Speaker Boundary 184

The Pressure Acoustics, Boundary Mode Interface 186
Initial Values 188
Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Boundary Mode Interface 188
Impedance 188

The Pressure Acoustics, Boundary Elements Interface 189
Domain, Boundary, Edge, and Pair Nodes for the Pressure Acoustics, Boundary Elements Interface 197
Pressure Acoustics 198
Background Pressure Field 199

Initial Values . . . 200
Sound Hard Boundary (Wall) . . . 200
Normal Acceleration . . . 200
Normal Velocity . . . 201
Normal Displacement . . . 201
Sound Soft Boundary . . . 201
Pressure . . . 202
Impedance . . . 202
Exclude Boundary . . . 202
Interior Sound Hard Boundary (Wall) . . . 202
Interior Normal Acceleration . . . 203
Interior Normal Velocity . . . 204
Interior Normal Displacement . . . 204

The Pressure Acoustics, Time Explicit Interface 206
Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Time Explicit Interface . . . 210
Pressure Acoustics, Time Explicit Model . . . 211
Background Acoustic Field . . . 214
Poroacoustics . . . 216
Initial Values . . . 218
Mass Source . . . 218
Heat Source . . . 219
Volume Force Source . . . 219
Sound Hard Boundary (Wall) . . . 220
Continuity of Total Fields . . . 220
Sound Soft Boundary . . . 220
Pressure . . . 220
Symmetry . . . 221
Normal Velocity . . . 221
Impedance . . . 221
Exterior Field Calculation . . . 224
Interior Sound Hard Boundary (Wall) . . . 225
Interior Normal Velocity . . . 225
Interior Impedance . . . 226
Material Discontinuity . . . 226
Continuity . . . 227
General Flux/Source . . . 227

General Interior Flux 228

The Pressure Acoustics, Asymptotic Scattering Interface 229
Domain and Boundary Nodes for the Pressure Acoustics, Asymptotic Scattering Interface. 231
Pressure Acoustics 232
Background Pressure Field 232
Scattering Object. 232
Wall 232
Exterior Field Calculation 233

The Pressure Acoustics, Kirchhoff–Helmholtz Interface 234
Domain and Boundary Nodes for the Pressure Acoustics, Asymptotic Scattering Interface. 235
Pressure Acoustics 236
Radiating Object 236
Wall 236
Exterior Field Calculation 237

Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces) 238
Physics-Controlled Mesh for Pressure Acoustics 238
Meshing: Resolving the Waves in Space 240
Lagrange and Serendipity Shape Functions 242
Time Stepping: Resolving the Waves in Time 243
Frequency Domain, Modal and Adaptive Frequency Sweep (AWE) . . . 245
Solving Large Acoustics Problems Using Iterative Solvers. 247
Perfectly Matched Layers (PMLs) 254
Postprocessing Variables 260
Evaluating the Acoustic Field in the Exterior: Near- and Far-Field . . . 265
Dedicated Acoustics Plots for Postprocessing 267
About the Material Databases for the Acoustics Module 273
Specifying Frequencies: Logarithmic and ISO Preferred 273

Modeling with the Pressure Acoustics Branch (BEM-Based Interface) 274
When to Use BEM 274
Selections: Infinite Void and Finite Voids 275

Solvers for BEM Models 276
Meshing BEM Models 277
Postprocessing BEM Results. 277

Modeling with the Pressure Acoustics Branch (DG-FEM-Based Interface) 280
Meshing, Discretization, and Solvers 280
Using the Partial Fraction Fit for Acoustic Data 280
Results: Variables and Quality 285
Absorbing Layers. 285
Storing Solution on Selections for Large Models 285
Assemblies and Pair Conditions 285

Modeling with the Aeroacoustic Flow Source: Flow-Induced Noise 286
The Lighthill Analogy 286
Hybrid Computational Aeroacoustics (CAA) 287
Setting up a Flow-Induced Noise Problem in COMSOL 288

Theory Background for the Pressure Acoustics Branch 290
The Governing Equations. 290
Pressure Acoustics, Frequency Domain Equations 294
Pressure Acoustics, Transient Equations 297
The Nonlinear Westervelt Equation 297
Pressure Acoustics, Boundary Mode Equations 299
Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions 300
Theory for the Exterior Field Calculation: The Helmholtz–Kirchhoff Integral 303

Theory for the Boundary Impedance Models 306
Impedance Conditions. 306
RCL Models. 307
Physiological Models 309
Waveguide End Impedance Models 317
Porous Layer Models 319
Characteristic Specific Impedance Models 320

Theory for the Interior Impedance Models **322**
Interior Perforated Plate Models 322

Theory for the Equivalent Fluid Models **328**
Introduction to the Equivalent Fluid Models. 328
About the Pressure Acoustics Fluid Models. 330
About the Poroacoustics Models. 332
About the Narrow Region Acoustics Models 347

Theory for the Perfectly Matched Layers in the Time Domain **353**
Introduction to Perfectly Matched Layers 353
Perfectly Matched Layers in the Time Domain. 354

References for the Pressure Acoustics Branch **355**

Chapter 3: Elastic Waves Interfaces

The Solid Mechanics (Elastic Waves) Interface **362**

The Poroelastic Waves Interface **364**
Domain, Boundary, and Pair Nodes for the Poroelastic Waves Interfaces 367
Poroelastic Material. 368
Anisotropic Poroelastic Material 374
Body Load 377
Impervious Layer. 378
Free. 378
Initial Values 378
Impervious Mass Layer. 378
Periodic Condition 379
Free, Sound Soft 379
Pressure 380
Symmetry 380
Symmetry Plane 381
Boundary Load 381

Fixed Constraint 381
Prescribed Displacement 381
Prescribed Velocity 382
Prescribed Acceleration 383
Roller 384
Thin Elastic Layer. 384
Added Mass. 387
Interior Impervious Layer 387
Edge Load 388
Point Load 388
Ring Load 388
Point Load (on Axis) 388

The Elastic Waves, Time Explicit Interface **389**
Domain, Boundary, Edge, Point, and Pair Nodes for the Elastic Waves, Time Explicit Interface 392
Elastic Waves, Time Explicit Model 393
Damping 398
Compute Displacement 399
Initial Values 400
Body Load 400
Axial Symmetry 400
Free. 401
Fixed 401
Prescribed Velocity 401
Roller 401
Boundary Load 401
Low-Reflecting Boundary. 402
Symmetry 403
Antisymmetry 403
Continuity 403
General Flux/Source 403
General Interior Flux 404
Fracture 404
Material Discontinuity 406

The Piezoelectric Waves, Time Explicit Interface **407**
Piezoelectric Material 408

Mechanical Damping 409

Modeling with the Elastic Waves Branch **410**
Meshing and Solving Wave Problems Solved with Solid Mechanics . . . 410
Meshing Poroelastic Waves Models 411
Solving Large Poroelastic Wave Models 411
Meshing and Solving Elastic Waves, Time Explicit Models. 412
Absorbing Layers in Elastic Waves, Time Explicit 414
Computing the Displacement in the Elastic Waves, Time Explicit. . . . 415

Theory for the Poroelastic Waves Interfaces **416**
Elastic Waves Introduction 416
Poroelastic Waves Theory 417
Anisotropic Poroelastic Material Theory 422
Boundary Conditions for Poroelastic Waves 423
Postprocessing Variables 426
References for the Poroelastic Waves Interfaces 426

Theory for the Elastic Waves, Time Explicit Interface **429**
Governing Equations 429
The Fracture Boundary Condition 429
References for the Elastic Waves, Time Explicit Interface 430

Chapter 4: Acoustic–Structure Interaction Interfaces

The Acoustic–Solid Interaction, Frequency Domain Interface **432**

The Acoustic–Solid Interaction, Transient Interface **435**

The Acoustic–Piezoelectric Interaction, Frequency Domain

Interface **438**

The Acoustic–Piezoelectric Interaction, Transient Interface **441**

The Acoustic–Poroelastic Waves Interaction Interface **444**

The Acoustic–Solid–Poroelastic Waves Interaction Interface **446**

The Acoustic–Solid Interaction, Time Explicit Interface **448**

The Acoustic–Shell Interaction, Frequency Domain Interface **451**

The Acoustic–Shell Interaction, Transient Interface **454**

Modeling with the Acoustic–Structure Interaction Branch **457**
Prestressed Acoustic–Structure Interaction 457
Solving Large Acoustic–Structure Interaction Models 458
Configuration of Perfectly Matched Layers (PMLs) for Acoustic–Structure Interaction Models 459

Chapter 5: Aeroacoustics Interfaces

The Linearized Euler, Frequency Domain Interface **463**
Domain, Boundary, and Pair Nodes for the Linearized Euler, Frequency Domain Interface 467
Linearized Euler Model 467
Rigid Wall 470
Initial Values 471
Axial Symmetry 471
Domain Sources 472
Background Acoustic Fields 472
Pressure (Isentropic) 473
Prescribed Acoustic Fields 474
Sound Soft Boundary (Isentropic) 475
Impedance 475
Symmetry 476

Periodic Condition 476
Moving Wall 479
Specific Acoustic Impedance (Isentropic) 480
Interior Wall 480
Interior Impedance 481
Asymptotic Far-Field Radiation 482
Outflow Boundary 483
Continuity 483

The Linearized Euler, Transient Interface 484
Domain, Boundary, and Pair Nodes for the Linearized Euler, Transient Interface 486
Initial Values 487
Moving Wall 487

The Linearized Euler, Boundary Mode Interface 489
Linearized Euler Model 490
Rigid Wall 490
Initial Values 490

The Linearized Navier–Stokes, Frequency Domain Interface 492
Domain, Boundary, and Pair Nodes for the Linearized Navier–Stokes, Frequency Domain and Transient Interfaces 497
Linearized Navier–Stokes Model 498
Domain Sources 503
First-Order Material Parameters 503
Background Acoustic Fields 504
Initial Values 504
Axial Symmetry 505
Wall 505
Pressure (Adiabatic). 506
Symmetry 507
Periodic Condition 507
Interior Wall 510
Interior Impedance 511
No Slip 512
Slip (perfect) 512
Prescribed Velocity 513

Prescribed Pressure 514
No Stress 514
Boundary Stress 514
Impedance 515
Isothermal 516
Adiabatic 517
Prescribed Temperature 517
Heat Flux 517
Continuity 518

The Linearized Navier–Stokes, Transient Interface 519
Impedance 521

The Linearized Navier–Stokes, Boundary Mode Interface 522
Linearized Navier–Stokes Model 523
Wall 523
Initial Values 523
Impedance 524

The Linearized Potential Flow, Frequency Domain Interface 525
Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Frequency Domain Interface 528
Linearized Potential Flow Model 529
Initial Values 531
Sound Hard Boundary (Wall) 531
Velocity Potential 532
Normal Mass Flow 532
Plane Wave Radiation 533
Incident Velocity Potential 534
Sound Soft Boundary 534
Periodic Condition 534
Normal Velocity 535
Impedance 536
Symmetry 536
Port 536
Port Reference Axis 542
Interior Impedance and Pair Impedance 542
Vortex Sheet 543

Interior Sound Hard Boundary (Wall) 543
Continuity 544
Mass Flow Edge Source 544
Mass Flow Point Source 545
Mass Flow Circular Source 545
Mass Flow Line Source on Axis 545
Axial Symmetry 546

The Linearized Potential Flow, Transient Interface 547
Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Transient Interface 548
Symmetry 549

The Linearized Potential Flow, Boundary Mode Interface 550
Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Boundary Mode Interface 552
Symmetry 552
Impedance 553

The Compressible Potential Flow Interface 554
Domain, Boundary, and Pair Nodes for the Compressible Potential Flow Interface 555
Compressible Potential Flow Model. 556
Initial Values 557
Slip Velocity. 557
Symmetry 558
Normal Flow 558
Mass Flow 558
Mean Flow Velocity Potential 558
Periodic Condition 559
Interior Wall (Slip Velocity) 560

Modeling with the Aeroacoustics Branch 561
Selecting an Aeroacoustics Interface 561
Meshing 562
Stabilization 563
Solver Suggestions for Large Aeroacoustic Models 564
Absorbing Layers for the Linearized Euler, Transient Interface 565

Lagrange and Serendipity Shape Functions 566
Time Stepping in Transient Models 567
Mapping Between Fluid Flow and Acoustics Mesh 567
Coupling to Turbulent Flows (Eddy Viscosity) 570
Eigenfrequency Studies 570
Suppressing Constraints on Lower Dimensions 571
Postprocessing Variables 572

Theory Background for the Aeroacoustics Branch 575
General Governing Equations 576
Linearized Navier–Stokes 579
Linearized Euler 580
Scattered Field Formulation for LE and LNS 583
Linearized Potential Flow. 584
Compressible Potential Flow 587

References for the Aeroacoustics Branch Interfaces 590

Chapter 6: Thermoviscous Acoustics Interfaces

The Thermoviscous Acoustics, Frequency Domain Interface 594
Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface 601
Thermoviscous Acoustics Model 602
Background Acoustic Fields 607
Heat Source 608
Initial Values 608
Axial Symmetry 608
Wall 608
Pressure (Adiabatic). 609
Symmetry 610
Slip Wall 611
Port. 613
Lumped Port 620
Lumped Speaker Boundary 624
Periodic Condition 627

Interior Wall 629
Interior Impedance 630
Interior Velocity 631
Interior Temperature Variation 632
Interior Lumped Speaker Boundary 633
Interior Slip Wall 634
Surface Tension 634
No Slip 635
Slip (Perfect) 636
Velocity 637
No Stress 637
Boundary Stress 638
Impedance 638
Isothermal 640
Adiabatic 640
Temperature Variation 640
Heat Flux 641

The Thermoviscous Acoustics, Transient Interface 642
Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Transient Interface 645
Thermoviscous Acoustics Model 647
Nonlinear Thermoviscous Acoustics Contributions 648
Background Acoustic Fields 650
Impedance 650
Lumped Speaker Boundary 651
Interior Lumped Speaker Boundary 652

The Thermoviscous Acoustics, Boundary Mode Interface 654
Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Boundary Mode Interface 657
Thermoviscous Acoustics Model 658
Impedance 659

The Thermoviscous Acoustics, SLNS Approximation Interface 660
Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, SLNS Approximation Interface 662
Thermoviscous Acoustics Model 663

Background Pressure Field 665
Initial Values 665
Monopole Domain Source 666
Dipole Domain Source 666
Axial Symmetry 666
Wall 666
Normal Acceleration 667
Normal Velocity 667
Normal Displacement 667
Pressure 668
Impedance 668
Symmetry 668
Periodic Condition 668
Lumped Port 669
Lumped Speaker Boundary 669

The Acoustic–Thermoviscous Acoustic Interaction, Frequency Domain Interface 670

The Thermoviscous Acoustic–Solid Interaction, Frequency Domain Interface 673

The Thermoviscous Acoustic–Shell Interaction, Frequency Domain Interface 675

The Thermoviscous Acoustic–Thermoelasticity Interaction Multiphysics Interfaces 678

Modeling with the Thermoviscous Acoustics Branch 680
Physics-Controlled Mesh 680
Meshing the Boundary Layer 681
Solver Suggestions for Large Thermoviscous Acoustics Models 682
Lagrange and Serendipity Shape Functions 685
Transient Solver Settings 686
Postprocessing Variables 686
Suppressing Constraints on Lower Dimensions 690

Theory Background for the Thermoviscous Acoustics Branch 692
The Viscous and Thermal Boundary Layers 693
General Linearized Compressible Flow Equations 694
Acoustic Perturbation and Linearization 695
Scattered Field Formulation and Background Acoustic Fields 700
Formulation for Eigenfrequency Studies 701
Formulation for Mode Analysis in 2D and 1D Axisymmetry. 703
Formulation for the Boundary Mode Interface. 704
The Nonlinear Thermoviscous Acoustics Equations. 705
Theory for the SLNS (Sequential Linearized Navier–Stokes) Formulation 706
Theory for the Slip Wall Boundary Condition 707
References for the Thermoviscous Acoustics, Frequency Domain Interface. 708

Chapter 7: Ultrasound Interfaces

The Convected Wave Equation, Time Explicit Interface 712
Domain, Boundary, Edge, Point, and Pair Nodes for the Convected Wave Equation Interface 715
Convected Wave Equation Model 716
Domain Sources 719
Sound Hard Wall. 720
Initial Values 720
Pressure 720
Symmetry 720
Normal Velocity 721
Specific Acoustic Impedance (Isentropic) 721
Interior Wall 722
Interior Normal Velocity 722
General Flux/Source 722
General Interior Flux 723

The Nonlinear Pressure Acoustics, Time Explicit Interface 725
Domain, Boundary, Edge, and Point Nodes for the Nonlinear Pressure Acoustics, Time Explicit Interface. 729

Nonlinear Pressure Acoustics, Time Explicit Model 729
Compute Minimum and Maximum Pressure. 731
Initial Values 731
Mass Source 732
Heat Source 732
Volume Force Source 733
Sound Hard Boundary (Wall) 733
Sound Soft Boundary 733
Pressure 733
Symmetry 734
Normal Velocity 734
Impedance 734
Interior Sound Hard Boundary (Wall) 735
Interior Normal Velocity 735
Material Discontinuity 736
Continuity 736
General Flux/Source 737
General Interior Flux 737

Modeling with the Convected Wave Equation Interface 738
Meshing, Discretization, and Solvers 738
Results: Variables and Quality 740
Absorbing Layers 741
Stabilizing Physical Instabilities (Filtering) 742
Storing Solution on Selections for Large Models 742
Assemblies and Pair Conditions 743

Modeling with the Nonlinear Pressure Acoustics, Time Explicit Interface 744
Solving Highly Nonlinear Problems 744
Meshing 745
Adaptive Mesh Refinement 745

Theory for the Convected Wave Equation Interface 747
Governing Equations of the Convected Wave Equation 747
Boundary Conditions 749
The Lax–Friedrichs Flux 750

Theory for the Nonlinear Pressure Acoustics, Time Explicit Interface **751**
Governing Equations for Nonlinear Pressure Acoustics, Time Explicit . . 751

References for the Ultrasound Interface **752**

Chapter 8: Geometrical Acoustics Interfaces

The Ray Acoustics Interface **754**
Domain, Boundary, and Global Nodes for the Ray Acoustics Interface . . . 762
Medium Properties . . . 763
Release . . . 766
Accumulator (Domain) . . . 774
Background Velocity . . . 775
Material Discontinuity . . . 776
Wall . . . 777
Accumulator (Boundary) . . . 784
Sound Pressure Level Calculation . . . 785
Axial Symmetry . . . 786
Receiver . . . 787
Release from Boundary . . . 788
Nonlocal Accumulator . . . 792
Release from Pressure Field . . . 793
Ray Detector . . . 795
Release from Symmetry Axis . . . 795
Ray Continuity . . . 796
Release from Edge . . . 797
Release from Point . . . 797
Release from Point on Axis . . . 798
Ray Properties . . . 798
Auxiliary Dependent Variable . . . 798
Release from Grid . . . 799
Release from Grid on Axis . . . 803
Release from Data File . . . 803
Release from Exterior Field Calculation . . . 805

Source with Directivity 808
Ray Termination 811

Modeling with the Ray Acoustics Interface **814**
Mixed Diffuse and Specular Wall Conditions 814
Assigning Directivity to a Source 815
Impulse Response Plot and Receiver Dataset 815
Stopping Rays for a Given Condition 824
Mesh Guidelines 825
Nonlocal Couplings 827
Using Ray Detectors 829
Other Results Plots, Datasets, and Derived Values 830

Theory for the Ray Acoustics Interface **832**
Introduction to Ray Acoustics 832
Initial Conditions: Position 833
Initial Conditions: Direction. 834
Material Discontinuity Theory 836
Intensity and Wavefront Curvature 837
Intensity and Phase Reinitialization 842
Wavefront Curvature Calculation in Graded Media. 845
Attenuation Within Domains 850
Ray Termination Theory 851
Accumulator Theory: Domains 853
Accumulator Theory: Boundaries 855
Sound Pressure Level Calculation Theory 856
Assigning Spatial Directivity to a Source 857
Impulse Response Reconstruction 858
References for the Ray Acoustics Interface 860

The Acoustic Diffusion Equation Interface **861**
Domain, Boundary, and Global Nodes for the Acoustic Diffusion Equation Interface 865
Acoustic Diffusion Model. 865
Room 866
Wall 867
Inward Energy Flux 868
Initial Values 868

Domain Source 869
Fitted Domain 869
Room Coupling 869
Mapped Room Coupling 870
Manual Destination Selection 871
Point Source 871

Modeling with the Acoustic Diffusion Equation Interface 872
The Eigenvalue Study Type 872
Combined Stationary and Time Dependent Study 872

Theory for the Acoustic Diffusion Interface 873
Statistical Model of Reverberation Time 873
The Acoustic Diffusion Equation 874
References for the Acoustic Diffusion Equation Interface. 880

Chapter 9: Pipe Acoustics Interfaces

The Pipe Acoustics Frequency Domain and Transient Interfaces 884
The Pipe Acoustics, Frequency Domain Interface. 884
The Pipe Acoustics, Transient Interface 886
Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces 887
Initial Values 888
Fluid Properties 888
Pipe Properties 889
Volume Force 890
Closed. 890
Pressure 891
Velocity 891
End Impedance 892

Theory for the Pipe Acoustics Interfaces 894
Governing Equations 894
Theory for the Pipe Acoustics Boundary Conditions 898

Solving Transient Problems 901
Cutoff Frequency 902
Flow Profile Correction Factor 902
References for the Pipe Acoustics Interfaces 903

Chapter 10: Acoustic Streaming

Acoustic Streaming Multiphysics **906**
Acoustic Streaming Multiphysics Interfaces 906

Modeling Acoustic Streaming **908**
Sources of Acoustic Streaming 908
Acoustic Streaming from Thermoviscous Acoustics 909
Acoustic Streaming from Pressure Acoustics 909
Lagrangian Velocity Field 910
Study Types 911

Theory Background for the Acoustic Streaming Multiphysics **912**
Governing Equations 912
Perturbation Theory 913
Effective Theory 914
References 916

Chapter 11: Multiphysics Couplings

Coupling Features **918**
Acoustic–Structure Boundary 919
Thermoviscous Boundary Layer Impedance (subfeature) 921
Pair Acoustic–Structure Boundary 922
Thermoviscous Acoustic–Structure Boundary 922
Pair Thermoviscous Acoustic–Structure Boundary 924
Thermoviscous Acoustic–Thermoelasticity Boundary 925
Thermoviscous Acoustic–Thermal Perturbation Boundary 927
Thermoviscous Acoustic SLNS-Structure Boundary 927
Aeroacoustic–Structure Boundary 928

Acoustic–Thermoviscous Acoustic Boundary 930
Pair Acoustic–Thermoviscous Acoustic Boundary 931
Acoustic–Porous Boundary 932
Porous–Structure Boundary 933
Background Potential Flow Coupling 933
Background Fluid Flow Coupling 934
Aeroacoustic Flow Source Coupling 936
Acoustic FEM–BEM Boundary 938
Impedance (subfeature) 939
Acoustic–Pipe Acoustic Connection 939
Acoustic–Structure Boundary, Time Explicit 940
Pair Acoustic–Structure Boundary, Time Explicit 941
Convected Acoustic–Structure Boundary, Time Explicit 942
Pair Convected Acoustic–Structure Boundary, Time Explicit 943
Piezoelectricity, Time Explicit 944
Piezoelectricity 944
Magnetomechanics 944
Acoustic Streaming Domain Coupling 945
Acoustic Streaming Boundary Coupling 946

Predefined Multiphysics Interfaces **949**

Modeling with Multiphysics Couplings **951**
Use Selections 951
The Override Behavior 952
The Solvers 952
Perfectly Matched Layers (PMLs) 953

Chapter 12: Structural Mechanics with the Acoustics Module

Vibroacoustic Applications 956
The Solid Mechanics Interface 956
The Piezoelectricity Interface 956
Acoustic–Structure Multiphysics Interaction 957

Chapter 13: Study Types

Acoustics Module Study Types **960**
Stationary Study 961
Frequency Domain Study 961
Eigenfrequency Study 962
Mode Analysis Study 964
Boundary Mode Analysis 966
Time Dependent Study 966
Frequency Domain, Modal and Time-Dependent, Modal Studies 967
Adaptive Frequency Sweep 968
Ray Tracing 968
Modal Reduced-Order Model 968
Additional Analysis Capabilities 969
Mapping 969
Transient Mapping 970
Frequency-Stationary 970
Frequency-Transient 971

Chapter 14: Acoustic Properties of Fluids

Material Properties **974**

The Bulk Viscosity **978**
The Value of the Bulk Viscosity 978

Attenuation and Loss Models **981**
Loss Mechanisms 981
The Atmosphere and Ocean Attenuation Models 982
Boundary Layer vs. Bulk Losses 983

References for The Acoustic Properties of Fluids **985**

Chapter 15: Glossary

Glossary of Terms **988**

1

Introduction

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

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

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

In this chapter:

- Acoustics Module Capabilities
- Fundamental of Acoustics
- Acoustics Module Physics Interface Guide
- Overview of the User's Guide

Acoustics Module Capabilities

In this section:

- What Can the Acoustics Module Do?
- What Are the Application Areas?
- Which Problems Can You Solve?

What Can the Acoustics Module Do?

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

The Acoustics Module also includes many specialized formulations and material models that can be used for dedicated application areas like thermoviscous acoustics used in miniature transducers and mobile devices or Biot's equations for modeling poroelastic waves. It also includes many predefined couplings between physics, called Multiphysics couplings, to model, for example, vibroacoustic problems.

The module supports time-harmonic (frequency domain), eigenfrequency, modal, and transient studies for all fluids (depending on the acoustic equations solved) as well as static, transient, eigenfrequency, modal, and frequency-response for the analyses of wave propagation in structures.

The multiphysics environment is further extended as the module combines several dedicated numerical methods, including the finite element method (FEM), the boundary element method (BEM), ray tracing, and the discontinuous Galerkin finite elements method (dG-FEM).

The available physics interfaces include the following functionality:

- *Pressure acoustics*: model the propagation of sound waves (pressure waves) in the frequency domain solving the Helmholtz equation or in the time domain solving the scalar wave equation. Pressure acoustics comes in different flavors depending on the numerical formulation used. This includes finite element (FEM) based interfaces for frequency and transient models, a boundary element (BEM) based interface only used in the frequency domain, and a discontinuous Galerkin (dG-FEM) formulation based interface used for transient simulations. The Acoustics Module has built-in

couplings between BEM and FEM that allows for modeling hybrid FEM-BEM problems.

- *Acoustic-structure interaction*: combine pressure waves in the fluid with elastic waves in the solid to model *vibroacoustic* problems. The physics interfaces provide predefined multiphysics couplings at the fluid-solid interface.
- *Boundary mode acoustics*: find propagating and evanescent modes in ducts and waveguides. The analysis exists for classical pressure acoustic, elastic waves, thermoviscous acoustics, and convected acoustic problems.
- *Thermoviscous acoustics*: model the detailed propagation of sound in geometries with small length scales. This is acoustics including thermal and viscous losses explicitly. Also known as *microacoustics*, *viscothermal acoustics*, or linearized compressible Navier–Stokes. In the time domain nonlinear effects can be included.
- *Aeroacoustics*: model the influence a background mean flow has on the propagation of sound waves in the flow, so-called, *convected acoustics* or flow borne noise/sound. Interfaces exist to solve the linearized potential flow, the linearized Euler equations, and the linearized Navier–Stokes equations in both time and frequency domain.
- *Flow-induced noise*: model the noise generated by a turbulent flow using the Lighthill analogy to set up an aeroacoustic flow source in pressure acoustics.
- *Compressible potential flow*: determine the flow of a compressible, irrotational, and inviscid fluid.
- *Solid mechanics and elastic waves*: solve structural mechanics problems and the propagation of elastic waves in solids.
- *Piezoelectricity*: model the behavior of piezoelectric materials in a multiphysics environment solving for the electric field and the coupling to the solid structure.
- *Poroelastic waves*: in porous materials model the coupled propagation of elastic waves in the solid porous matrix and the pressure waves in the saturation fluid. Biot's equations are solved here. Includes options to include both thermal and viscous losses.
- *Ultrasound*: in ultrasound problems transient propagation is important and it is also important to be able to solve models with many wavelengths. These interfaces are based in the discontinuous Galerkin or dG-FEM formulation.
- *Nonlinear acoustics*: model nonlinear acoustic phenomena like the cumulative nonlinear effects by solving the Westervelt equation. Local nonlinear effects such as vortex shedding can be included using the nonlinear thermoviscous acoustic functionality.

- *Acoustic diffusion equation*: solve a diffusion equation for the acoustic energy density distribution for systems of coupled rooms in room acoustic applications.
- *Ray acoustics*: compute trajectories and intensity of acoustic rays in room acoustic as well as underwater acoustic applications. Determine the impulse response and room acoustic metrics with dedicated features in postprocessing.
- *Pipe acoustics*: use this physics interface to model the propagation of sound waves in pipe systems including the elastic properties of the pipe. The equations are formulated in 1D for fast computation and can include a stationary background flow.
- *Acoustic streaming*: use these multiphysics capabilities to couple pressure acoustics or thermoviscous acoustics to a fluid flow interface to model acoustic streaming, that is flow induced by sound.

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

pressure acoustics models, you can compute the exterior acoustic field (phase and magnitude) and plot it in predefined radiation pattern plots.

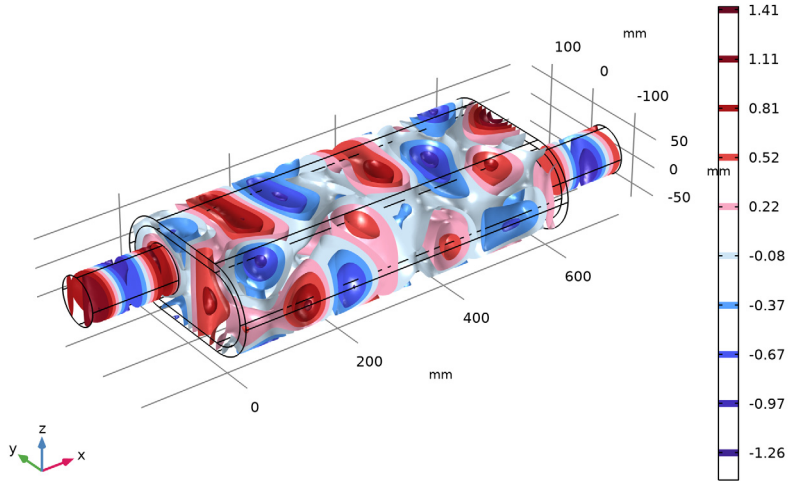


Figure 1-1: An application example is the modeling of mufflers. Here a plot from the Absorptive Muffler model from the COMSOL Multiphysics Application Library.

What Are the Application Areas?

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

Typical application areas for the Acoustics Module include:

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

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

- Mobile applications such as feedback analysis, optimized transducer placement, and directivity assessment.
- Convected acoustics for jet engine noise, muffler systems with nonisothermal flow, and flowmeters.
- Ultrasound piezoelectric transducers.
- Musical instruments.
- Bioacoustic applications with ultrasound and more.
- Underwater acoustics and sonar applications.
- Pressure and elastic waves in geophysics.
- Room acoustics using the ray tracing method or an acoustic diffusion equation approach.
- Advanced multiphysics applications such as photoacoustics, optoacoustics, thermoacoustic cooling, acoustofluidics, acoustic streaming and radiation, and combustion instabilities.

Using the full multiphysics couplings within the COMSOL Multiphysics environment, you can couple the acoustic waves to, for example, an electromagnetic analysis or a

structural analysis for acoustic-structure interaction. The module smoothly integrates with all of the COMSOL Multiphysics functionality.

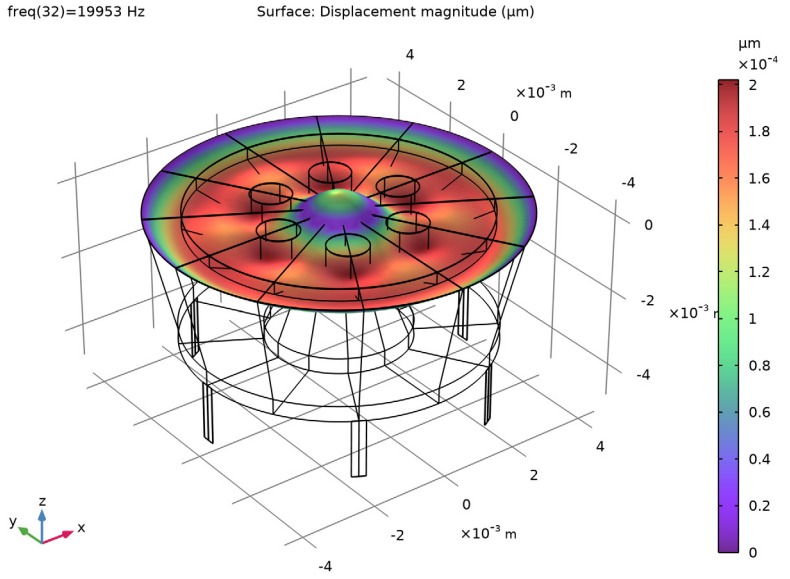


Figure 1-2: Modeling a transducer is a true multiphysics application, comprising thermoviscous acoustics, electrostatics, and a membrane. Here the displacement of the microphone diaphragm from the Brüel & Kjær 4134 Condenser Microphone model from the Acoustics Module Application Library.

Which Problems Can You Solve?

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

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

- The Acoustic-Shell Interaction interfaces are supported in 2D axisymmetric and 3D, and also require the addition of the Structural Mechanics Module.
- The Pipe Acoustics interfaces exist on edges in 2D and 3D.
- In 2D, the module has in-plane physics interfaces for problems with a planar symmetry as well as axisymmetric physics interfaces for problems with a cylindrical symmetry.
- Use the fluid acoustics interfaces with 1D and 1D axisymmetric geometries.

When using the axisymmetric models, the horizontal axis represents the r direction and the vertical axis the z direction. The geometry is in the right half plane.

Fundamental of Acoustics

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

In this section:

- Acoustics Explained
- Mathematical Models for Acoustic Analysis
- Damping
- Artificial Boundaries

Acoustics Explained

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

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

The propagation of sound in solids happens through small-amplitude elastic oscillations of its shape. These elastic waves are transmitted to surrounding fluids as ordinary sound waves. The elastic sound waves in the solid are the counterpart to the pressure waves or compressible waves propagating in the fluid.

Standard acoustic problems involve solving for the small acoustic pressure variations (often denoted p' or p_1) on top of the stationary/quiescent background pressure (often denoted p_0 or p_A). Mathematically this represents a linearization (small parameter expansion) around the stationary quiescent values.

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

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p$$
$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

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

$$\begin{aligned} p &= p_0 + p_1 \\ \rho &= \rho_0 + \rho_1 \qquad \text{with} \qquad \rho_1 \ll \rho_0 \\ \mathbf{u} &= \mathbf{0} + \mathbf{u}_1 \qquad\qquad\quad |\mathbf{u}_1| \ll c \end{aligned}$$

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

$$\frac{\partial \mathbf{u}_1}{\partial t} = -\frac{1}{\rho_0}\nabla p_1$$
$$\frac{\partial \rho_1}{\partial t} + \rho_0(\nabla \cdot \mathbf{u}_1) = 0$$

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

$$\rho_1 = \left.\frac{\partial \rho_0}{\partial p}\right|_s p_1 = \frac{1}{c_s^2} p_1$$

where c_s is recognized as the (isentropic) speed of sound (SI unit: m/s) at constant entropy s. It should be noted that this equation is valid for constant valued (not space dependent) background density ρ_0 and background pressure p_0. The subscripts s is dropped in the following. From the above expression it also follows that another requirement for linear acoustics (the perturbation approximation) to be valid is that

$$p_1 \ll \rho_0 c^2$$

Finally, rearranging the equations (divergence of momentum equation inserted into the continuity equation) and dropping the subscript 1 yields the wave equation for sound waves in a lossless medium

$$\frac{1}{c^2}\frac{\partial^2 p}{\partial t^2} - \nabla \cdot (\nabla p) = 0 \tag{1-1}$$

The speed of sound is related to the compressibility of the fluid where the waves are propagating. The combination ρc^2 is called the *bulk modulus*, commonly denoted K (SI unit: N /m^2).

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

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

where $\omega = 2\pi f$ (SI unit: rad/s) is the angular frequency and f (SI unit: Hz) is denoting the frequency. The wave equation for acoustic waves reduces to the Helmholtz equation:

$$\nabla \cdot (\nabla p) + \frac{\omega^2}{c^2} p = 0 \tag{1-2}$$

where the ratio ω/c is recognized as the wave number k. This equation can also be treated as an eigenvalue PDE to solve for eigenmodes and eigenfrequencies.

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

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

A detailed derivation of the governing equations is given in Theory Background for the Pressure Acoustics Branch. For the propagation of compressional (acoustic) waves

in a viscous and thermally conducting fluid the theory is presented in Theory Background for the Thermoviscous Acoustics Branch and for acoustics in moving media (aeroacoustics) in Theory Background for the Aeroacoustics Branch.

Damping

Fluids like air or water — by far the most common media in acoustics simulations — exhibit practically no internal damping (so-called bulk attenuation) over the number of wavelengths that can be resolved with the finite element method and in the audio frequency range. However, in ultrasound applications or when using ray tracing to model room and underwater acoustics, these become important.

In smaller systems, damping takes place through interaction with solids, either because of friction between the fluid and a porous material filling the domain, or because acoustic energy is transferred to a surrounding solid where it is absorbed. In systems with small length scales, significant losses can occur in the viscous and thermal acoustic boundary layer at walls.

ATMOSPHERE AND OCEAN ATTENUATION

When performing ray acoustics simulations or modeling ultrasound applications, the bulk or internal attenuation of atmospheric air or the ocean sea water can be modeled using the built-in *Atmosphere attenuation* or the *Ocean attenuation* models. Both models are semi analytical and fitted to extensive experimental data. The atmosphere model complies with the ANSI standard.

For further details about material properties and attenuation models see the Acoustic Properties of Fluids chapter.

POROUS ABSORBING MATERIALS

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

It is possible to directly measure the complex wave number and impedance in an impedance tube in order to produce curves of the real and imaginary parts (the

resistance and reactance, respectively) as functions of frequency. These data can be used directly as input to COMSOL Multiphysics interpolation functions to define k and Z.

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

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

BOUNDARY LAYER ABSORPTION (THERMOVISCOUS ACOUSTICS)

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

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

the Thermoviscous Boundary Layer Impedance boundary condition, also available in Pressure Acoustics.

More details on the detailed acoustic model for viscous and thermal losses are described in Thermoviscous Acoustics Interfaces. See the boundary layer absorption fluid models in Narrow Region Acoustics for simplified modeling in uniform waveguide structures or the Thermoviscous Boundary Layer Impedance boundary condition.

DAMPING AT BOUNDARIES

The losses associated with the acoustic field often stem from the interaction with boundaries, for example, when interacting with a rubber material. In this case, it may be necessary to include the acoustic-structure interaction using the appropriate multiphysics coupling. Another way of including the losses is to use an impedance boundary condition. The Acoustics Module provides a series of impedance models to model, for example, the human ear, human skin, or a simple mechanical lumped RCL system.

Artificial Boundaries

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

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

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

Another way to model an open nonreflecting boundary is to add a so-called perfectly matched layer (PML) domain or an absorbing layer domain. These domains use two different techniques to dampens all outgoing waves with no or minimal reflections. See, for example, Perfectly Matched Layers (PMLs) for more information. In pressure acoustics a special version of the PML exists as a boundary condition, the Perfectly Matched Boundary.

Acoustics Module Physics Interface Guide

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

In the *COMSOL Multiphysics Reference Manual:*

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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Acoustics				
Pressure Acoustics				
Pressure Acoustics, Frequency Domain[1]		acpr	all dimensions	eigenfrequency; frequency domain; frequency domain, modal; adaptive frequency sweep; mode analysis (2D and 1D axisymmetric models only); boundary mode analysis (3D and 2D axisymmetric models only)
Pressure Acoustics, Transient		actd	all dimensions	eigenfrequency; frequency domain; frequency domain, modal; time dependent; time dependent, modal; mode analysis (2D and 1D axisymmetric models only)

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Pressure Acoustics, Boundary Mode		acbm	3D, 2D axisymmetric	mode analysis
Pressure Acoustics, Boundary Elements		pabe	3D, 2D	frequency domain
Pressure Acoustics, Time Explicit		pate	3D, 2D, 2D axisymmetric	time dependent
Pressure Acoustics, Asymptotic Scattering		paas	3D	frequency domain
Pressure Acoustics, Kirchhoff–Helmholtz		pakh	3D	frequency domain
Elastic Waves				
Solid Mechanics (Elastic Waves)[1]		solid	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; mode analysis; time dependent; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; frequency domain, modal reduced-order model; frequency domain, AWE reduced-order model; modal reduced-order model
Poroelastic Waves		pelw	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal
Elastic Waves, Time Explicit		elte	3D, 2D, 2D axisymmetric	time dependent
Piezoelectric Waves, Time Explicit		—	3D, 2D, 2D axisymmetric	time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Acoustic-Structure Interaction				
Acoustic–Solid Interaction, Frequency Domain[3]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal
Acoustic–Solid Interaction, Transient[3]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal; time dependent; time dependent, modal
Acoustic–Shell Interaction, Frequency Domain[2,3]		—	3D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal
Acoustic–Shell Interaction, Transient[2,3]		—	3D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal; time dependent; time dependent, modal
Acoustic–Piezoelectric Interaction, Frequency Domain[3]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal
Acoustic–Piezoelectric Interaction, Transient[3]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal; time dependent; time dependent, modal
Acoustic–Solid–Poroelastic Waves Interaction[3]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal
Acoustic–Poroelastic Waves Interaction[3]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal
Acoustic–Solid Interaction, Time Explicit		—	3D, 2D	time domain
Aeroacoustics				
Linearized Euler, Frequency Domain		lef	3D, 2D, 2D axisymmetric, 1D	frequency domain; eigenfrequency

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Linearized Euler, Transient		let	3D, 2D, 2D axisymmetric, 1D	time dependent
Linearized Euler, Boundary Mode		lebm	3D, 2D axisymmetric	mode analysis
Linearized Potential Flow, Frequency Domain		ae	all dimensions	frequency domain; mode analysis (2D and 1D axisymmetric models only)
Linearized Potential Flow, Transient		aetd	all dimensions	frequency domain; time dependent; mode analysis (2D and 1D axisymmetric models only)
Linearized Potential Flow, Boundary Mode		aebm	3D, 2D axisymmetric	mode analysis
Compressible Potential Flow		cpf	all dimensions	stationary; time dependent
Linearized Navier–Stokes, Frequency Domain		lnsf	3D, 2D, 2D axisymmetric, 1D	frequency domain; eigenfrequency
Linearized Navier–Stokes, Transient		lnst	3D, 2D, 2D axisymmetric, and 1D	time dependent
Linearized Navier–Stokes, Boundary Mode		lnsbm	3D, 2D axisymmetric	mode analysis
Thermoviscous Acoustics				
Thermoviscous Acoustics, Frequency Domain		ta	all dimensions	eigenfrequency; frequency domain; frequency domain, modal; mode analysis (2D and 1D axisymmetric models only)
Thermoviscous Acoustics, Transient		tatd	all dimensions	time dependent
Thermoviscous Acoustics, Boundary Mode		tabm	3D, 2D axisymmetric	mode analysis

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Thermoviscous Acoustics, SLNS Approximation		slns	3D, 2D, 2D axisymmetric	frequency domain
Acoustic–Thermoviscous Acoustic Interaction, Frequency Domain[3]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal; boundary mode analysis (3D and 2D axisymmetric only); mode analysis (2D only)
Thermoviscous Acoustic–Solid Interaction, Frequency Domain[3]		—	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal; mode analysis (2D only)
Thermoviscous Acoustic–Shell Interaction, Frequency Domain[2,3]		—	3D, 2D axisymmetric	eigenfrequency; frequency domain; frequency domain, modal
Thermoviscous Acoustic–Thermoelasticity Interaction, Frequency Domain[5]		—	3D, 2D, 2D axisymmetric	frequency domain perturbation
Thermoviscous Acoustic–Thermoelasticity Interaction, Transient[5]		—	3D, 2D, 2D axisymmetric	time dependent
Ultrasound				
Convected Wave Equation, Time Explicit		cwe	3D, 2D, 2D axisymmetric	time dependent
Nonlinear Pressure Acoustics, Time Explicit		nate	3D, 2D, 2D axisymmetric	time dependent
Geometrical Acoustics				
Ray Acoustics		rac	3D, 2D, 2D axisymmetric	ray tracing; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Acoustic Diffusion Equation		ade	3D	eigenvalue; stationary; time dependent
Pipe Acoustics				
Pipe Acoustics, Frequency Domain		pafd	3D, 2D	eigenfrequency; frequency domain
Pipe Acoustics, Transient		patd	3D, 2D	time dependent
Acoustic Streaming				
Acoustic Streaming from Pressure Acoustics		—	3D, 2D, 2D axisymmetric	frequency-stationary, frequency-transient
Acoustic Streaming from Thermoviscous Acoustics		—	3D, 2D, 2D axisymmetric	frequency-stationary, frequency-transient
Structural Mechanics				
Solid Mechanics[1]		solid	3D, 2D, 2D axisymmetric, 1D, 1D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; mode analysis; time dependent; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; frequency domain, modal reduced-order model; frequency domain, AWE reduced-order model
Piezoelectricity				

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Piezoelectricity, Solid		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; eigenfrequency, prestressed; time dependent; time dependent, modal; frequency domain; frequency domain, modal; frequency domain, prestressed; frequency domain, prestressed, modal; small-signal analysis, frequency domain
Magnetomechanics				
Piezomagnetism[3,4]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Nonlinear Magnetostriction[3,4]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed
Magnetomechanics[3,4]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Magnetomechanics, No Currents[3,4]		—	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; time dependent; frequency domain; small-signal analysis, frequency domain; eigenfrequency, prestressed; frequency domain, prestressed

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

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

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

[4] Requires the addition of the AC/DC Module.

[5] Requires both the MEMS Module and the Acoustics Module.

Common Physics Interface and Feature Settings and Nodes

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

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

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Where Do I Access the Documentation and Application Libraries?

A number of online resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or

context-based) help, and the Application Libraries are all accessed through the COMSOL Desktop.

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

THE DOCUMENTATION AND ONLINE HELP

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

Opening Topic-Based Help

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

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

Mac Linux	To open the **Help** window: • In the **Model Builder** or **Physics Builder**, click a node or window and then press F1. • In the main toolbar, click the **Help** (?) button. • From the main menu, select **Help>Help**.

Opening the Documentation Window

Mac Linux	To open the **Documentation** window: • Press Ctrl+F1. • In the main toolbar, click the **Documentation** ( ) button. • From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

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

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

Opening the Application Libraries Window

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

CONTACTING COMSOL BY EMAIL

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

COMSOL ACCESS AND TECHNICAL SUPPORT

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

by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/product-update
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/forum
Events	www.comsol.com/events
COMSOL Application Gallery	www.comsol.com/models
COMSOL Video Gallery	www.comsol.com/videos
Learning Center	www.comsol.com/support/learning-center
Support Knowledge Base	www.comsol.com/support/knowledgebase

Overview of the User's Guide

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

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

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

- Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces)
- Modeling with the Pressure Acoustics Branch (BEM-Based Interface)
- Modeling with the Pressure Acoustics Branch (DG-FEM-Based Interface)
- Modeling with the Aeroacoustic Flow Source: Flow-Induced Noise
- Modeling with the Elastic Waves Branch
- Modeling with the Acoustic–Structure Interaction Branch
- Modeling with the Aeroacoustics Branch
- Modeling with the Thermoviscous Acoustics Branch
- Modeling with the Convected Wave Equation Interface
- Modeling with the Nonlinear Pressure Acoustics, Time Explicit Interface
- Modeling with the Ray Acoustics Interface
- Modeling with the Acoustic Diffusion Equation Interface
- Modeling Acoustic Streaming
- Modeling with Multiphysics Couplings

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

TABLE OF CONTENTS, GLOSSARY, AND INDEX

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

THE PRESSURE ACOUSTICS BRANCH

The Pressure Acoustics Interfaces chapter describes the following interfaces:

The Pressure Acoustics, Frequency Domain Interface is the core physics interface that models sound waves in the frequency domain, and The Pressure Acoustics, Transient Interface is the core physics interface that models sound waves in the time domain with the possibility to include nonlinear effects. The Pressure Acoustics, Boundary Mode Interface solves for modes that propagate through a cross section of waveguides in a geometry.

The Pressure Acoustics, Boundary Elements Interface uses the boundary element (BEM) method to solve the Helmholtz equation in the frequency domain. It is well suited for radiation and scattering problems and can be seamlessly coupled to structures and the finite element-based The Pressure Acoustics, Frequency Domain Interface.

The Pressure Acoustics, Time Explicit Interface uses the discontinuous Galerkin (dG-FEM) formulation to solve transient models using a time-explicit method. The interface can solve large problems as the method is very computationally efficient.

The Pressure Acoustics, Asymptotic Scattering Interface and The Pressure Acoustics, Kirchhoff–Helmholtz Interface are dedicated high-frequency interfaces. The method used is also referred to as high frequency BEM or HFB.

THE ELASTIC WAVES BRANCH

The Elastic Waves Interfaces chapter describes the following interfaces:

The Solid Mechanics (Elastic Waves) Interface is a shortcut to add the Solid Mechanics interface, which is used to compute the displacement field in solids with propagating elastic waves.

The Poroelastic Waves Interface is used to compute the displacement field and acoustic pressure fluctuation in porous materials with propagating poroelastic waves. Isotropic as well as anisotropic material models are supported.

The Elastic Waves, Time Explicit Interface uses the discontinuous Galerkin (dG-FEM) formulation to solve transient linear elastic wave models using a time explicit method. The interface can solve large problems as the method is very computationally efficient.

The Piezoelectric Waves, Time Explicit Interface is a multiphysics interface that couples The Elastic Waves, Time Explicit Interface and Electrostatics in order to model the propagation of linear piezoelectric waves in the time domain, using a hybrid FEM-DG time-explicit approach. The method is very computationally efficient.

THE ACOUSTIC-STRUCTURE INTERACTION BRANCH

The Acoustic–Structure Interaction Interfaces chapter describes the following interfaces:

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

The Acoustic–Shell Interaction, Frequency Domain Interface and The Acoustic–Shell Interaction, Transient Interface requires a Structural Mechanics Module license. This multiphysics interface combines the Pressure Acoustics, Frequency Domain (or the Pressure Acoustics, Transient) interface, the Shell interface, and the Acoustic-Structure Boundary multiphysics coupling. The physics interface is available for 2D axisymmetric and 3D geometries.

The Acoustic–Piezoelectric Interaction, Frequency Domain Interface is a combination of Pressure Acoustics, Frequency Domain; solid Mechanics; Electrostatics; Acoustic-structure boundary; Piezoelectric Effect. The Acoustic–Piezoelectric Interaction, Transient Interface combines Pressure Acoustics, Transient; Solid Mechanics; Electrostatics; Acoustic-structure boundary; Piezoelectric Effect.

The Acoustic–Poroelastic Waves Interaction Interface combines Pressure Acoustics, Frequency Domain and Poroelastic Waves together with the Acoustic-Porous Boundary multiphysics coupling.

The Acoustic–Solid–Poroelastic Waves Interaction Interface combines Pressure Acoustics, Frequency Domain and Solid Mechanics together with the Acoustic-Structure Boundary and Acoustic-Porous Boundary multiphysics coupling.

The Acoustic–Solid Interaction, Time Explicit Interface combines Pressure Acoustics, Time Explicit and Elastic Waves, Time Explicit together with the Acoustic-Structure Boundary, Time Explicit multiphysics coupling.

THE AEROACOUSTICS BRANCH

The Aeroacoustics Interfaces chapter describes the following interfaces for modeling convected acoustic phenomena:

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

The Linearized Navier–Stokes, Frequency Domain Interface and The Linearized Navier–Stokes, Transient Interface model the acoustic variations in pressure, velocity, and temperature in the presence of any stationary isothermal or nonisothermal background mean flow. These physics interfaces are used for aeroacoustic simulations that can be described by the linearized Navier–Stokes equations.

The Linearized Navier–Stokes, Boundary Mode Interface and The Linearized Euler, Boundary Mode Interface are used to solve for modes that propagate through a cross section of waveguides in your geometry in the presence of a stationary mean-flow.

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

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

THE THERMOVISCOUS ACOUSTICS BRANCH

The Thermoviscous Acoustics Interfaces chapter describes The Thermoviscous Acoustics, Frequency Domain Interface and The Thermoviscous Acoustics, Transient Interface, which are necessary when modeling acoustics accurately in geometries with small dimensions. Near walls, viscosity and thermal conduction become important because they create a viscous and a thermal boundary layer where losses are significant. Nonlinear effects can be modeled in the time domain. The Thermoviscous Acoustics, Boundary Mode Interface is used to identify propagating and nonpropagating modes in waveguides and ducts of small dimensions, including thermal and viscous losses. The Thermoviscous Acoustics, SLNS Approximation Interface also solves the equations in the frequency domain, while using an approximate and computationally lean method valid in most cases.

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

The Thermoviscous Acoustic–Solid Interaction, Frequency Domain Interface is also described here. This physics interface combines the Thermoviscous Acoustics, Frequency Interface domain and Solid Mechanics Interface together with the Thermoviscous Acoustic-Structure Boundary multiphysics coupling.

The Thermoviscous Acoustic–Shell Interaction, Frequency Domain Interface requires a Structural Mechanics Module license. The multiphysics interface combines the Thermoviscous Acoustics, Frequency domain Interface and the Shell Interface together with the Thermoviscous Acoustic–Structure Boundary multiphysics coupling.

The Thermoviscous Acoustic–Thermoelasticity Interaction Multiphysics Interfaces requires a MEMS Module license. These multiphysics interfaces combine and couple thermoviscous acoustics in the fluid and thermoelasticity in the structures for detailed modeling of damping.

ULTRASOUND

The Ultrasound Interfaces chapter describes the following interfaces:

The Convected Wave Equation, Time Explicit Interface uses the discontinuous Galerkin (dG-FEM) formulation to model the propagating of linear ultrasound waves in the time domain, including the effects of a stationary background flow.

The Nonlinear Pressure Acoustics, Time Explicit Interface uses the discontinuous Galerkin (dG-FEM) formulation to model the propagation of nonlinear high amplitude pressure waves in fluids including dissipation effects.

Both interfaces can couple to The Elastic Waves, Time Explicit Interface using built-in multiphysics couplings.

THE GEOMETRICAL ACOUSTICS BRANCH

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

THE PIPE ACOUSTICS BRANCH

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

THE ACOUSTIC STREAMING BRANCH

The Acoustic Streaming chapter described the multiphysics capabilities necessary to model the phenomenon of acoustic streaming, that is, fluid flow induced by sound. Two formulations exist; the full formulation coupling thermoviscous acoustics to fluid flow and the effective formulation coupling pressure acoustics to fluid flow.

THE MULTIPHYSICS COUPLINGS

The Multiphysics Couplings chapter describes all the multiphysics couplings available with the Acoustics Module. These are the built-in conditions that can couple the physics together; for example, the Acoustic-Structure Boundary couples pressure acoustics to any solid boundary.

STRUCTURAL MECHANICS WITH THE ACOUSTICS MODULE

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

The Piezoelectricity Interface interface is also shortly presented. It combines Solid Mechanics and Electrostatics together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled and the piezoelectric coupling can be formulated using either the strain-charge or stress-charge forms. This interfaces is based on the finite element method (FEM) and can be used in both frequency and time domain.

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

2

Pressure Acoustics Interfaces

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

- The Pressure Acoustics, Frequency Domain Interface
- The Pressure Acoustics, Transient Interface
- The Pressure Acoustics, Boundary Mode Interface
- The Pressure Acoustics, Boundary Elements Interface
- The Pressure Acoustics, Time Explicit Interface
- The Pressure Acoustics, Asymptotic Scattering Interface
- The Pressure Acoustics, Kirchhoff–Helmholtz Interface
- Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces)
- Modeling with the Pressure Acoustics Branch (BEM-Based Interface)
- Modeling with the Pressure Acoustics Branch (DG-FEM-Based Interface)
- Modeling with the Aeroacoustic Flow Source: Flow-Induced Noise
- Theory Background for the Pressure Acoustics Branch
- Theory for the Boundary Impedance Models
- Theory for the Interior Impedance Models

- Theory for the Equivalent Fluid Models
- Theory for the Perfectly Matched Layers in the Time Domain
- References for the Pressure Acoustics Branch

The Pressure Acoustics, Frequency Domain Interface

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

The physics interface can be used for linear acoustics described by a scalar pressure variable. It includes domain conditions to model losses in a homogenized way, so-called equivalent fluid models, for porous materials (Poroacoustics and Anisotropic Poroacoustics), as well as losses in narrow regions waveguides or slits (Narrow Region Acoustics). The plane wave attenuation behavior of the acoustic waves may also be entered as a user-defined quantity, or based on the losses in the atmosphere or ocean. Effective anisotropic material properties can be modeled with an effective anisotropic density formulation. Domain features also include Background Pressure Field, a Heat Source, a Flame Model, as well as general user-defined monopole and dipole domain sources. An Aeroacoustic Flow Source or flow-induced noise source from turbulence can be included when coupled to an LES or DES simulation (this requires the CFD Module).

The physics interface solves the Helmholtz equation in the frequency domain for given frequencies, or as an eigenfrequency or mode analysis study. The harmonic variation of all fields and sources is given by $e^{i\omega t}$ using the $+i\omega$ convention.

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

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

The governing equations and boundary conditions are formulated using the total pressure p_t with a so-called scattered field formulation. In the presence of a Background Pressure Field defining a background pressure wave p_b (this could, for

example, be an incident plane wave), the total acoustic pressure p_t is the sum of the pressure solved for p (which is then equal to the scattered pressure p_s) and the background pressure wave: $p_t = p + p_b$. The equations then contain the information about both the scattered field and the background pressure field.

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

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

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

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

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

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

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Normal displacement	d_n	meter	m
Source location	$\mathbf{x}_0$	meter	m
Wave direction	$\mathbf{e}_k$	(dimensionless)	I

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

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

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

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

SETTINGS

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

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

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

EQUATION

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

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

PHYSICS SYMBOLS

Select to **Enable physics symbols** (selected per default). This shows the physics symbols, like for example, normals or symmetry planes in the Graphics window when applicable.

PRESSURE ACOUSTICS EQUATION SETTINGS

In this section, you can add out-of-plane information defining an out-of-plane wave number k_z or an azimuthal wave number $k_\mathrm{m} = m/r$ through the mode number m. Add if applicable:

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

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

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

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

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

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

Note that when performing a mode analysis study in 2D (solving for the out-of-plane wave number) the **Out-of-plane wave number** property is not used as this is now the variable solved for.

Scaling Factor and Nonreflecting Boundary Condition Approximation

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

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

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

GLOBAL PORT SETTINGS

Select the **Port sweep settings** as:

- **No port sweep** (the default).
- **Activate port sweep** to enable the port sweep functionality**.**
- The **Legacy port sweep** option is only present for backward compatibility reasons.

The **Activate port sweep** option is used to compute the full scattering matrix when Port conditions are used. For more details see The Port Sweep Functionality subsection. The section only exists for 3D and 2D axisymmetry.

Select the **Mode shape normalization** as **Amplitude normalization** (the default) or **Power normalization**. This setting controls if the mode shapes are normalized to have a unit maximum amplitude or carry unit power. The selection determines how the scattering matrix is to be interpreted.

SOUND PRESSURE LEVEL SETTINGS

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

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

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

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

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

TYPICAL WAVE SPEED

Enter a value or expression for the **Typical wave speed for perfectly matched layers** c_ref (SI unit m/s). The default is `real(acpr.c_c)` and the value is automatically taken from the material model. If several materials or material models are used it is best practice to add one PML for each. This will ensure that the typical wavelength is continuous within each PML feature.

DISCRETIZATION

From the list select the element order and type (Lagrange or serendipity) for the **Pressure**, the default is **Quadratic Lagrange**.

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

To see all settings in this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

DEPENDENT VARIABLES

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

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

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

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

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

The Pressure Acoustics, Frequency Domain Interface has Domain, Boundary, Edge, Point, and Pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows® users), **Physics** context menu (macOS or Linux® users), or

right-click to access the context menu (all users). Continuity in the total pressure is the default condition on interior boundaries.

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

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

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

- Aeroacoustic Flow Source
- Anisotropic Acoustics
- Anisotropic Poroacoustics
- AWE Expression
- Axial Symmetry
- Background Pressure Field
- Circular Source (for 2D Axisymmetric Components)
- Continuity
- Cylindrical Wave Radiation
- Dipole Point Source
- Dipole Domain Source
- Exterior Field Calculation
- Flame Model
- Heat Source
- Impedance
- Incident Pressure Field
- Interior Normal Acceleration
- Interior Normal Displacement
- Interior Normal Velocity
- Interior Impedance/Pair Impedance
- Interior Perforated Plate/Pair Perforated Plate
- Interior Sound Hard Boundary (Wall)
- Interior Lumped Speaker Boundary
- Initial Values
- Line Source
- Line Source on Axis
- Lumped Port
- Lumped Speaker Boundary
- Matched Boundary
- Monopole Point Source
- Monopole Domain Source
- Narrow Region Acoustics
- Normal Acceleration
- Normal Displacement
- Normal Velocity
- Quadrupole Point Source
- Perfectly Matched Boundary
- Periodic Condition
- Plane Wave Radiation
- Point Sources (for 2D Components)
- Poroacoustics
- Port
- Port Reference Axis
- Pressure Acoustics
- Pressure
- Pressure (Point Condition)
- Sound Hard Boundary (Wall)
- Sound Soft Boundary
- Spherical Wave Radiation
- Symmetry
- Thermoviscous Boundary Layer Impedance
- Transfer Matrix Coupling

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

In the *COMSOL Multiphysics Reference Manual*, see Table 2-4 for links to common sections and Table 2-5 for common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Pressure Acoustics

The **Pressure Acoustics** node adds the equations for time-harmonic (frequency domain) and eigenfrequency acoustics modeling of classical acoustics solving Helmholtz equation. In the **Settings** window, define the properties for the acoustics model such as material properties and bulk attenuation. Define the so-called **Model Inputs** to the properties including the absolute background pressure and temperature, that is the quiescent ambient properties.

The advanced fluid models are defined using individual physics feature nodes: Poroacoustics, Narrow Region Acoustics, and Anisotropic Acoustics. In the time domain, nonlinear effects can be included using the Nonlinear Acoustics (Westervelt) Contributions node.

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

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

MODEL INPUTS

Select, **User defined** (the default), **Common model input**, or an input from another physics interface, if applicable.

The values of the quiescent (or background) **Temperature** T (SI unit: K) and **Absolute pressure** p_A (SI unit: Pa) are entered in the **Model Inputs** section if required by the material properties (other inputs like relative humidity or salinity can also appear). The

values of the model inputs can influence the material properties like, for example, the density and speed of sound, depending on their definition in the **Materials** node. In general, if the model includes a material property that depends on a model input, the corresponding text field will be enabled.

Details about the Model Input and the Default Model Inputs are found in the Global and Local Definitions chapter of the *COMSOL Multiphysics Reference Manual*.

If **Ideal gas** is selected as the **Fluid model**, both the **Temperature** T and **Absolute pressure** p_A fields are enabled. In this case, these input options are always available.

In addition, the **Temperature** T and **Absolute pressure** p_A can be picked up from another physics interface where the fields have been calculated. For example, select a temperature field defined by a *Heat Transfer* interface or a *Nonisothermal Flow* interface (if any). Or, if applicable, select a pressure as defined by a *Fluid Flow* interface present in the model. For example, select **Absolute pressure (spf)** to use the absolute pressure defined by a *Laminar Flow* interface **spf**.

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

Detailed convected acoustic models that take into account the full background flow (including the movement of the fluid) are available among the Aeroacoustics Interfaces. For flow-induced noise simulations based on the Lighthill analogy see the Aeroacoustic Flow Source.

PRESSURE ACOUSTICS MODEL

To define the properties of the fluid, select a **Fluid model** from the list:

- **Linear elastic** (the default) for defining the classical lossless fluid: Go to Linear Elastic Fluid Model.
- **User-defined attenuation** for defining losses through an attenuation coefficient: Go to User-Defined Attenuation Fluid Model.
- **Atmosphere attenuation** defines the standard attenuation of atmospheric (moist) air: Go to Atmosphere Attenuation Fluid Model.

- **Ocean attenuation** defines attenuation in the seawater of the ocean: Go to Ocean Attenuation Fluid Model.
- **Viscous**, **Thermally conducting**, and **Thermally conducting and viscous** defines the classical thermoviscous attenuation model: Go to Thermally Conducting and/or Viscous Fluid Model.
- **General dissipation** defines losses through the sound diffusivity: Go to General Dissipation Fluid Model.
- **Ideal gas** to define the properties of an ideal gas: Go to Ideal Gas Fluid Model.

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

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

Losses in porous materials are defined in Poroacoustics. Thermoviscous boundary layer losses in narrow regions of constant cross section (like waveguides) can be modeled using Narrow Region Acoustics.

Linear Elastic Fluid Model

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

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

- For **Density and speed of sound**, define the **Speed of sound** c (SI unit: m/s) and **Density** ρ (SI unit: kg/m^3).

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

User-Defined Attenuation Fluid Model

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

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

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

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

Do not confuse the *amplitude attenuation coefficient* α with the *intensity attenuation coefficient* m most often defined in room acoustics. The two are simply related through $2\alpha = m$. The difference stems from how the intensity scales in the propagation direction: $I(x) = I_0 \cdot \exp(-2\alpha x) = I_0 \cdot \exp(-mx)$.

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

Select an **Attenuation type**: **Attenuation coefficient, Np per unit length** (the default) to define an attenuation coefficient α in Np/m (nepers per meter), **Attenuation coefficient, dB per unit length** to define an attenuation coefficient α in dB/m (decibel per meter), or **Attenuation coefficient, dB per wavelength** to define an attenuation coefficient α in dB/λ (decibel per wavelength). The last option is often used in underwater acoustics

and uses a slightly different formulation than the first two, see the theory section. For any selection, enter a value or expression in the **Attenuation coefficient** α field.

	Theory: User-Defined Attenuation Fluid Model

Atmosphere Attenuation Fluid Model

The **Atmospheric attenuation** model defines attenuation in atmospheric air that follows the ANSI standard S1.26-2014, see Ref. 6 (Appendix B), 44, 45, and 46 for details. The model describes attenuation due to thermal and viscous effects (primarily pure air), the relaxation processes for nitrogen and oxygen, and the dependency on atmospheric pressure (absolute pressure), temperature, and relative humidity. The attenuation in air is important for propagation over large distances and for high frequency processes. This also means that the attenuation effect is more important in ray tracing simulations where propagation can be simulated over much larger distances, see The Ray Acoustics Interface.

When the model is selected the **Model Input** section includes inputs for the **Temperature** T (SI unit: K), the **Absolute pressure** p_A (SI unit: Pa), and the **Relative humidity** ϕ (SI unit: 1). In the **Pressure Acoustic Model** section the default **Speed of sound** c (SI unit: m/s) and **Density** ρ (SI unit: kg/m^3) are taken **From material**. For **User defined**, enter other values or expressions for one or both options.

Ocean Attenuation Fluid Model

The **Ocean attenuation** model defines attenuation in seawater of the ocean. The model is based on a semianalytical model with parameters that are based on extensive measurement data. It includes effects due to viscosity in pure water, the relaxation processes of boric acid and magnesium sulfate, as well as depth, temperature, practical salinity, and pH value. For further details, see Ref. 47, 48, 49, 50, and 51. The ocean attenuation model is important in ray tracing simulations where propagation can be simulated over much larger distances, see The Ray Acoustics Interface.

When the model is selected, the **Model Input** section includes inputs for the **Temperature** T (SI unit: K), the **Depth** D (SI unit: m), and the **Practical salinity** S_p (SI unit: 1); the default is 35. In the **Pressure Acoustic Model** section, the default **Speed of sound** c (SI unit: m/s) and **Density** ρ (SI unit: kg/m^3) are taken **From material**. For **User defined**,

enter other values or expressions for one or both options. Enter a value for the **pH value** pH (SI unit: 1); the default is 8.

	The **Practical salinity** is defined on the Practical Salinity Scale. It represents a specific way of measuring salinity through electric conductivity. Practical salinity is a unit-less quantity, although it can be thought of as given in units of g/kg. The default value for the **Practical salinity** S_p is 35. For details see Ref. 57.

	For a discussion about air and water materials as commonly used in acoustics, the attenuation in (sea) water and (humid) air, and other material properties, see the Acoustic Properties of Fluids chapter of this manual.

Thermally Conducting and/or Viscous Fluid Model

The **Viscous**, the **Thermally conducting**, and the **Thermally conducting and viscous** fluid models essentially add the same attenuation model. Here, the attenuation is due to bulk viscous and/or thermal losses. This type of model is relevant in highly viscous fluids or thermally conducting fluids when acoustic waves are traveling over large distances (relative to the wavelength). This is not a model for viscous and thermal boundary layer losses in narrow regions (see Narrow Region Acoustics). The models define the classical thermoviscous attenuation α_{tv} properties of a fluid and can be applied in cases when, for example, relaxation processes are not important. The classical thermoviscous amplitude attenuation is given by

$$\alpha_{tv} = \frac{\omega^2}{2\rho c^3}\left(\frac{4}{3}\mu + \mu_B + \frac{k(\gamma-1)}{C_p}\right)$$

In atmospheric air or salt water where relaxation processes are important use the **Atmosphere attenuation** or the **Ocean attenuation** model as they capture these effects and also their dependency on, for example, relative humidity or salinity.

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

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

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

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

- Viscous power dissipation density (`acpr.diss_visc`)
- Thermal power dissipation density (`acpr.diss_therm`)
- Combined total power dissipation density (`acpr.diss_tot`)

- Viscous Fluid Model
- Thermally Conducting Fluid Model
- Thermally Conducting and Viscous Fluid Model

General Dissipation Fluid Model

For **General dissipation** enter the **Speed of sound** c (SI unit: m/s), the **Density** ρ (SI unit: kg/m^3), and the **Sound diffusivity** δ (SI unit: m^2/s). By default they are taken **From material** and for **User defined** enter other values or expressions.

The sound diffusivity is automatically calculated for the classical thermoviscous case using either the **Viscous**, **Thermally conducting** or the **Thermally conducting and viscous** options (see below). For nonideal fluids like tissue, the sound diffusivity is measured directly instead and can be entered with the **General dissipation**.

Ideal Gas Fluid Model

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

- Select a **Gas constant type**: **Specific gas constant** R_s (SI unit: J/(kg·K) (the default) or **Mean molar mass** M_n (SI unit: kg/mol). For **Mean molar mass**, the molar gas

constant (universal gas constant) $R = 8.314$ J/(mol·K) is used as the built-in physical constant.

- From the **Specify Cp or** γ list, select **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)) (the default) or **Ratio of specific heats** γ. For common diatomic gases such as air, $\gamma = 1.4$ is the standard value.

Poroacoustics

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

About the Poroacoustics Models

POROACOUSTICS MODEL

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

FLUID PROPERTIES

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

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

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

If any other than the default **Delany-Bazley-Miki** is selected, the following properties are also required, depending on the selection. These material parameters are necessary as

the more advanced models include the losses associated with viscosity and thermal conduction in a more or less detailed way:

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

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

POROUS MATRIX PROPERTIES

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

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

- Delany–Bazley–Miki
- Zwikker–Kosten
- Attenborough
- Wilson
- Johnson–Champoux–Allard
- Johnson–Champoux–Allard–Lafarge
- Johnson–Champoux–Allard–Pride–Lafarge
- Three-Parameter Approximation JCAL Model
- Williams Effective Density Fluid Model (EDFM)
- Wood

Delany–Bazley–Miki

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

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

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

The exception is the **Komatsu** model, where $X = 2 - \log_{10}(f/R_f)$ (which is a completely empirical quantity). All the models are applicable for materials with a porosity ε_p close to 1. The model information is listed in Table 2-2. See also Ref. 23 and the relevant section in About the Poroacoustics Models for further details.

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

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

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

MODEL NAME	APPLICABILITY
Mechel, rock fiber, high X	Rock fiber: $X \geq 0.025$
Komatsu	Glass and rock wool: $6 \cdot 10^3 \leq R_f \leq 73 \cdot 10^3$ Pasm^{-2}
Modified Allard and Champoux	$45 \leq f \leq 11 \cdot 10^3$ Hz

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

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

Zwikker–Kosten

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

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

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

Attenborough

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

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

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

Wilson

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

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

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

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

Johnson–Champoux–Allard

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

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

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

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

Johnson–Champoux–Allard–Lafarge

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

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

Johnson–Champoux–Allard–Pride–Lafarge

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

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

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

Three-Parameter Approximation JCAL Model

The **Three-parameter approximation JCAL model** represents an approximation to the JCAL model. Instead of requiring 6 parameters as input for the porous matrix it only requires three. The porosity as well as two parameters that relate to the topology of the pores are needed (their size and distribution). Two different approximation methods exist for the density and bulk modulus. Select the **Density and bulk modulus approximation** to either **Johnson-Champoux-Allard-Pride-Lafarge** (the default) or **Padé approximation**. Then if necessary enter the values for (default is **From material**):

- **Porosity** ε_p (dimensionless).
- **Median pore size** $\bar{s}$ (SI unit: m).
- **Standard deviation in pore size distribution** σ_s (dimensionless).

Details about the model can be found in Ref. 58, 59, 60, 61,62, and About the Poroacoustics Models.

Williams Effective Density Fluid Model (EDFM)

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

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

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

Wood

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

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

Narrow Region Acoustics

The **Narrow Region Acoustics** node defines a fluid model for viscous and thermal boundary-layer-induced losses present in narrow channels and ducts of constant cross section. The losses due to the viscous and thermal dissipation in the acoustic boundary layer are homogenized and smeared on the fluid. This type of model has a low computational cost compared to a thermoviscous acoustic model.

The equivalent-fluid model can be used in long tubes of constant cross section (or only in a slowly varying cross section) instead of a fully detailed thermoviscous acoustics model. Accurate results, with respect to the overall damping, will be obtained when the waveguide length is (much) larger than the radius. As soon as the length becomes comparable to the radius, the resistive losses associated with an impedance jump at the end of the waveguide (if a cross section change is present in the model), will start to become relatively more important. The impedance jump losses are not captured by the **Narrow Region Acoustics** formulation. A full thermoviscous acoustic model is necessary here. From an engineering point-of-view, the use of the **Narrow Region Acoustics** model can be a good choice, even if not all losses are captured, as the method is fast and can yield useful results for comparative studies.

The models are applicable for all fluids, that is, both gases and liquids. In the case of waveguides of varying thicknesses or in general other structures with curved surfaces, an alternative is to use the Thermoviscous Boundary Layer Impedance condition.

About the Narrow Region Acoustics Models

DUCT PROPERTIES

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

The **Slit**, **Circular duct**, **Rectangular duct**, and **Equilateral triangular duct** models are applicable as long as the cross-section dimension is much smaller than the wavelength and the boundary layer thickness is smaller than the wavelength. The cross-section parameter can be a slowly varying function of space. These are known as a low reduced frequency (LRF) model.

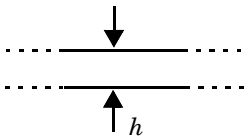
- For **Wide duct approximation** enter a **Hydraulic diameter** H_d (SI unit: m).

  This model can be used in ducts of any cross section as long as the hydraulic diameter of the duct (four times the cross-section area divided by the circumference) is much larger than the viscous boundary layer thickness.

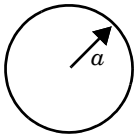
- For **Very narrow circular duct (isothermal)** enter the duct **Radius** a (SI unit: m).

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

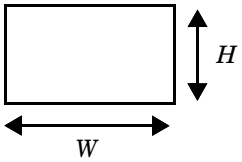
- For **Slit** enter the slit **Height** h (SI unit: m). Use this model in narrow slit domains to include the damping and attenuation that occurs here because of the losses in the viscous and thermal boundary layer.



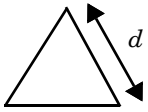
- For **Circular duct** enter the duct **Radius** a (SI unit: m). This model is useful for modeling the damping and attenuation that occurs when acoustic waves propagate in all tubing systems of small cross-section dimensions.



- For **Rectangular duct** enter the duct **Side lengths** W and H (SI unit: m). Use this model for waveguides and ducts with a rectangular cross section. Also see Advanced Physics Options for additional settings.



- For **Equilateral triangular duct** enter the duct **Side length** d (SI unit: m). Use this model for waveguides and ducts with an equilateral triangular cross section.



To determine the complex propagation constants for a waveguide, of arbitrary cross section, use The Thermoviscous Acoustics, Boundary Mode Interface. Apply it on the cross-section geometry of the waveguide. The interface solves for the propagating modes and includes all losses in detail. The complex wave number k_c is then given by the plane wave mode solved for. This is the variable `tabm.kn`. The predefined variable `tabm.Zc` gives the (lumped) specific characteristic complex impedance Z_c. Search for the mode nearest to the (lossless) plane wave mode.

FLUID PROPERTIES

Based on your selection in **Duct Properties**, different fluid properties need to be entered.

For **User defined**, enter the value for the **Complex wave number** k_c (SI unit: rad/m) and (specific characteristic) **Complex acoustic impedance** Z_c (SI unit: Pa·s/m). These values can be used to model the propagation in ducts of arbitrary cross sections.

For the other duct types, several other fluid properties are needed. The default values for the following are taken **From material**.

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

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

- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K))
- **Thermal conductivity** k (SI unit: W/(m·K))

ADVANCED PHYSICS OPTIONS

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

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

- *Lumped Receiver Connected to Test Setup with a 0.4-cc Coupler*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_receiver_04cc**
- *Generic 711 Coupler — An Occluded Ear-Canal Simulator*: Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/generic_711_coupler**

The **Narrow Region Acoustics** models are so-called equivalent fluid models that have a nontrivial (nonlinear) dependency on the frequency. This means that performing an eigenfrequency analysis should be considered carefully, see Eigenfrequency Study.

Moreover, the linearization process (linearization with respect to the frequency) of the underlying mathematical models can cause numerical problems (an error message is thrown). For the **Rectangular duct** one remedy is to decrease the number of terms used in the sum, for the **Circular duct** it can be necessary to switch to another model, like the **Very narrow circular duct (isothermal)**.

Anisotropic Acoustics

The **Anisotropic Acoustics** node is used to define a fluid with properties that can be modeled with an effective anisotropic density. The feature can be used to model metamaterials in a homogenized manner or when modeling fibrous materials with anisotropic properties. The material properties are defined through an effective bulk modulus K and an effective density tensor ρ. The density variable can be defined as being isotropic (a scalar), a diagonal tensor, or a symmetric tensor. The resulting (generalized) Helmholtz equation solved in the frequency domain is then

$$\nabla \cdot (-\rho^{-1}(\nabla p_t - \mathbf{q}_d)) - \frac{\omega^2 p_t}{K} = Q_m$$

where it should been emphasized that ρ^{-1} is here a tensor. The resulting system will have principal speeds of sound that are (square root of) the eigenvalues and directions that are the eigenvectors of the quantity $\mathbf{c}^2 = \rho^{-1}K$.

COORDINATE SYSTEM SELECTION

Select the **Coordinate system** in which the density tensor is defined. The default is the **Global coordinate system**.

Coordinate systems with directions which change with time should not be used.

ANISOTROPIC ACOUSTICS

Enter the **Effective bulk modulus** K (SI unit: Pa) taken **From material**. For **User defined** enter other values or expressions for any or all options.

Enter the **Effective density** ρ (SI unit: kg/m^3) taken **From material**. For **User defined** enter other values or expressions for any or all options. Also select the representation for the density tensor either **Isotropic** (the default), **Diagonal**, or **Symmetric**.

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

Anisotropic Poroacoustics

The **Anisotropic Poroacoustics** node defines a domain with a porous material that has anisotropic properties. The material is modeled in a homogenized way using a so-called equivalent fluid model. Several models exist to define the attenuation and dispersion experienced by pressure waves as they propagate in the porous domain. Anisotropic properties can be defined for the relevant poroacoustic properties, namely the flow resistivity R_f, the tortuosity factor τ_∞, and the viscous characteristic length L_v.

Three material models from the (isotropic) Poroacoustics feature exist for the anisotropic formulation (see below). These models define properties that result in an anisotropic effective density $[\rho]_{ij}$ and a scalar effective bulk modulus K. This is required for the formulation fit into the Anisotropic Acoustics framework.

For detailed modeling of anisotropic porous materials including the elastic porous skeleton, see the Anisotropic Poroelastic Material of the The Poroelastic Waves Interface.

Anisotropic Porous Absorber: Application Library path **Acoustics_Module/Building_and_Room_Acoustics/anisotropic_porous_absorber**

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains all applicable coordinate systems in the component. The coordinate system is used for interpreting directions of orthotropic and anisotropic material data when porous properties are presented in a local system.

POROACOUSTICS MODEL

Select the **Poroacoustics model** as **Johnson-Champoux-Allard (JCA)** (the default), **Johnson-Champoux-Allard-Lafarge (JCAL)**, or **Johnson-Champoux-Allard-Pride-Lafarge (JCAPL)**.

FLUID PROPERTIES

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

- **Density** ρ_f (SI unit: kg/m^3) of the saturating fluid (the fluid density)
- **Heat capacity at constant pressure** C_p (SI unit: $J/(kg \cdot K)$).
- **Ratio of specific heats** γ (dimensionless).
- **Thermal conductivity** k (SI unit: $W/(m \cdot K)$).
- **Dynamic viscosity** μ (SI unit: $Pa \cdot s$).

POROUS MATRIX PROPERTIES

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

In the **Anisotropic Poroacoustics** model, the **Flow resistivity** $[R_f]_{ij}$, the **Tortuosity factor** $[\tau_\infty]_{ij}$, and the **Viscous characteristic length** $[L_v]_{ij}$ can have anisotropic properties. The remaining (isotropic) parameters are described in the relevant models in the Poroacoustics section.

For the three inputs, select either **From material** (the default) or **User defined**. For the **User defined** option, select if the material property is entered as **Isotropic** or **Diagonal** (the properties only have nonzero components in the diagonal for i = j). When taken **From material**, the properties can now be defined as diagonal tensor data, when defining a material in the **Materials** node.

Background Pressure Field

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

$$p_t = p_b + p_s$$

This feature sets up the equations in a so-called scattered field formulation where the dependent variable is the scattered field $p = p_s$. In a model where the background pressure field is not defined in all acoustic domains (or it is different), continuity is automatically applied in the total field p_t on interior boundaries between domains.

The background pressure field has built-in options for plane waves, spherical waves, and cylindrical waves. It can also be defined by a user defined analytical expression, that is a function of space, or it can be given by the solution to another acoustics problem.

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

For exciting waveguide structures with a given wave type, it is recommended to use the Port or Lumped Port conditions, instead of a background pressure field. If still used, the feature should be backed by a PML or radiation condition to model an infinite waveguide.

BACKGROUND PRESSURE FIELD

Select a **Background pressure field type**: **Plane wave** (the default), **Cylindrical Wave**, **Spherical Wave**, or **User defined**. For 2D axisymmetric models, it is possible to expand the plane wave in their cylindrical harmonics.

To set up and calculate the intensity variables for the total, background, and scattered acoustic fields, select **Calculate background and scattered field intensity** at the bottom of the settings window. This requires an additional input of the **Density** ρ (SI unit: kg/m^3) for the defined background pressure field. Selecting this option will also define the velocity variables for the various fields. Enabling this is useful for analyzing reflection problems, where the ratio between incident (background) and reflected (scattered) field power is of interest.

Plane Wave

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

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

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

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

In 2D axisymmetric models, the incident fields take a slightly different form due to the geometrical restrictions. Only the z-component for the **Wave direction** $\mathbf{e}_k$ can be entered. Per default, the wave can only travel in the axial direction since this is the only axisymmetric form of a plane wave.

By selecting **Enable plane wave expansion**, the plane wave is expanded in its cylindrical harmonics and a general **Wave direction** $\mathbf{e}_k$ can be selected. This sets up a wave of the form:

$$p_i = e^{i\phi} 2^{\text{sign(m)}} i^m J_m\left(-k_s \frac{(\mathbf{x}\cdot(\mathbf{e}_k\cdot\mathbf{e}_r)\mathbf{e}_r)}{|\mathbf{e}_k|}\right)\exp\left(-ik_s \frac{(\mathbf{x}\cdot(\mathbf{e}_k\cdot\mathbf{e}_z)\mathbf{e}_z)}{|\mathbf{e}_k|}\right)$$

where m is the **Azimuthal mode number** specified in the Pressure Acoustics Equation Settings. To expand the solution, it is necessary to run a **Parametric Sweep** over the mode number from 0 up to the desired resolution.

An example using the **Enable plane wave expansion** functionality can be found online in the Application Gallery. The model *Plane Wave Scattering off a 2D Axisymmetric Object: Plane Wave Expansion Approach* is found at www.comsol.com/model/51311

Cylindrical Wave

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

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

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

For **Cylindrical Wave**, enter a **Pressure amplitude at reference distance** p_0 (SI unit: Pa), the **Speed of Sound** c (SI unit: m/s) of the fluid defining the field, either **From material** or **User defined**, enter a **Source Location** $\mathbf{x}_0$ (SI unit: m), a **Source Axis** $\mathbf{e}_{sa}$ (the vector does not need to be normalized), and a **Phase** ϕ (SI unit: rad).

In 2D axisymmetric models, the incident fields take a slightly different form due to the geometrical restrictions. Neither source location nor axis is needed as the source is always located on the z-axis.

Spherical Wave

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

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

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

For **Spherical Wave**, enter a **Pressure amplitude at reference distance** p_0 (SI unit: Pa), the **Speed of Sound** c (SI unit: m/s) of the fluid defining the field, either **From material** or **User defined**, enter a **Source Location** $\mathbf{x}_0$ (SI unit: m), and a **Phase** ϕ (SI unit: rad).

In 2D axisymmetric models, the incident fields take a slightly different form due to the geometrical restrictions. Only enter the **Source Location, z coordinate** z_0 (SI unit: m) since the source is always located on the z-axis in a 2D axisymmetric model.

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

User Defined

For the **User defined** option, enter an expression for the **Background pressure field** p_b (SI unit: Pa).

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

Initial Values

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

INITIAL VALUES

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

Monopole Domain Source

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

Dedicated features define advanced domain sources of the monopole type, such as the Heat Source or the Flame Model. In the time domain, there is the Nonlinear Acoustics (Westervelt) Contributions.

MONOPOLE SOURCE

Enter a **Monopole domain source** Q_m (SI unit: $1/s^2$). This can be any user defined expression or constant.

Dipole Domain Source

Use the **Dipole Domain Source** node to add the domain source term $\mathbf{q}_d$ to the governing equation. This source represents a domain volumetric force. This source is typically stronger in two opposite directions. In 2D axisymmetric models, a force expression should typically have 0 as the r-component. In advanced models, this term can, for example, be used to represent a uniform constant background flow convecting the sound field. It can, in general, be used to model user-defined contributions to the governing equations. Add this node from the **More** submenu.

DIPOLE SOURCE

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

Heat Source

Use the **Heat Source** feature to add a domain heat source that generates sound. Add this node from the **More** submenu. If a fluid is heated and cooled rapidly, thermal expansion and contraction will generate acoustic waves. This could be a source representing a pulsating laser beam or another general heat source. For a flame source in a combustion simulation use the Flame Model. The feature adds a monopole domain source to the right-hand side defined as

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

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

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

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

MODEL INPUT

The **Model Input** section is always visible in the **Heat Source** feature as the ambient temperature T is always needed to compute the coefficient of thermal expansion.

DOMAIN HEAT SOURCE

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

FLUID PROPERTIES

Enter the additional fluid properties necessary to define the heat source. Enter the **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)) and the **Ratio of specific heats** γ (dimensionless). The default for both is **From material**. Select **User defined** to enter a value or expression directly.

Aeroacoustic Flow Source

The **Aeroacoustic Flow Source** feature is used to set up aeroacoustic flow-induced noise sources in the pressure acoustics model. The feature is best used when coupling an LES or DES fluid flow model to the **Aeroacoustic Flow Source** feature using the Aeroacoustic Flow Source Coupling multiphysics coupling and the dedicated Transient Mapping study. The feature only exists in 3D. The feature can also be used with user-defined expressions for a Lighthill like source expressed in terms of a stress tensor T_{ij} (SI unit: N/m^2). Add this node from the **More** submenu.

AEROACOUSTIC FLOW SOURCE

Select the **Acoustic analogy** as **Lighthill** (the default) or **Aeroacoustic wave equation**. The selection should match the option in the Aeroacoustic Flow Source Coupling multiphysics coupling. When the multiphysics coupling is used, the **Stress tensor** (T), **Normal mass flow on exterior boundaries** (m_n), or the **Flow pressure variation, second time derivative** (p_{tt}^{var}) field are automatically coupled. If user-defined expressions are selected, enter an expression. The user-defined expressions correspond to the Fourier components of the different flow sources terms.

Cavity Flow Noise tutorial available in the COMSOL Application Gallery at: https://www.comsol.com/model/cavity-flow-noise-105021

Best practices for modeling with the **Aeroacoustic Flow Source** and the associated multiphysics functionality is presented in the Modeling with the Aeroacoustic Flow Source: Flow-Induced Noise section.

Flame Model

The **Flame Model** feature is used to define the heat source using a flame model, typically for the stability analysis in a combustion setup. The heat source depends on the

acoustic field and is defined in terms of the so-called n-τ model. Add this node from the **More** submenu.

In combustion engines, the heat release depends on acoustic oscillations of the fresh fuel supply and the acoustic oscillations are effected by the heat release. This can result in acoustic modes becoming either unstable or damped. The heat release in the **Flame Model** is modeled with the n-τ model where the heat release in the domain at time t is coupled to the acoustic field at a point or boundary at the time $t - \tau_u$, see Ref. 75.

The feature adds a monopole domain source Q_m to the right-hand side, defined as

$$Q_\mathrm{m} = \frac{\alpha_\mathrm{p}}{\rho C_\mathrm{p}} i\omega Q_\mathrm{heat} \qquad \alpha_\mathrm{p} = \frac{1}{c}\sqrt{\frac{C_\mathrm{p}(\gamma-1)}{T}}$$

$$Q_\mathrm{heat} = \frac{Q_s n_u}{U_s}\exp(-i\omega\tau_u)(\mathbf{u}(x_\mathrm{ref})\cdot\mathbf{n}_\mathrm{ref})$$

where the (isobaric) coefficient of thermal expansion α_p is defined in terms of the speed of sound c, heat capacity at constant pressure C_p, ambient temperature T (this value is taken from the **Model Input** section and can be space dependent), and the ratio of specific heats γ. Q_s is the steady heat release, U_s is the steady flow speed, n_u is the interaction index, τ_u is the time-lag, x_ref is the reference position, and $\mathbf{n}_\mathrm{ref}$ is the acoustic reference vector.

MODEL INPUT

The **Model Input** section is always visible in the **Flame Model** feature as the ambient temperature T is always needed to compute the coefficient of thermal expansion.

FLAME MODEL

Enter the properties defining the flame model. Specify the **Interaction index** n_u, the **Time lag** τ_u (SI unit: s), the **Steady heat release** Q_s (SI unit: $\mathrm{W/m^3}$), and the **Steady flow** U_s (SI unit: m/s).

ACOUSTIC REFERENCE

Select the acoustic reference point or surface that defines the reference velocity for the flame model. The heat release depends on the velocity at a point or surface and this point or surface has to be selected independently from the domain selection for the flame model. The heat release depends on the reference velocity because of its dependency of the rate of supplied fuel. Therefore, the acoustic reference point or surface should be upstream from the **Flame Model** domain.

Select the **Acoustic reference** as either **Reference point** or **Reference boundary**. Depending on the selection, make sure to select either a point in the **Reference Point** section or a boundary in the **Reference Boundary** section.

- For the **Reference point** option, enter the **Acoustic reference vector** N_{ref} that defines the evaluation direction for the acoustic velocity.
- For the **Reference boundary** option, select the **Source of acoustic reference vector** as either **Automatic** or **User defined**. For the **Automatic** option, the direction is taken as the normal to the boundary. The direction is shown as a red arrow in the Graphics window. To reverse the direction, select **Reverse normal direction**. For the **User defined** option enter the **Acoustic reference vector** N_{ref} that defines the evaluation direction for the acoustic velocity.

FLUID PROPERTIES

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

Active Flame Validation: Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/active_flame_validation**

Sound Hard Boundary (Wall)

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

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_{\text{c}}}(\nabla p_{\text{t}}-\mathbf{q}_{\text{d}})\right)=0$$

For zero dipole domain source ($\mathbf{q}_{\text{d}}=\mathbf{0}$) and constant fluid density ρ_{c}, this means that the normal derivative of the pressure is zero at the boundary:

$$\frac{\partial p_{\text{t}}}{\partial \mathbf{n}}=0$$

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

Axial Symmetry

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

Normal Acceleration

The **Normal Acceleration** adds an inward normal acceleration a_n:

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

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

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

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

NORMAL ACCELERATION

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

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

Normal Velocity

The **Normal Velocity** adds an inward normal velocity v_n, which is assumed to be harmonically oscillating:

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

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

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

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

NORMAL VELOCITY

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

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

Normal Displacement

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

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

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

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

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

NORMAL DISPLACEMENT

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

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

Sound Soft Boundary

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

CONSTRAINT SETTINGS

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

Pressure

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

PRESSURE

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

CONSTRAINT SETTINGS

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

Impedance

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

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

Here, Z_n is the (normal) specific input impedance of the external domain and it has the SI unit Pa·s/m — a pressure divided by a velocity. From a physical point of view, the acoustic input impedance is the ratio between the local pressure and local normal particle velocity. The **Impedance** boundary condition is a good approximation of a

locally reacting surface — a surface for which the normal velocity at any point depends only on the pressure at that exact point.

For plane waves, the specific impedance Z_n is related to the acoustic impedance Z_{ac} (ratio of average surface pressure and flow rate) and the mechanical impedance Z_{mech} (ratio of force and velocity) via the area A of the boundary, according to

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

This relation only defines the relation between average or lumped impedance values at a surface.

All built-in impedance models only exist in the frequency domain. The only exception is the **User defined** impedance, which can be applied also in the time domain. In the frequency domain, the impedance can be any complex-valued number and thus models a surface that is both resistive and reactive. In the time domain, only resistive (real-valued impedance) effects can be included.

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

COMSOL Multiphysics supports the use of the rayl unit when specifying a value of the impedance (1 rayl = 1 Pa·s/m). Two variants exist: `[rayl]` and `[rayl_cgs]`. The latter is the definition of the unit in the cgs unit system. Notice that inconsistent definitions give: `1[rayl_cgs] = 10[rayl] = 10[Pa*s/m]`.

On an impedance boundary, postprocessing variables exist to evaluate the equivalent absorption coefficient. They are the absorption coefficient for normal incidence `acpr.imp1.alpha_n` and for random incidence `acpr.imp1.alpha_ran` (here given for the impedance feature `imp1`).

Notice that the **Impedance** boundary condition cannot directly be combined with a source like **Normal Acceleration**. In cases where such a behavior is desired, modeling a source impedance, this can be achieved by coupling the boundary to an Electrical Circuit model. See, for example, the *Lumped Loudspeaker Driver* model under *Electroacoustic Transducers* in the Application Library.

IMPEDANCE

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

- Simple RCL models define lumped-parameter circuits describing, for example, the mechanical response of a microphone diaphragm. This model is only fully valid for plane waves at normal incidence.
- A set of physiological models enable easy modeling of the surface impedance of human skin and the acoustic loads of the human ear and eardrum. The ear and eardrum models are based on lumped models and only really valid for plane waves at normal incidence. They can be used as engineering approximations in other cases.
- The waveguide end impedances define simplified models of the acoustic losses at the end of pipes and ducts. Like for the RCL impedance, this model is only valid for plane waves at normal incidence at the end of waveguides. It is not valid when higher order modes start to propagate.
- A porous layer backed by a sound-hard wall model can be defined. This condition is of particular to room acoustic simulations as it has some options to handle random incidence with an effective formulation.
- Simple wave-type specific characteristic impedance models exist to address open boundaries in a simplified way.
- An option to define the impedance through the absorption coefficient of the surface.

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

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

Choose an **Impedance model** — **User defined** (the default), **RCL**, **Physiological**, **Waveguide end impedance**, **Porous layer**, **Specific characteristic impedance**, or **Absorption Coefficient**.

User Defined

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

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

RCL

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

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

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

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

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

Physiological

This is a set of simple models to address applications involving interactions of acoustics with the human body. The models comprise human skin, the impedance of the entire

human ear including or excluding the pinna, the outward radiation impedance caused by the pinna, and the inward impedance experienced at the ear drum comprising the drum and the entire inner ear. For the two models of the human ear (with/without pinna), the pressure at the ear drum is automatically calculated. The variable has the form `acpr.imp1.p_ear_drum` and is available for postprocessing.

Note that the model for the ear drum impedance (see Ref. 28–30), also known as the model of Hudde and Engel, is defined in the references up to a maximum frequency of 16 kHz. The whole-ear models are based on the geometry of the ear canal and pinna of a specific ear (see Ref. 31 and appendix in Ref. 30), but person-to-person variations are to be expected. The ear canal geometry used here has an ear canal entrance corresponding to a circle of radius 4.25 mm, that is an area of 56.7 mm^2. For applications where a specific ear canal geometry can be obtained, better results are expected by explicitly modeling this and applying the eardrum impedance at the end. For more details see the theory sections for the Human skin model and Models related to the human ear.

Choose an option from the list: **Human skin**, **Outward human ear radiation**, **Human ear drum**, **Human ear without pinna**, or **Human ear, full**. Then select material properties either **From material** (the default) or **User defined**, for the properties of air (the fluid). These are used to define the losses in the fluid. Select as required:

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

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

Waveguide End Impedance

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

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

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

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

Porous Layer

This choice models the acoustic losses of an incident field on a porous layer of user-defined thickness d backed by a sound-hard wall. The angle of incidence can be controlled to be normal to the surface or to use a specific angle or direction. An automatic option assigns an effective angle of incidence useful for room acoustics simulation. Use this boundary condition as an alternative to modeling the porous layer explicitly using the Poroacoustics feature. All material models from Poroacoustics are implemented in this feature.

Note that the **Porous Layer** condition is not compatible with the Anisotropic Acoustics domain condition. To model a porous layer next to an Anisotropic Acoustics domain, it has to be modeled as a domain using the Poroacoustics feature.

Enter the **Thickness of porous layer** d (Si unit: m), select the **Direction of incident wave**, and select a **Poroacoustic model**. The rest of the settings are the same as for Poroacoustics. For the **Direction of incident wave** select **Normal**, **Automatic**, **User defined**, or **From angle of incidence**.

- For **Normal**, the normal incidence impedance value is used (the angle of incidence is set to 0^o).
- For **Automatic**, the angle of incidence is set to 50^o behind the scenes. This angle gives an on average value, valid for random incidence, and is useful when modeling closed spaces like rooms.
- For **User defined**, enter the **Wave direction** $\mathbf{e}_k$, the default is the surface normal. If a Background Pressure Field feature is present you can, for example, use the wave direction components: `acpr.bpf1.kdirx`, `acpr.bpf1.kdiry`, and `acpr.bpf1.kdirz` (here in 3D, use the appropriate tag).
- For **From angle of incidence**, enter the **Incidence angle** θ (default value: `50[deg]`).

Specific Characteristic Impedance

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

Select a **Wave type**: **Plane wave** (the default), **Cylindrical wave**, or **Spherical wave**. Then enter the **Wave direction** $\mathbf{e}_k$ for the plane wave (default is normal to the surface); the **Radiating field source location** $\mathbf{r}_0$ and **Radiating field source axis** $\mathbf{r}_{ac}$ for the cylindrical wave; or the **Radiating field source location** $\mathbf{r}_0$ for the spherical wave.

Absorption Coefficient

With this option the specific impedance of a boundary is defined through the normal incidence absorption coefficient α_n (SI unit: 1) of the boundary. Since the absorption coefficient carries no phase information, it is also possible to define the phase of the associated reflection coefficient. If no phase is entered, the impedance will be purely resistive (no reactive component is defined). This is typically an acceptable approximation at higher frequencies and it is also the assumption in ray tracing models. Sometimes surfaces are only specified through an absorption coefficient and in such cases using the **Absorption coefficient** option can be a first good approximation.

Enter the **Normal incidence absorption coefficient** α_n (SI unit: 1) and the **Phase** (SI unit: rad).

Symmetry

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

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and

physics modeled. This feature works well for cases like opposing parallel boundaries. In other cases, use the **Destination Selection** section to control the destination. By default, it contains the selection that COMSOL Multiphysics identifies.

BOUNDARY SELECTION

The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default, it contains the selection that COMSOL Multiphysics identifies.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODICITY SETTINGS

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

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

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

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

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

In the time domain, both the **Cyclic symmetry** and the **Floquet periodicity** boundary conditions reduce to the continuity condition. The option exists in 3D and 2D.

	To get correct behavior of the **Periodic Condition**, in particular the **Floquet periodicity** and the **Cyclic symmetry** conditions, it is recommended that the source and destination meshes are identical. This can be achieved using the **Identical Mesh** mesh feature. When the **Periodic Condition** stretches across regions with a mix of default material models, PMLs, background pressure fields, or background acoustic fields, it is recommended to add one **Periodic Condition** for each set of such boundaries.

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

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

EXCLUDED EDGES

To display this section, select **Advanced Physics Options** in the **Show More Options** dialog.

EXCLUDED POINTS

To display this section, select **Advanced Physics Options** in the **Show More Options** dialog.

For more information about the **Excluded Edges** and **Excluded Points** sections, see Excluded Surfaces, Excluded Edges, and Excluded Points in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF DESTINATION

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog. For information about the **Orientation of Destination** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Mapping Between Source and Destination** section, see Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Matched Boundary

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

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

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

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

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

The matched boundary condition is useful for modeling acoustic waves in ducts and waveguides at frequencies below the cutoff frequency for the second excited transverse

mode. In such situations, set $k_1 = \omega/c_c$ and $k_2 = \omega_1/c_c$, where $\omega_1 = 2\pi f_1$, and f_1 is the cutoff frequency for the first excited mode. In general, if several modes exist, use the Port boundary condition. The cutoff frequency or wave number may be found using a Boundary Mode Acoustics model. When $k_1 = k_2 = \omega/c_c$, the matched boundary condition reduces to the time-harmonic plane-wave radiation boundary condition.

MATCHED BOUNDARY

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

Exterior Field Calculation

Use the **Exterior Field Calculation** node to apply the source boundaries for the exterior field transformation set up by the Helmholtz–Kirchhoff integral. You also specify a name for the acoustic exterior field variable (default name is `pext`) used in subsequent postprocessing. The feature allows the calculation and visualization of the pressure field outside the computational domain at any distance including amplitude and phase. Note that when a Background Pressure Field is present, the feature only operates on the scattered field variables and is thus also well suited for analyzing the results of a scattering problem.

The exterior field boundary should be continuous and needs to enclose all sources and scatterers with the desired symmetry types (see below). The feature should be applied on an exterior boundary, a boundary on the inside of a PML, or an interior boundary where the normal direction is continuous along the boundary. Note also that if an interior boundary is used, the material properties should be continuous across the boundary. On exterior radiation boundaries and on the inside boundary of a PML the normals are always automatically adjusted. In all other cases, the direction of the feature normals can be visualized with an arrow surface or line plot in results. The feature uses dedicated item-scoped variables, for example, in 3D the variables to plot are `acpr.efc1.nx`, `acpr.efc1.ny`, and `acpr.efc1.nz`. The normals should point into the source domain (which is outward to the exterior domain). The advanced

physics option **Reverse normal direction** checkbox can be marked if the normal is pointing in the wrong direction.

The exterior field operator and its associated variables are optimized for use in postprocessing. They cannot be used for gradient based optimization studies, like shape and topology optimization, where the sensitivity is necessary.

For gradient based optimization a dedicated operator exists in 3D and 2D axisymmetry for the pressure `pext_opt(x,y,z)` and for the sound pressure level `Lp_pext_opt(x,y,z)`. These two operators can be used to define objective functions, like specifying a target spatial response. The operator only exists when the **Symmetry type** option is set to **Symmetry planes** (the default), this is in particular true when all symmetry planes are set to **Off**.

Several example using the operator exist:

- *Shape Optimization of a Rectangular Loudspeaker Horn in 3D*. Application Library path **Acoustics_Module/Optimization/rectangular_horn_shape_optimization**
- *Optimizing the Shape of a Horn*. Application Library path **Acoustics_Module/Optimization/horn_shape_optimization**
- *Tweeter Dome and Waveguide Shape Optimization*. Application library path **Acoustics_Module/Optimization/tweeter_shape_optimization**

EXTERIOR FIELD CALCULATION

Enter an **Exterior field variable name** for the exterior field acoustic pressure field (the default is `pext`).

Select the **Symmetry type** that applies to the problem. The symmetry type and subsequent settings should match the conditions used in the underlying acoustics problem, for example, if using Symmetry or the Periodic Condition.

Select **Symmetry planes** (the default), **Sector symmetry**, or **Sector symmetry with one symmetry plane**:

- For **Symmetry planes**, if no symmetry conditions are used (the default), the exterior field feature boundaries should form a fully closed surface. If necessary, select and add a condition for one of the Cartesian coordinate planes (with a possible offset)

to model either a symmetry condition in the plane (which is the same as an infinite sound hard boundary) or an antisymmetry condition in the plane (which is the same as an infinite sound soft boundary). The infinite sound hard boundary option is especially useful when modeling systems with an infinite baffle configuration.

For each of these planes, select the type for the condition to be applied in the $x = x_0$, $y = y_0$, or $z = z_0$ planes. Select the type of condition: **Off** (the default), **Symmetry/ Infinite sound hard boundary**, or **Antisymmetric/Infinite sound soft boundary**. Then enter the value for the plane location x_0, y_0, or z_0 (the default is 0 m). This allows an offset of the infinite condition planes along the main coordinate axes.

- The **Sector symmetry** options allows the underlying geometry to represent only a sector (an subdivision of a full 360° rotation). Select and define the properties of the sector. First select the **Transformation** applied to the sector, either **Rotation** (the default) or **Reflection and rotation**. For the latter enter the **Reflection plane** used for the initial reflection of the sector. The selection is restricted to the main coordinate planes (move and rotate the geometry if necessary). The offset distance (x_0, y_0, z_0) is given by the point $\mathbf{a}_0$. Enter the coordinates for the **Point on sector symmetry axis**, $\mathbf{a}_0$ as well as the **Sector symmetry axis direction**, $\mathbf{a}_{\text{dir}}$ (default is $\mathbf{a}_{\text{dir}} = (0, 0, 1)$). Enter the **Number of sectors** n, which has to be an integer and for the **Reflection and rotation** option an even integer. Finally, if the pure **Rotation** option is selected enter an optional **Azimuthal mode number** m (the default it 0). This option is typically only relevant when used together with Periodic Condition.

  Note that the **Sector symmetry** option cannot be combined with an infinite baffle (symmetry plane) used in many transducer models. For this type of setup use the **Sector symmetry with one symmetry plane** option.

- The **Sector symmetry with one symmetry plane** option allows for the combination of sector symmetry and a single symmetry plane, under certain restrictions. First, select the single **Symmetry plane** and symmetry **Condition**. The offset distance (x_0, y_0, z_0) is given by the point $\mathbf{a}_0$. Then follow the same settings as for the **Sector symmetry** option. Note that the sector symmetry axis direction $\mathbf{a}_{\text{dir}}$ is automatically defined as being normal to the single symmetry plane.

When one of the symmetry plane conditions is enabled the infinite plane, where it applies, is rendered and can be visualized in the *Graphics* window. This rendering can be turned off in the **Physics Symbol** section by clearing the checkbox **Show physics symbols**.

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

The exterior field pressure is evaluated using the exterior field operator (the name is defined in the **Exterior field variable name** input field, the default is `pext`).

To evaluate the pressure in a point `(x0,y0,z0)`, simply write `pext(x0,y0,z0)`. To evaluate the sound pressure level in the same point, it is advantageous to use the `subst()` operator and write, for example, `subst(acpr.efc1.Lp_pext,x,x0,y,y0,z,z0)`.

An example of this is given in the *Loudspeaker Driver — Frequency-Domain Analysis* tutorial model from the Acoustics Application Libraries.

If the `subst()` operator and the exterior field operator `pext()` are used together, you should never use the frame coordinates as arguments. Instead use dummy variables and write, for example:

```
subst(pext(a,b,c),a,0[m],b,0[m],c,1[m])
```

Never use `x`, `y`, and `z` in this type of expressions.

Predefined variables exist for the radiated power associated with an **Exterior Field Calculation** feature. The variable is `acpr.efc1.P_rad` (for the feature with tag `efc1`, and so on.) and can readily be plotted in results. Note that the variable includes scaling to include symmetries used in the feature.

In 2D axisymmetry, the evaluation of the exterior-field integral automatically includes the **Azimuthal mode number** m from the Pressure Acoustics Equation Settings. For postprocessing purposes, for example in a **Radiation Pattern** plot, it is necessary to include the azimuthal component explicitly by writing: `pext(r,z)*exp(-i*acpr.m*phi)`.

ADVANCED SETTINGS

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

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

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

In these cases, use a single boundary layer mesh on the inside of the outer boundary or on the inside of the PML to enhance the precision of the exterior field calculation. Note that this layer is automatically added when using the Physics-Controlled Mesh for Pressure Acoustics.

The option **Reverse normal direction on interior boundaries** allows reversing the normal used in the Helmholtz–Kirchhoff integral. To get the correct phase the normal has to point inward. Typically, if the exterior field is calculated in an interior boundary to the physics (not a boundary next to a PML), the normals will point outward and the option should be used.

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

- *Acoustic Scattering off an Ellipsoid*. Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/acoustic_scattering**

Port

The **Port** boundary condition is used to excite and absorb acoustic waves that enter or leave waveguide structures, like a duct or channel, in an acoustic model. A given port condition supports one specific propagating mode. To provide the full acoustic description, combine several port conditions on the same boundary. Make sure that all propagating modes in the studied frequency range are included (all modes that have a cutoff frequency in the frequency range). By doing this, the combined port conditions provide a superior nonreflecting or radiation condition for waveguides to, for example, the Plane Wave Radiation condition or a perfectly matched layer (PML) configuration. The same port boundary condition feature should not be applied to several waveguide inlets/outlets. The port condition supports S-parameter (scattering parameter) calculation but it can also be used as a source to just excite a system. The **Port** boundary condition exists for 3D, 2D, and 2D axisymmetric models.

- *Duct with Right-Angled Bend*: Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/duct_right_angled_bend**
- *Absorptive Muffler*: Application Library path **Acoustics_Module/Automotive/absorptive_muffler**
- *Muffler with Perforates*: Application Library path **Acoustics_Module/Automotive/perforated_muffler**

On a given boundary, a combination of ports will define the total acoustic field (sum of incident and outgoing waves) as

$$p_t = \sum_{i \in \text{bnd}} (S_{ij} + A^{in} e^{i\varphi} \delta_{ij}) p_i$$

where the summation "i" is over all ports on the given boundary "bnd", S_{ij} is the scattering parameter, A^{in} is the amplitude of the incident field (at port "j"), φ is the phase of the incident field, and p_i is the mode shape of the i^{th} port. The mode shape p_i is normalized to have either a unit maximum amplitude or a unit power (see the normalization option in the Global Port Settings section). This means that the scattering parameter S_{ij} defines the amplitude of mode i when a system is exited at port j (with mode j). This corresponds to a multimode expansion of the solution on the given port. The scattering parameters are automatically calculated when an

acoustic model is set up with just one port exciting the system. To get the full scattering matrix The Port Sweep Functionality can be used.

Only one port should be excited at a time if the purpose is to compute S-parameters. The S-parameters are defined as `acpr.S11`, `acpr.S21`, and so on, and can be used in postprocessing.

An evaluation group for the **Scattering Coefficient** exists as a **Predefined Plot**. When evaluating, change the **Data Series Operation** for the port sweep parameter (`PortName`) to **Sum** to get the proper data formatting.

In other cases, having more than one port exciting the system might be wanted, but the S-parameter variables cannot be correctly computed. So when several ports are excited, the S-parameter output is turned off.

If only two ports are added to the system and a port sweep is performed, the system transfer matrix and system impedance matrices are also computed.

PORT PROPERTIES

Enter a unique **Port name**. Only nonnegative integer numbers can be used as **Port name** as it is used to define the elements of the S-parameter matrix. The numeric port names are also required for port sweep functionality. The port name is automatically incremented by one every time a port condition is added.

Select a **Type of port**: **User defined** (the default), **Numeric**, **Circular**, **Annular**, **Rectangular**, **Slit**, or **User defined (nondispersive)**. Depending on the selection, different options appear in the **Port Mode Settings** section (see below). Use the **Circular**, **Annular**, and **Rectangular** for ports with the given cross section in 3D, the **Circular** and **Annular** options in 2D Axisymmetric, and the **Slit** option in 2D. If the port has a different cross section, then either use the **User defined**, the **Numeric**, or the **User defined (nondispersive)** port options. The **User defined** or the **Numeric** options can be used in cases where, for example, the waveguide is lined (an impedance is applied at the outer boundary).

PORT MODE SETTINGS

Depending on the option selected for the **Type of port** (see above):

- For **User defined** enter user defined expressions for the **Mode shape** p_{n} and the **Mode wave number** k_{n} (SI unit: rad/m). The modes shape will automatically be scaled before it is used in the port condition. The normalized mode shape can be visualized

by plotting `acpr.port1.pn` (here for Port 1, and so on). Use the user defined option to enter a known analytical expression or to use the solution from The Pressure Acoustics, Boundary Mode Interface. The solutions from the boundary mode analysis can be referenced using the `withsol()` operator.

The **User defined** option can be used to easily define a plane-wave mode. Simply enter `1` for the mode shape and `acpr.k` as the mode wave number.

- The **Numeric** port option is used for waveguide cross sections that are neither circular nor rectangular. In this case, a boundary mode problem is solved on the port face to compute the desired propagating mode. This option requires the use of a **Boundary Mode Analysis** step in the study. It should be placed before the **Frequency Domain** step. In the study, add one **Boundary Mode Analysis** step for each **Numeric** port and make sure to reference the proper **Port name** in the study step.

  Select the **Mode wave number from** option to decide how the mode wave number k_n is determined:

  - In general, for a model with losses, use the default **Out-of-plane wave number** option; then the wave number is taken from the **Boundary Mode Analysis** step. In this case, it is not possible to perform a frequency sweep in the **Frequency Domain** study step. Only one frequency can be used and it should correspond to the **Mode analysis frequency** entered in the **Boundary Mode Analysis** step(s). One option is to add a **Parametric Sweep** and define a parameter for the frequency used in both the steps. In this case, care should be taken when setting up the search criteria in the mode analysis.
  - For a model without any losses (nondispersive), select the **Computed lossless mode cutoff frequency** option. In this case, a frequency sweep is possible. The **Boundary Mode Analysis** should only be carried out at the highest frequency in the sweep. The wave number is computed analytically for all the other frequencies.

When the **Numeric** port option is used and the boundary mode analysis is run, the boundary conditions from the Pressure Acoustics model are automatically inherited in the boundary problem. For this automatic procedure, there is only support for the **Sound Hard Boundary (Wall)**, **Symmetry**, **Pressure**, and **Sound Soft** conditions. If you need a more complex behavior, use the Pressure Acoustics, Boundary Mode interface in combination with the **User defined** port type.

When running a frequency domain sweep, in a model that uses either the **Circular**, the **Annular**, or the **Rectangular** port options, you can get a solver warning: `New constraint force nodes detected: These are not stored`.

This warning can be disregarded. The reason for the warning is that certain constraints, in underlying equations, only are activated when the frequency is above the mode cutoff frequency.

- The **Circular** port option is used for a port with a circular cross section. Enter the **Mode number, azimuthal** m (in 3D) and the **Mode number, radial** n to define the mode captured by the port. In 3D, also right-click the **Port** condition to add the Port Reference Axis when the **Circular** port type is selected. In 3D, also select the **Azimuthal angle dependency** as **Sine** (the default) or **Cosine**. This option is used to add the two orthogonal modes that exist for a given azimuthal mode number, add a new Port feature for each variant. The cutoff frequency of the mode can be evaluated in Results using the variable `acpr.port1.fc` (here for Port 1, and so on).

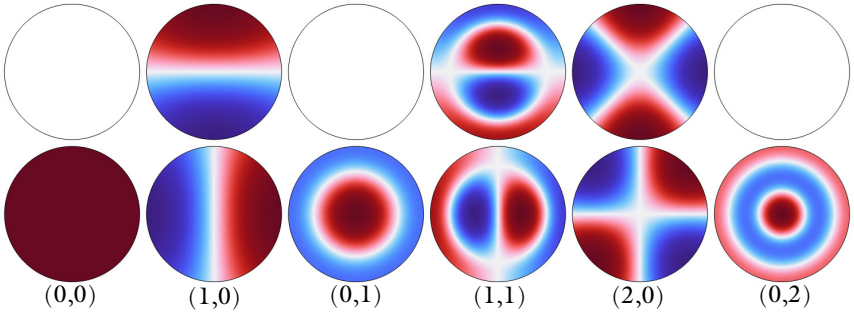


Figure 2-1: The first 6 modes (m,n) of a waveguide with circular cross section. Including the sine and cosine azimuthal angle dependencies (top/bottom rows, when different).

- The **Annular** port option is used for a port with an annular cross section. Enter the **Mode number, azimuthal** m (in 3D) and the **Mode number, radial** n to define the mode captured by the port. In 3D, also right-click the **Port** condition to add the Port Reference Axis when the **Annular** port type is selected. In 3D, also select the **Azimuthal angle dependency** as **Sine** (the default) or **Cosine**. This option is used to add the two orthogonal modes that exist for a given azimuthal mode number; add a new Port feature for each variant. Also enter the **Tolerance** (default value `1e-8`) used when computing the shape of the analytical modes (used for solving the associated Bessel-like equation). The default value is in general a good choice. The cutoff

frequency of the mode can be evaluated in Results using the variable `acpr.port1.fc` (here for Port 1, and so on).

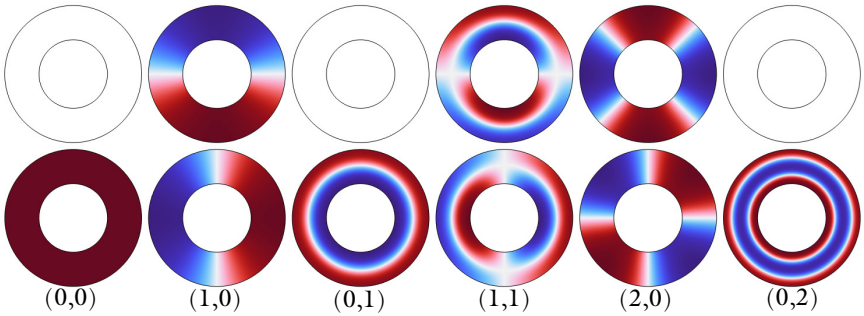


Figure 2-2: The first 6 modes (m,n) of a waveguide with annular cross section. Including the sine and cosine azimuthal angle dependencies (top/bottom rows, when different).

For the **Annular** option, if performing a parametric sweep over the port mode numbers m or n, only the **Parametric Sweep** can be used. The **Auxiliary sweep** option cannot be used, for this specific case.

The **Rectangular** port option, only available in 3D, is used for a port with a rectangular cross section. Enter the **Mode number, longest side** m and the **Mode number, shortest side** n to define the mode captured by the port.

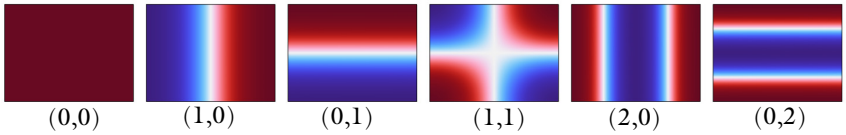


Figure 2-3: The first 6 modes (m,n) of a waveguide of rectangular cross section.

- The **Slit** port option is only valid in 2D geometries. Enter the **Mode number** m to define the mode captured by the port.

The cutoff frequency of the port modes can be evaluated in **Results** using the variable `acpr.port1.fc` (here for Port 1, and so on).

To ensure that the port conditions describe the acoustic field fully at an inlet/outlet it is important to include all propagating modes to form a full basis. For the **Circular** and **Annular** ports this also means including both the **Sine** an **Cosine** azimuthal angle dependencies. Add one port for each angle dependency, as soon as an azimuthal mode number m larger than 0 is included. Simply duplicate the port and switch the option.

- The **User defined (nondispersive)** option is useful for models without losses (nondispersive waveguides). In this case, the mode shape remains constant over a frequency sweep and only the mode wave number k_{n} changes. Enter an expression for the user-defined **Mode shape** p_{n}; then enter the **Mode wave number** k_0 and the **Mode analysis frequency** f_0 at which the mode wave number k_0 is defined. The wave number k_{n} for a given frequency is then automatically computed and used for setting up the port.

PORT GEOMETRY

The settings in this section ensure that the correct port area (including symmetries) is used for computing important quantities like power, volume flow, and transfer matrix elements, associated with the port

Select the **Port area** as **Use symmetries** (the default) or **Selected boundaries**.

- When **Use symmetries** is selected, symmetry conditions adjacent to the port will automatically be taken into account if the **Port area multiplication factor** is set to **Automatic** (the default); if set to **User defined**, enter the area multiplication factor A_{scale} manually.
- When **Selected boundaries** is selected, the port will have the area of the selected boundaries, without taking any symmetry conditions into account.

The **Port** condition can only be applied to connected boundaries and the boundaries should typically be coplanar. The port condition can in principle be applied to a curved continuous boundary, but care should be taken that the setup is mathematically and physically consistent.

The **Port** boundary conditions should in general be placed at least one waveguide diameter from other geometry features to ensure that only a pure fully developed propagating wave exists at the port. Note also that the port condition does not treat possible evanescent waves.

INCIDENT MODE SETTINGS

Activate if the given port is excited by an incident wave of the given mode shape. For the first **Port** condition added in a model, the **Incident wave excitation at this port** is set to **On**. For subsequent conditions added, the excitation is set to **Off** per default. If more than one port in a model is excited, the S-parameter calculation is not performed.

When the **Incident wave excitation at this port** is set to **On**, then select how to define the incident wave. Set **Define incident wave** to **Amplitude** (the default) or **Power**.

- For **Amplitude**, enter the amplitude A_p^{in} (SI unit: Pa) of the incident wave. This is in general defined as the maximum amplitude for a given mode shape.
- For **Power**, enter the power P^{in} (SI unit: W) of the incident wave. This is in general defined as the RMS power of the incident wave.
- For both options, enter the phase φ (SI unit: rad) of the incident wave. This phase contribution is multiplied with the amplitude defined by the above options. The **Amplitude** input can be a complex number.

Note that when the **Activate port sweep** option is selected at the physics level, the options in the **Incident Mode Settings** section are deactivated. This is because this option automatically sends in a mode of unit amplitude, sweeping through one port at a time.

All ports with incident wave excitation turned off have an arrow symbol, added in the Graphics window, that points outward. When incident wave excitation is turned on, the arrow symbol points inward.

If a waveguide is cut with one or several symmetry planes, care should be taken when setting up the port modes and when postprocessing.

For the **Circular** and **Rectangular** options make sure to only select modes that are actually symmetric according to the symmetry planes.

The port condition is in general not compatible with the Background Pressure Field domain feature. Combining the two will generate unphysical results if placed next to each other.

CONSTRAINT SETTINGS

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

THE PORT SWEEP FUNCTIONALITY

The port sweep functionality is used to reconstruct the full scattering matrix S_{ij} by automatically sweeping the port excitation through all the ports included in the model. When the port sweep is activated, the options in the **Incident Mode Settings** in the port conditions are deactivated and COMSOL controls which port that is excited with an incident mode.

The port sweep functionality is activated at the main physics interface level by selecting **Activate port sweep** in the Global Port Settings section. Enter the **Sweep parameter name**, the default is `PortName`. Create a parameter with the same name under **Global Definitions** > **Parameters 1**. This is the name of the parameter to be used in a parametric sweep, here it should represent the **Port name** integer values (defined when adding the port conditions). Add a parametric sweep study step and run the sweep over the `PortName` parameter with an integer number of values representing all the ports in the model. Once the model is solved the full scattering matrix can be evaluated using the defined global variables `acpr.S11`, `acpr.S21`, `acpr.S12`, and so on. The transmission loss (TL) between two given ports is also computed, for example, the variable for the TL loss from port 1 to 2 is given by `acpr.TL_12`.

If only two ports are added to the pressure acoustics model, COMSOL also automatically computes the transfer matrix (two-port representation) of the system (variables `acpr.T11`, `acpr.T12`, `acpr.T21`, `acpr.T22`) and the impedance matrix of

the system (`acpr.Z11`, `acpr.Z12`, `acpr.Z21`, `acpr.Z22`). These expressions are only true if plane wave modes are used.

- An evaluation group for the **Scattering Coefficient** exists as a **Predefined Plot**. The **Global Matrix Evaluation** is used to evaluate the full scattering matrix `acpr.S`. When evaluating, change the **Data Series Operation** for the port sweep parameter (typically `PortName`) to **Sum** to get the proper data formatting.
- An **Predefined Plot** evaluation group also exists for the **Transfer Matrix**. This plot evaluates the real and imaginary parts of the four matrix components separately. The output format can readily be exported to a file and used as interpolation data for subsequent use in other models, for example, in connection to a Lumped Port condition.

Port Reference Axis

The **Port Reference Axis** is necessary when the port type is set to **Circular** or **Annular** in the Port feature. Select two points in the geometry to define the azimuthal reference axis for the port mode orientation.

Lumped Port

The **Lumped Port** feature connects a waveguide or duct inlet/outlet to a lumped representation element. This can be an *Electrical Circuit*, a two-port network defined through a transfer matrix, or a lumped representation of a waveguide. Basically, it couples the end of a waveguide to an exterior system that has a given lumped representation. Several representations and sources exist to describe the lumped system as well as excite the system. When using the lumped port representation, it is assumed that only plane waves propagate in the acoustic waveguide.

PORT GEOMETRY

The settings in this section ensure that the correct port area (including symmetries) is used for computing important quantities like power, volume flow, and transfer matrix elements, associated with the port

Select the **Port area** as **Use symmetries** (the default) or **Selected boundaries**.

- When **Use symmetries** is selected, symmetry conditions adjacent to the port will automatically be taken into account if the **Port area multiplication factor** is set to

Automatic (the default); if set to **User defined,** enter the area multiplication factor A_{scale} manually.

- When **Selected boundaries** is selected, the port will have the area of the selected boundaries, without taking any symmetry conditions into account.

The **Lumped Port** condition can only be applied to connected boundaries and the boundaries should typically be coplanar. The port condition can in principle be applied to a curved continuous boundary, but care should be taken that the setup is mathematically and physically consistent.

The **Lumped Port** boundary conditions should in general be placed at least one waveguide diameter from other geometry features to ensure that only a pure fully developed propagating wave exists at the port. Note also that the port condition does not treat possible evanescent waves.

LUMPED PROPERTIES

Select the type of lumped representation connected to the port by selecting the **Connection type** as **Circuit**, **Two-port** (the default), or **Waveguide**. The selection made here influences the settings and options in the **Connection Type** section (see below).

- The **Circuit** option is used to connect the port boundary to an *Electrical Circuit* interface (this requires the AC/DC module). The connection is set up by adding the **External I vs. U** element in the circuit model and selecting the lumped port as the **Electric potential**, for example, **Voltage from lumped port (acpr/lport1)**. The feature automatically introduces a unit transformation from electric to acoustic units. The classical equivalence between the voltage V (SI units: V) and the pressure p (SI unit: Pa) as well as the current I (SI unit: A) and the volume velocity Q (SI unit: m^3/s) is made.
- For the **Two-port** option it is assumed that the port boundary is connected to a two-port system described through a transfer matrix $\mathbf{T}$. See the **Connection Type** section for details.
- For the **Waveguide** option it is assumed that the port boundary is extended with a waveguide of constant and identical cross section. The connection is equivalent to the two-port as the waveguide is defined through the appropriate transfer matrix $\mathbf{T}$, of a waveguide. See the **Connection Type** section for details.

For the **Two-port** and the **Waveguide** options also select the **Exterior condition** as **Circuit**, **Ideal source** (the default), **Impedance**, or **Port**. The selection made here influences what section is visible in the user interface. When **Port** is selected enter the **Port name**, similar to the Port condition. See below for a description of the options.

Two electroacoustic tutorial models that couple a circuit (lumped representation of a microspeaker) to an acoustic domain using the **Lumped Port** and the **Circuit** option are:

- *Lumped Receiver Connected to Test Setup with a 0.4-cc Coupler*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_receiver_04cc**
- *Lumped Receiver with Full Vibroacoustic Coupling*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_receiver_vibroacoustic**

CONNECTION TYPE

This section is visible if **Two-port** or **Waveguide** is selected as the **Connection type**.

- For **Two-port**, select the **Two-port representation** as **Acoustic** (the default) or **Electroacoustic**. The selection indicates if the transfer matrix is from acoustic to acoustic variables or from electric to acoustic variables. The first can be used to include a transfer matrix of an acoustic subsystem, for example, extracted using the Port condition. The latter should, for example, be used if the transfer matrix represents a transducer with electric inputs and acoustic outputs.

  Enter expressions, for example, calling interpolation functions that depend on frequency, for the elements of the transfer matrix T_{11}, T_{12}, T_{21}, and T_{22}.

  For the **Acoustic** option, you can choose to invert the transfer matrix (reverse its direction) by selecting **Invert transfer matrix direction**. This option will invert and correct the signs of the transfer matrix $\mathbf{T}$ behind the scene.
- For **Waveguide**, select the **Waveguide model** as **Automatic** (the default) or **User defined**. The **Automatic** option will use the cross-section data and fluid properties of the boundary for the model. If **User defined** is selected, enter values for the **Wave number**

k, the **Specific characteristic acoustic impedance** Z. For both options, enter the **Waveguide length** L (SI unit: m) modeled with the lumped representation.

The transfer matrix $\mathbf{T}$ couples the inlet (1) and outlet (2) pressure p and flow rate Q, defined in the feature, according to the following usual convention:

$$\begin{bmatrix} p_1 \\ Q_1 \end{bmatrix} = \begin{bmatrix} T_{11} & T_{12} \\ T_{21} & T_{22} \end{bmatrix} \begin{bmatrix} p_2 \\ Q_2 \end{bmatrix}$$

What is inlet and outlet depends on if the port excites the system (with a source) or it is passive (with an end impedance).

For the **Two-port** option, variables exist for evaluating the pressure and sound pressure level at both ends of the two-port element. The variables are `acpr.lport1.p1`, `acpr.lport1.p2`, `acpr.lport1.Lp1`, and `acpr.lport1.Lp2`, where the physics tag and feature tag should be selected adequately. The number 1 and 2 in the variable name, refers to the inlet and outlet of the element.

SOURCE SETTINGS

This section is visible if **Ideal source** is selected as the **Exterior condition**.

Select the **Input source** as **Average pressure** (the default) or **Volume flow**. Enter values for the average pressure $p_{\text{av,in}}$ or the volume flow Q_{in}. These represent ideal sources equivalent to an ideal voltage or ideal current source.

IMPEDANCE SETTINGS

This section is visible if **Impedance** is selected as the **Exterior condition**.

In this case, the lumped port is not active and an impedance condition can be applied to terminate the lumped representation. This could, for example, be the acoustic impedance of a microphone membrane. Select the **Impedance model** as **User defined** (the default), **RCL**, or **Waveguide end impedance**.

- For the **User defined** option select the input format as an **Acoustic impedance** Z_{ac} (SI unit: kg/m^4·s) or an **Impedance** Z (SI unit: Pa·s/m). If the electroacoustic representation is used for the two-port enter the **Electric impedance** Z (SI unit: Ω).

- The **RCL** and **Waveguide end impedance** options have the same options as the Impedance boundary condition.

INCIDENT MODE SETTINGS

This section is visible if **Port** is selected as the **Exterior condition**.

With this option the exterior behavior of the lumped system is like a Port condition with only plane wave modes propagating. The system can be excited if **Incident wave excitation at this port** is turned **On**. Enter the amplitude A^{in} and phase ϕ of the incident wave. The units depend on the representation used. For this option, the scattering parameters `acpr.S11`, `acpr.S21`, and so on, of the system are also computed automatically.

CIRCUIT

This section gives information about the state of coupling/connection to an Electrical Circuit interface. If the **Lumped Port** is not connected, the text **Connect to 'External I vs. U' in the Electrical Circuit interface** text will be displayed. Once connected, a reference with a tag to the associated **External I vs. U** node, in the Electrical Circuit interface, is displayed.

Lumped Speaker Boundary

The **Lumped Speaker Boundary** condition is used to model a loudspeaker (dust cap, cone, surround assembly, and motor) or another transducer using a lumped representation with a coupling to an **Electrical Circuit** interface. The condition is applied at the location of the speaker diaphragm (on a surface with the driver equivalent area). The mechanical and electric properties of the speaker can, for example, be described through the classical Thiele-Small parameters and associated lumped circuit. In general, any circuit representation can be set up in the **Electrical Circuit** interface and used.

The properties of a back volume can also be modeled using a lumped representation through either a simple equivalent acoustic compliance, a more general RCL circuit representation, or a user-defined impedance. On the other hand, if the air domains on both sides of the speaker are modeled explicitly (included in the model) then there are two possible setups:

1. If the speaker diaphragm is represented by an interior boundary (a "shell" with air domains on both sides), then use the Interior Lumped Speaker Boundary condition.

2 If the speaker diaphragm is represented by a domain (a “solid” with air domains on both sides), then use two **Lumped Speaker Boundary** features, one applied to the upper side and one to the lower side. Make sure the speaker axis is the same on both sides and that the back volume correction is disabled (the **No correction** option). Finally, in the **Electrical Circuit,** add two **External I vs. U** nodes in series, when setting up the coupling.

Note that the lumped representation of a speaker is an approximation that assumes piston motion. So in effect, it is a low frequency approximation that does, for example, not model breakup of the membrane. The method is efficient to get fast results in early development phases, for testing several driver configurations, or to use as a validation guide of full multiphysics driver models.

The approach is typically also valid in a larger frequency range when modeling tweeters and especially microtransducers. For the latter application, the Lumped Speaker Boundary also exists for The Thermoviscous Acoustics, Frequency Domain Interface. For time domain simulations, see the Lumped Speaker Boundary for the The Pressure Acoustics, Transient Interface.

SPEAKER GEOMETRY

Define how the **Speaker area** (projected) of the speaker is computed by selecting either **Selected boundaries** (the default) if all boundaries are present, or **Use symmetries** if the speaker surface is only partially represented.

When **Use symmetries** is selected, select the **Speaker area multiplication factor** A_{scale} as **Automatic** (the default, the model is analyzed for the presence of symmetry conditions); or **User defined** and enter a value (default is 1). The settings are required in order to compute the effective area of the speaker which is used to compute the acoustic load and radiated power.

Define the **Speaker axis direction** $\mathbf{e}_{\text{ax}}$ by selecting **Automatic** (the default) or **User defined**. For the automatic option, the axis is computed as the average of the surface normals; this option is valid if a full speaker surface is selected. The **User defined** option should be used if the speaker surface is only partially represented and is the only option when **Use symmetries** is selected.

BACK VOLUME CORRECTION

The **Lumped Speaker Boundary** does not model the acoustics on the back side of the speaker cone explicitly. The properties can be entered either through a volume giving a simple compliance effect, through an RCL circuit, or through a user defined acoustic impedance of the back volume.

Select the **Acoustic impedance of the back volume** as **Volume compliance** (the default), **RCL**, **User defined**, or **No correction**.

- For **Volume compliance**, enter the **Volume** V_{back} (SI unit: m^3), this results in an equivalent acoustic compliance $C_{ac} = V_{back}/(\rho c^2)$.
- For **RCL**, the back volume impedance is represented by a serial RCL circuit. Enter the **Equivalent acoustic resistance** R_{ac} (SI unit: $kg/(m^4 \cdot s)$), the **Equivalent acoustic compliance** C_{ac} (SI unit: $(m^4 \cdot s^2)/kg$), and the **Equivalent acoustic inertance** L_{ac} (SI unit: kg/m^4).
- For **User defined**, enter an **Acoustic impedance** Z_{ac} (SI unit: $kg/(m^4 \cdot s)$)
- For **No correction**, the correction impedance is 0.

CIRCUIT

This section gives information about the state of coupling/connection to an Electrical Circuit interface. If the **Interior Lumped Speaker Boundary** is not connected, the text **Connect to 'External I vs. U' in the Electrical Circuit interface** text will be displayed. Once connected, a reference with a tag to the associated **External I vs. U** node, in the Electrical Circuit interface, is displayed.

Thermoviscous Boundary Layer Impedance

The **Thermoviscous Boundary Layer Impedance** condition adds the losses due to thermal and viscous dissipation in the acoustic boundary layers at a wall. The condition is sometimes known simply as the BLI model. The losses are included in a locally homogenized manner, where the losses are integrated through the boundary layers analytically. The condition is applicable in cases where boundary layers are not overlapping. That is, it is not applicable in a very narrow waveguide (with dimensions comparable to the boundary layer thickness) or on very curved boundaries. Other than that, there are no restrictions on the shape of the geometry. This is in contrast to the Narrow Region Acoustics feature which is applicable only in waveguides of constant cross section, but also applicable for all frequencies, that is, also the very narrow case where boundary layers are overlapping. The thickness of the viscous and thermal boundary layers is given by

$$\delta_v = \sqrt{\frac{2\mu}{\omega\rho}} \qquad \delta_{th} = \sqrt{\frac{2k}{\omega\rho C_p}}$$

where ω is the angular frequency, μ the dynamic viscosity, ρ the density, $\mathbf{k}$ the coefficient of thermal conductivity, and C_p the (specific) heat capacity at constant pressure.

The **Thermoviscous Boundary Layer Impedance** condition adds an impedance-like boundary condition by defining the inward normal velocity $-\mathbf{n} \cdot \mathbf{v}$ at the boundary in terms of the pressure and its tangential derivatives:

$$-\mathbf{n} \cdot \mathbf{v} = -i\omega\left(T_{\mathrm{bnd}} - \frac{\alpha_p T}{\rho C_p} p_t\right)\frac{\delta_{\mathrm{th}}\alpha_p}{1+i} - v_n + \frac{\delta_v}{1+i}\left(\nabla_{||} \cdot \mathbf{v}_{||}^0 + \frac{1}{i\omega\rho}\Delta_{||} p_t\right)$$

where T_{bnd} is a possible boundary temperature variation source, v_n is a possible normal velocity source, and $\mathbf{v}_{||}^0$ is a possible tangential velocity source (normal and tangential components are computed from a velocity vector). The expression implemented is a generalization of the expression presented in Ref. 54 including boundary temperature and velocity sources. Velocity sources are presented in Ref. 55 but without including the thermal effects. Thermal effects (without sources) are included in Ref. 56. The general idea of the boundary layer impedance formulation can be found in Pierce, Ref. 5 (see equation 10-4.12 in section 10-4 about the Acoustic Boundary-Layer Theory).

The thermoviscous boundary layer impedance is used in the tutorial *Piezoelectric MEMS Speaker*. Application Library path **Acoustics_Module/Electroacoustic_Transducers/piezo_mems_speaker**

When coupling a vibrating structure to an acoustic domain with the Acoustic–Structure Boundary, the Thermoviscous Boundary Layer Impedance (subfeature) can be added to the multiphysics coupling.

MODEL INPUT

Enter a value for the (equilibrium or boundary) **Temperature** T (SI unit: K). This field is always necessary as the temperature enters the expression for the boundary condition. This corresponds to the equilibrium temperature T_0 in the thermoviscous interfaces.

COORDINATE SYSTEM SELECTION

Select a **Coordinate system**: **Global coordinate system** (default) or **Boundary boundary System 1 (sys1)**. If more coordinate systems are present in the model, they will also show in the list. The option is used for entering the velocity vector.

MECHANICAL CONDITION

Select the **Mechanical condition** that applies for the wall/boundary: **No slip** (default), **Slip**, **Velocity**, or **Normal velocity (slip)**. These options give most of the relevant mechanical conditions that can be set up using The Thermoviscous Acoustics, Frequency Domain Interface.

- **No slip**, default condition at a wall that is not moving.
- **Slip**, condition at a wall that is not moving and where the viscous boundary layer effects are disregarded.
- **Velocity**, to model a moving/vibrating wall where the viscous boundary layer effects are taken into account. For **User defined** (the default), enter an expression for the wall velocity vector $\mathbf{v}_0$ (SI unit: m/s). Note that a pure tangential wall movement will generate sound when viscous effects are included. This is not the case in the lossless acoustics.

  For coupling the acoustic domain to a vibrating structure, including the boundary layer effects, the simplest is to add the Thermoviscous Boundary Layer Impedance (subfeature) to the Acoustic–Structure Boundary multiphysics coupling. For a manual coupling, select the announced solid velocity from the list, for example, **Velocity (solid/lemm1)**. The coupling back to the solid can be achieved by applying a **Boundary Load** to the structure and selecting the announced **Acoustic load per unit area** (this is a traction force).
- **Normal velocity (slip)**, to model a moving/vibrating wall where viscous boundary layer effects are disregarded, only the normal component of the wall movement will contribute to the sound generation.

THERMAL CONDITION

Select the **Thermal condition** that applies for the wall/boundary: **Isothermal** (default), **Adiabatic**, **Temperature variation**, or **Thermally Conductive Wall**. These options give most of the relevant thermal conditions that can be set up using The Thermoviscous Acoustics, Frequency Domain Interface; for detailed thermal conditions at walls

consider The Thermoviscous Acoustic–Thermoelasticity Interaction Multiphysics Interfaces.

- **Isothermal**, default condition at a solid wall. Since the thermal conductivity of solids are orders of magnitude larger than fluids the isothermal condition for the fluid is a good approximation in most cases.
- **Adiabatic**, condition at a wall where the thermal boundary effects are disregarded.
- **Temperature variation**, applies a fluctuating temperature condition at the boundary.
- For **Thermally Conductive Wall,** a new section **Wall Properties** (see below) appears.

WALL PROPERTIES

This section is visible when **Thermally Conductive Wall** option is selected. Select where the **Wall Material** is taken from (these are the properties of the solid wall adjacent to the fluid). The default is the **Boundary material** (remember to assign a material to the boundary if this option is used and use a different option than for the **Fluid material**) or select a specific material if desired.

Enter the necessary material properties for the wall **Density** ρ_w, **Thermal conductivity** k_w, and **Heat capacity at constant pressure** C_p^w. Per default they are taken **From material**. For **User defined** enter a value for the property.

Select the **Wall type** as **Infinite wall** or **Finite wall**. These options control the analytical temperature profile used to model the solid wall adjacent to the fluid domain (derived from work presented in Ref. 73). If **Finite wall** is selected enter the **Wall thickness** d (SI unit: m) and select the **Backside wall condition** as **Isothermal** or **Adiabatic**.

FLUID PROPERTIES

Select where the **Fluid material** is taken from. The default is the **Boundary material** (remember to assign a material to the boundary if this option is used) or select a specific material if desired.

Enter the necessary material properties for the **Speed of sound** c, **Density** ρ, **Heat capacity at constant pressure** C_p, **Ratio of specific heats** γ, **Thermal conductivity** k, and the **Dynamic viscosity** μ. Per default they are taken **From material**. For **User defined** enter a value for the property.

Transfer Matrix Coupling

The **Transfer Matrix Coupling** boundary feature is used to couple two boundaries (source and destination) using a transfer matrix representation. The transfer matrix is

a reduced or lumped representation of the physical domain connecting the two boundaries. Examples of applications range from modeling of diesel particulate filter (DPF) in an exhaust system using an analytical model to represent the filter, as a lumped representation of a solid structure, or simply as a lumped model for an acoustic subsystem. The feature has two options, a pointwise coupling and a lumped representation.

- The pointwise option **Transfer matrix, pointwise** couples the velocity and pressure independently at every point on the source/destination boundaries. This options requires the source and destination boundaries to be identical (they can be rotated). This option is typically used if the system that couples the source and destination can be assumed to only depend on the local acoustic properties, for example, parallel nearly independent waveguides in a DPF.
- The lumped option **Transfer matrix, lumped two-port** couples the average pressure and volume flow of the source/destination boundaries. This option is valid for plane wave propagation in the system and can be applied to boundaries that have different sized/shapes as it couples average surfaces values.

TRANSFER MATRIX COUPLING

Select the transfer matrix **Representation** as either **Transfer matrix, pointwise** (the default) or **Transfer matrix, lumped two-port**. Enter the components of the **Transfer matrix** T_{11}, T_{12}, T_{21}, and T_{22}. These are complex-valued expressions, constants, analytical expressions, or data from an interpolation function. In all cases, the values will typically depend on the studied frequency parameter `freq`.

When the **Transfer matrix, lumped two-port** option is chosen, also select the **Source and destination area** as **Use symmetries** (the default) or **Selected boundaries**.

- For **Use symmetries**, select the **Area multiplication factor** A_{scale} as **Automatic** (the default) or enter a value for A_{scale}. The automatic option will analyze the geometry and look for symmetry conditions to calculate A_{scale}. The area scaling is necessary for computing the correct volume flow at the boundaries.
- For the **Selected boundaries**, the area of the source and destination boundaries are taken as they are and no multiplication factor is applied.

Diesel Particulate Filter Analysis Using an Acoustic Transfer Matrix: Application Library path **Acoustics_Module/Automotive/diesel_particulate_filter_transfer_matrix**

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Mapping Between Source and Destination** section, see Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Plane Wave Radiation

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

The *plane wave* type is suitable for modeling open boundaries and ports (in the plane-wave regime only). The condition can be used for waveguide structures as long as they are only interesting in the plane-wave region, but in general the Port boundary condition should be used. When using the radiation condition on an open far-field boundary, it is recommended to construct the boundary such that the incidence angle is near to normal. This of course requires *a priori* knowledge of the problem and the solution. See the theory section Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions for details about the equations and the formulation of this nonreflecting boundary condition.

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

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

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

Spherical Wave Radiation

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

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

SPHERICAL WAVE RADIATION

Enter coordinates for the **Radiating field source location** $\mathbf{r}_0$ (SI unit: m) for 3D and 2D models. In a 2D axisymmetric model, enter the **Radiating field source location, z coordinate** z_0 (SI unit: m).

Cylindrical Wave Radiation

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

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

CYLINDRICAL WAVE RADIATION

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

Acoustic Cloaking. Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/acoustic_cloaking**

Incident Pressure Field

The **Incident Pressure Field** node is a subnode to most nonreflecting boundary conditions (plane, cylindrical, spherical wave radiation, and matched boundary). From the **Physics** toolbar, add to Matched Boundary, Plane Wave Radiation, Spherical Wave Radiation, or Cylindrical Wave Radiation nodes. In the frequency domain, four options exist for the **Pressure field type**: **Plane wave**, **Cylindrical wave**, **Spherical wave**, and **User defined**.

To set up and calculate the intensity variables for both the total, incident, and reflected acoustic fields at the boundary select **Calculate incident and reflected field intensity** at the bottom of the settings window. This requires and additional input of the **Density** ρ (SI unit: Pa) for the defined incident pressure field. The variables are available in postprocessing the results.

The **Incident Pressure Field** has built-in functionality to perform a plane wave expansion. This allows modeling scattering problems involving 2D axisymmetry using a 2D axisymmetric, model even though a plane wave is not in general axisymmetric.

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

When a perfectly matched layer (PML) is present in the model, do not apply incident fields on its outer boundaries. The PML is applied to absorb waves that move out of the computational domain. Defining an incident field on its boundary will lead to unphysical results.

INCIDENT PRESSURE FIELD

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

Plane Wave

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

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

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

For **Plane Wave**, enter a **Pressure amplitude** p_0 (SI unit: Pa), the **Speed of Sound** c (SI unit: m/s) of the medium defining the incident wave, the **Wave direction** $\mathbf{e}_k$ (SI unit: m), and a **Phase** ϕ (SI unit: rad).

In 2D axisymmetric models, the incident fields take a slightly different form due to the geometrical restrictions. Only the z-component for the **Wave direction** $\mathbf{e}_k$ can be entered. Per default, the wave can only travel in the axial direction, since this is the only axisymmetric form of a plane wave.

By selecting **Enable plane wave expansion**, the plane wave is expanded in its cylindrical harmonics and a general **Wave direction** $\mathbf{e}_k$ can be selected. This sets up a wave of the form:

$$p_i = e^{i\phi} 2^{\text{sign}(m)} i^m J_m\left(-k_s \frac{(\mathbf{x}\cdot(\mathbf{e}_k\cdot\mathbf{e}_r)\mathbf{e}_r)}{|\mathbf{e}_k|}\right)\exp\left(-ik_s \frac{(\mathbf{x}\cdot(\mathbf{e}_k\cdot\mathbf{e}_r)\mathbf{e}_r)}{|\mathbf{e}_k|}\right)$$

where m is the **Azimuthal mode number** specified in the Pressure Acoustics Equation Settings. To expand the solution, it is necessary to run a **Parametric Sweep** over the mode number from 0 up to the desired resolution.

Cylindrical Wave

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

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

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

For **Cylindrical wave**, enter a **Pressure amplitude at reference distance** p_0 (SI unit: Pa), the **Speed of sound** c (SI unit: m/s) of the medium defining the incident wave, a **Source location** $\mathbf{x}_0$ (SI unit: m), a **Source axis** $\mathbf{e}_{sa}$ (the vector is automatically normalized), and a **Phase** ϕ (SI unit: rad).

In 2D axisymmetric models, the incident fields take a slightly different form due to the geometrical restrictions. No source location nor axis is needed, as the source is always located on the z-axis.

Spherical Wave

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

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

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

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

In 2D axisymmetric models, the incident fields take a slightly different form due to the geometrical restrictions. Only enter the z-coordinate for the **Source location** z_0 (SI unit: m) since the source is always located on the z-axis in a 2D axisymmetric model.

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

User Defined

If **User defined** is selected, enter the expression for the **Incident pressure field** p_i (SI unit: Pa) as a function of space.

Perfectly Matched Boundary

The **Perfectly Matched Boundary** is effectively a perfectly matched layer (PML) that is applied to the open boundary without the need to define a domain (a layer in the geometry). The condition automatically applies the PML formulation using the extra dimension functionality of COMSOL Multiphysics.

The **Perfectly Matched Boundary** condition only exists for the Pressure Acoustics, Frequency Domain interface. For other interfaces use the classical definition and setup for Perfectly Matched Layers (PMLs).

GEOMETRY

Select the **Attenuation direction** as **Radial** (the default), **Normal**, or **Custom**. For the **Radial** option, enter the **Center coordinate** $\mathbf{r}_0$, that is a coordinate that approximately gives the location of the radiation source. The **Radial** option works well for most open problems and it will be applicable for most geometries of the radiation boundary. The **Normal** option will apply the attenuation in the normal direction to the boundary where the condition is applied. In this case, the shape of the radiating boundary is important. For the **Custom** option, enter the **Custom attenuation direction** in the edit fields.

SCALING AND MESH

Select the **Coordinate stretching type** as **Logarithmic**, **Polynomial** (the default), **Rational**, or **User defined**.

- For the **Polynomial** and **Rational** options, enter the **PML scaling factor** and the **PML scaling curvature parameter**. The default values depend on the selected **Coordinate stretching type**. The settings correspond to the settings entered for the classical Perfectly Matched Layers (PMLs).
- For the **Logarithmic** option, enter the **PML scaling factor** and the **Real term scaling factor**. The logarithmic scaling applies different scaling to the real and imaginary part of the coordinate transform. The imaginary part follows $\log(1 - \xi)$ where ξ is the normalized distance in the layer (goes from 0 to 1); this leads to a rational scaling of the Jacobian (see Ref. 74). The real part follows a linear scaling ξ. The **PML scaling factor** applies to the full scaling function while the **Real term scaling factor** only applies to the real scaling functions.
- When **User defined** is selected, choose the functions to be used for the real and imaginary part of the stretching functions, respectively. These should be **Functions** defined under the **Definitions** node.

Enter the **Number of mesh points**, the default depends on the selected **Coordinate stretching type**. This represents the mesh resolution in the PML along the extra dimension.

Select **Use uniform thickness** (not selected per default) to enforce an equivalent uniform thickness in the attenuation direction. This option is only available when the **Attenuation direction** is **Radial**. If selected, a slight gain may be achieved in numerical precision, but the performance/convergence of iterative solvers may be compromised.

For spherical geometries this will make the attenuation behave as a PML with the same settings and a uniform mesh.

- *Piezoacoustic Transducer*. Application Library path **Acoustics_Module/Piezoelectric_Devices/piezoacoustic_transducer**
- *Acoustic Scattering off an Ellipsoid*. Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/acoustic_scattering**

Boundary conditions that intersect a **Perfectly Matched Boundary** are not continued/extended through the boundary. This setup is possible with a classical PML where, for example, an interior wall, a periodic, or an impedance condition can be defined inside the layer. This fact puts some restrictions on the general use of the **Perfectly Matched Boundary**.

Interior Sound Hard Boundary (Wall)

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

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

This is applied to the two sides of the boundary (the up and down sides). For zero dipole source $\mathbf{q}_d$ and constant fluid density ρ_c, this means that the normal derivative of the pressure is zero at the boundary. On an interior sound hard boundary, the pressure is not continuous but is treated as a slit feature.

Interior Normal Acceleration

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

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

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

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

INTERIOR NORMAL ACCELERATION

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

- For **Acceleration,** enter values for the components of the **Acceleration** $\mathbf{a}_0$ (SI unit: m/s^2).
- For **Normal acceleration,** enter the value of the **Normal acceleration** a_n (SI unit: m/s^2) in the normal direction. The normal is depicted as a red arrow in the graphics window on the selected boundary.

Interior Normal Velocity

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

INTERIOR NORMAL VELOCITY

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

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

Interior Normal Displacement

The **Interior Normal Displacement** node adds a normal displacement on an interior boundary and ensures that the pressure is noncontinuous. Add this node from the **Interior Conditions** submenu. The pressure has a slit condition on this boundary. The

condition is very similar to the Interior Normal Acceleration condition, simply having the acceleration given by $(i\omega)^2$ times the displacement.

INTERIOR NORMAL DISPLACEMENT

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

- For **Displacement** enter values for the components of the **Displacement** $\mathbf{d}_0$ (SI unit: $\mathrm{m/s^2}$).
- For **Normal displacement** enter the value of the **Normal displacement** d_n (SI unit: $\mathrm{m/s^2}$) in the normal direction. The normal is depicted as a red arrow in the graphics window on the selected boundary.

Interior Impedance/Pair Impedance

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

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_\mathrm{c}}(\nabla p_\mathrm{t}-\mathbf{q}_\mathrm{d})\right)_\mathrm{up} = (p_\mathrm{t,down}-p_\mathrm{t,up})\frac{-i\omega}{Z_\mathrm{n}}$$

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_\mathrm{c}}(\nabla p_\mathrm{t}-\mathbf{q}_\mathrm{d})\right)_\mathrm{down} = (p_\mathrm{t,down}-p_\mathrm{t,up})\frac{-i\omega}{Z_\mathrm{n}}$$

Z_n is the specific transfer impedance, which from a physical point, is the ratio between pressure drop and local normal particle velocity. On pair (interior) impedance conditions, the up and down, which refers to the up and down sides of an interior boundary, is replaced by a dst and src subscript in the equation display. These refer to the destination and source faces in the pair. In the pair feature, the normal on the left-hand side in the equation refers to the destination normal direction $\mathbf{n}_\mathrm{dst}$.

Additional information is found in Identity, Contact, and Sector Symmetry Pairs in the *COMSOL Multiphysics Reference Manual*.

INTERIOR IMPEDANCE/PAIR IMPEDANCE

Select the (interior) **Impedance model** as **User defined** (the default) or **Perforated plate**.

- For the **User defined** option, enter the value of the **Specific transfer impedance** Z_n (SI unit: Pa·s/m). The default is 0 Pa·s/m.
- For the **Perforated plate** option, see the options and details for the Interior Perforated Plate/Pair Perforated Plate feature. The two ways of setting up an interior perforated plate are identical.

When coupling a FEM and BEM based formulation with the Acoustic FEM–BEM Boundary, the Impedance (subfeature) can be added to the multiphysics coupling to model an interior impedance.

Interior Perforated Plate/Pair Perforated Plate

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

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

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

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

INTERIOR PERFORATED PLATE/PAIR PERFORATED PLATE

- **Hole diameter** d_h (SI unit: m). The default is 1 mm (10^{-3} m).
- **Plate thickness** t_p (SI unit: m). The default is 1.5 mm ($1.5{\cdot}10^{-3}$ m).
- **Area porosity** σ, that is, the fraction of the boundary surface area that consists of holes, a dimensionless number between 0 and 1. The default is 0.1, which means that 10% of the plate area consists of holes.

- **End correction** to the resistance δ_{resist} and the reactance δ_{reactt} (SI unit: m). The default built-in sets $\delta_{\text{resist}} = \delta_{\text{resist}} = 4d_{\text{h}}/3\pi$. Otherwise, select the user defined option.
- **Hole-hole interaction** f_{int}, a dimensionless function that accounts for the influence of the porosity on the end correction. The default is the built-in Fok function (using eight terms):

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

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

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

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

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

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

FLUID PROPERTIES

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

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

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

- **Ratio of specific heats** γ (SI unit: 1). Only for the **Thick plate** model.
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)). Only for the **Thick plate** model.
- **Thermal conductivity** k (SI unit: W/(m·K)). Only for the **Thick plate** model.

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

Interior Lumped Speaker Boundary

The **Interior Lumped Speaker Boundary** condition is used to model a loudspeaker (dust cap, cone, surround assembly, and motor) or another transducer using a lumped representation with a coupling to an **Electrical Circuit** interface. The condition is applied at the location of the speaker diaphragm (on an interior surface with the driver equivalent area). The mechanical and electric properties of the speaker can, for example, be described through the classical Thiele-Small parameters and associated lumped circuit. In general, any circuit representation can be set up in the **Electrical Circuit** interface and used. Add this node from the **Interior Conditions** submenu. The interior version of the Lumped Speaker Boundary includes the effects of the fluid domains on both sides of the speaker boundary explicitly.

Note that the lumped representation of a speaker is an approximation that assumes piston motion. So in effect, it is a low frequency approximation that does, for example, not model breakup of the membrane. The method is efficient to get fast results in early development phases, for testing several driver configurations, or to use as a validation guide of full multiphysics driver models.

The approach is typically also valid in a larger frequency range when modeling tweeters and especially microtransducers. For the latter application the Interior Lumped Speaker Boundary also exists for The Thermoviscous Acoustics, Frequency Domain

Interface. For time domain simulations see the Interior Lumped Speaker Boundary for the The Pressure Acoustics, Transient Interface.

SPEAKER GEOMETRY

Define how the **Speaker area** (projected) of the speaker is computed by selecting either **Selected boundaries** (the default) if all boundaries are present, or **Use symmetries** if the speaker surface is only partially represented.

When **Use symmetries** is selected, select the **Speaker area multiplication factor** A_{scale} as **Automatic** (the default, the model is analyzed for the presence of symmetry conditions); or **User defined** and enter a value (default is 1). The settings are required in order to compute the effective area of the speaker which is used to compute the acoustic load and radiated power.

Define the **Speaker axis direction** $\mathbf{e}_{\text{ax}}$ by selecting **Automatic** (the default) or **User defined**. For the automatic option, the axis is computed as the average of the surface normals; this option is valid if a full speaker surface is selected. The **User defined** option should be used if the speaker surface is only partially represented and is the only option when **Use symmetries** is selected.

For the **Automatic** option select **Reverse automatic speaker axis direction** if necessary. The axis orientation can be visualized by plotting the axis components `acpr.ilsb1.e_axx`, `acpr.ilsb1.e_axy`, and `acpr.ilsb1.e_axz`. (using the proper item and physics scope).

CIRCUIT

This section gives information about the state of coupling/connection to an Electrical Circuit interface. If the **Interior Lumped Speaker Boundary** is not connected, the text **Connect to 'External I vs. U' in the Electrical Circuit interface** text will be displayed. Once connected, a reference with a tag to the associated **External I vs. U** node, in the Electrical Circuit interface, is displayed.

- *Lumped Loudspeaker Driver*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver**
- *Headphone on an Artificial Ear*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/headphone_artificial_ear**

Continuity

Continuity is available as an option at interfaces between parts in an assembly, from the **Pairs** submenu. The condition gives continuity in total pressure and in the normal acceleration over the pair:

$$p_{\mathrm{t,src}} = p_{\mathrm{t,dst}}$$

$$-\mathbf{n}\cdot\left[-\left(\frac{1}{\rho_c}(\nabla p_{\mathrm{t}}-\mathbf{q}_{\mathrm{d}})\right)_{\mathrm{src}}-\left(-\frac{1}{\rho_c}(\nabla p_{\mathrm{t}}-\mathbf{q}_{\mathrm{d}})\right)_{\mathrm{dst}}\right]=0$$

Subscripts `src` and `dst` in the equation refer to the source and destination in the pair; that is, the two sides in the pair.

CONSTRAINT SETTINGS

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

Line Source

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

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

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

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

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

LINE SOURCE

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

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

Flow

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

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

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

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

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

Intensity

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

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

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

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

Power

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

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

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

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

Line Source on Axis

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

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

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

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

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

LINE SOURCE ON AXIS

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

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

Monopole Point Source

Use the **Monopole Point Source** node to add a monopole point source in 3D components on any point and in 2D axisymmetric components on points on the axis of symmetry (valid for 0 azimuthal mode number m). This is a source that is uniform and equally strong in all directions. A monopole represents a radially pulsating sphere in the limit where the radius tends to zero. Add this node from the **Points** submenu.

The monopole point source adds a point source term to the right-hand side of the governing Helmholtz equation such that:

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

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

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

MONOPOLE POINT SOURCE

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

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

Flow

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

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

Enter values or expressions for:

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

Intensity

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

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

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

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

Power

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

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

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

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

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

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

Dipole Point Source

Use the **Dipole Point Source** node to add a dipole point source. A dipole is mathematically a source that corresponds to two monopoles close to each other that are completely out of phase. Dipoles appear when there are fluctuating forces in the medium; for example, a small object that vibrates back and forth. A complex acoustic source may be expanded and approximated by a collection of point sources (Monopole Point Source, Dipole Point Source, and Quadrupole Point Source). Add this node from the **Points** submenu.

The dipole point source adds a point source term to the right-hand side of the governing Helmholtz equation such that:

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

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

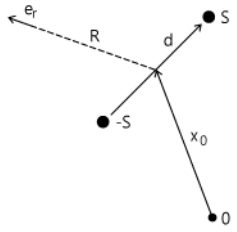


Figure 2-4: Schematic illustration of a dipole source.

DIPOLE POINT SOURCE

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

User Defined

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

Power

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

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

Enter the following:

- **Free space reference power (RMS)**, P_rms (SI unit: W). In a homogeneous medium, the specified power is radiated (the reference), but with other objects and boundaries present the actual power is different.

- The source **Phase** ϕ (SI unit: rad).
- **Dipole moment direction** $\mathbf{e}_D$. Defines the direction of the dipole moment vector. In 2D axisymmetric components the direction is aligned along the z-axis such that $\mathbf{e}_D = \mathbf{e}_z$.

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

Quadrupole Point Source

Use the **Quadrupole Point Source** node to enter a quadrupole point source. A quadrupole is mathematically a source that corresponds to two dipoles close to each other. A complex acoustic source may be expanded and approximated by a collection of point sources (Monopole Point Source, Dipole Point Source, and Quadrupole Point Source). Add this node from the **Points** submenu.

The quadrupole point source adds a point source term to the right-hand side of the governing Helmholtz equation such that:

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

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

QUADRUPOLE POINT SOURCE

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

User Defined

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

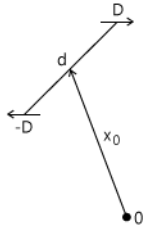


Figure 2-5: Schematic illustration of the user-defined quadrupole point source.

Power (Longitudinal)

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

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

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

Enter the following:

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

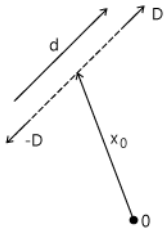


Figure 2-6: Schematic illustration of the longitudinal power quadrupole point source.

Power (Lateral)

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

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

$$Q^2 = \frac{15\rho_c c_c P_{rms}}{2\pi k^4}$$

Enter the following:

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

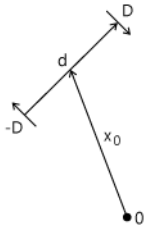


Figure 2-7: Schematic illustration of lateral quadrupole point source.

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

Point Sources (for 2D Components)

For 2D components, the monopole, dipole, and quadrupole point sources also exist. All the sources actually correspond to an infinite line source in the out-of-plane direction. The monopole is uniform and equally strong in all the in-plane directions —

an infinitely long pulsating cylinder. The dipole source represents an infinitely long cylinder moving back and forth. The quadrupole represents two dipoles close to each other. Add this node from the **Points** submenu.

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

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

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

MONOPOLE POINT SOURCE (2D COMPONENTS)

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

Flow

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

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

Enter values or expressions for:

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

Intensity

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

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

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

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

Power

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

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

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

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

DIPOLE POINT SOURCE (2D COMPONENTS)

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

QUADRUPOLE POINT SOURCE (2D COMPONENTS)

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

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

Circular Source (for 2D Axisymmetric Components)

Use the **Circular Source** node to add a source in a 2D axisymmetric component on points off the axis of symmetry. Such points correspond to circular sources or ring

sources. This type of source is, for example, used to mimic source terms from rotors. Add this node from the **Points** submenu.

The circular source adds a point source term to the right-hand side of the governing Helmholtz equation such that

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

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

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

CIRCULAR SOURCE

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

Flow

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

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

Enter values or expressions for:

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

Pressure (Point Condition)

Use the **Pressure** condition on points to define pressure monopole-like sources with a specific pressure. The condition is, for example, useful when setting up room acoustic

models where a prescribed pressure is needed (ideal pressure source). The condition acts as a constraint. Add this node from the **Points** submenu.

PRESSURE

Enter a value or expression for the **Pressure** p_0 (SI unit: Pa). In the time domain, the expression should depend on time.

AWE Expression

Use the **AWE Expression** feature to define the asymptotic waveform expansion (AWE) expression/metric used for the Adaptive Frequency Sweep study type. Add this node from the **Variables** submenu.

When the **AWE Expression** feature is added to a **Pressure Acoustics, Frequency Domain** model, the default **Physics controlled** option for the **AWE expression type** setting, in the **Adaptive Frequency Sweep** study step, will work, the expression will be defined by the physics.

ASYMPTOTIC WAVEFORM EVALUATION EXPRESSION

Select the AWE expression option as **User defined** (the default), **Point integration**, or select one of the predefined expression defined by the Port feature (if present in the model).

- For **User defined**, enter a global expression that defined the AWE expression/metric. This is an expression that represents the response of the system. This can, for example, be the sound pressure level (SPL) evaluated using an integration or average operator.
- For **Point integration**, select the points where the pressure is probed (in the **Point Selection** section). Select one or more points to automatically give the summed acoustic power metric $|p|^2$. It is important that the selected points probe the solution spatially, if it changes significantly.
- If a **Port** feature is present in the model, the list will be populated with a **Port asymptotic waveform evaluation expression** for each added port. In this case, the AWE expression/metric will be given by the scattering coefficient of the given port.

The Pressure Acoustics, Transient Interface

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

The physics interface can be used to model linear and nonlinear acoustics that can be well described by the scalar pressure variable. Domain conditions include the Nonlinear Acoustics (Westervelt) Contributions to include nonlinear effects, the Anisotropic Acoustics feature to model fluids with an effective anisotropic density, and the Background Pressure Field (for Transient Models) for defining a background acoustic field to model scattering problems or defining incident waves. User-defined sources can also be added via the Monopole Domain Source or the Dipole Domain Source. For open problems, Perfectly Matched Layers (PMLs) can be applied, also in the time domain for Pressure Acoustics, as efficient nonreflecting boundary conditions. The interface has built-in options that allow setting up frequency dependent Impedance conditions in the time domain, by using the very general and flexible General local reacting (rational approximation) impedance option.

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

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

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

SETTINGS

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

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

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

TYPICAL WAVE SPEED FOR PERFECTLY MATCHED LAYERS

Enter a value or expression for the **Typical wave speed for perfectly matched layers** c_{ref} (SI unit m/s). The default is `actd.c` and the value is automatically taken from the material model. If several materials or material models are used it is best practice to add one PML for each. This will ensure that the typical wavelength is continuous within each PML feature.

It is important to set this speed correctly as the performance of the Perfectly Matched Layers (PMLs) in the time domain depends only on the wave speed and not the wavelength. For further details see the Time Domain Perfectly Matched Layers section.

TRANSIENT SOLVER AND MESH SETTINGS

Enter the **Maximum frequency to resolve** in the model. The value is used for setting up the solver as well as the physics-controlled mesh. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. Select the **Time stepping** (method) as **Fixed (preferred)** the default and recommended or **Free**. The **Free** option is in general not recommended for wave problems. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study. It is also important to **Reset Solver to Defaults** if the Nonlinear Acoustics (Westervelt) Contributions feature is added as special handling of the nonlinear term is enabled. Remember also that in this case the **Maximum frequency to resolve** should be a value including a number of harmonics generated. For highly nonlinear problems set up with user-defined terms, manual tuning of the solver may be necessary. In

nonlinear models the maximum frequency to resolve should be selected based on the number of harmonics to be resolved.

	Further details on Time Stepping: Resolving the Waves in Time in the Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces) section.

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

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

	• *Transient Gaussian Explosion*: Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/gaussian_explosion** • *Gaussian Pulse Absorption by Perfectly Matched Layers: Pressure Acoustics, Transient*: Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/gaussian_pulse_perfectly_matched_layers** • *Nonlinear Acoustics — Modeling of the 1D Westervelt Equation*: Application Library path **Acoustics_Module/Nonlinear_Acoustics/nonlinear_acoustics_westervelt_1d**

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

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

- Nonlinear Acoustics (Westervelt) Contributions
- Background Pressure Field (for Transient Models)
- Exterior Field Calculation (for Transient Models)
- Incident Pressure Field (for Transient Models)
- Interior Impedance/Pair Impedance
- Interior Lumped Speaker Boundary
- Interior Normal Acceleration
- Interior Normal Velocity
- Interior Normal Displacement
- Impedance
- Lumped Speaker Boundary
- Normal Acceleration
- Normal Velocity
- Normal Displacement
- Transient Pressure Acoustics Model
- The Gaussian Pulse Source Type Settings

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

Transient Pressure Acoustics Model

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

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

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

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

TRANSIENT PRESSURE ACOUSTICS MODEL

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

- Linear Elastic Fluid Model
- Thermally Conducting and/or Viscous Fluid Model
- Ideal Gas Fluid Model

See Pressure Acoustics (frequency domain) for details of the fluid model equations. Nonlinear effects can be included using the Nonlinear Acoustics (Westervelt) Contributions node.

MODEL INPUTS

When **Ideal gas** is selected as the **Fluid model**, both the **Temperature** T and **Absolute pressure** p_A fields are always enabled. Select, **User defined** (the default), **Common model input**, or an input from another physics interface, if applicable.

In addition, the **Temperature** T and **Absolute pressure** p_A can be picked up from another physics interface where the fields have been calculated. For example, select a temperature field defined by a *Heat Transfer* interface or a *Nonisothermal Flow* interface (if any). Or, if applicable, select a pressure as defined by a *Fluid Flow* interface present in the model. For example, select **Absolute pressure (spf)** to use the absolute pressure defined by a *Laminar Flow* interface **spf**.

The *Nonisothermal Flow* interface requires the addition of the Heat Transfer Module or CFD Module.

Details about the Model Input and the Default Model Inputs are found in the Global and Local Definitions chapter of the *COMSOL Multiphysics Reference Manual.*

In transient models, when space dependent material data is given by an interpolation function, significant performance gains can be achieved with the following setup of the **Interpolation** function:

- Select **Use spatial coordinates as arguments**
- Set the **Frame** to **Mesh**
- Select **Reinterpolate interpolation data on computational mesh**

No arguments should be used when calling the interpolation function, for example, use `int` and not `int(x,y,z)`.

See the Interpolation section in the *COMSOL Multiphysics Reference Manual*.

Nonlinear Acoustics (Westervelt) Contributions

The propagation of finite-amplitude pressure waves cannot be described by the linear acoustic wave equation (small perturbation assumption) and requires solving the full nonlinear second-order wave equation. The latter allows a simplification when cumulative nonlinear effects dominate local nonlinear effects, for example, when the propagation distance is greater than the wavelength. The simplified equation is referred to as the Westervelt equation and reads

$$\frac{1}{\rho c^2}\frac{\partial^2 p_t}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d) - \frac{\delta}{\rho c^2}\frac{\partial \nabla p_t}{\partial t} \right) = \frac{\beta}{\rho^2 c^4}\frac{\partial^2 p_t^2}{\partial t^2} + Q_m ,$$

where δ is the diffusivity of sound and β is the coefficient of nonlinearity. In general, the linear assumption is only valid when

$$|p| << \rho c^2$$

which has a value of approximately $1.5 \cdot 10^5$ Pa for air and $2 \cdot 10^9$ Pa for water. So, whenever the acoustic pressure is say a factor 100 less than these values, the linear assumption holds. When this is not the case, the **Nonlinear Acoustics (Westervelt) Contributions** feature should be added.

The **Nonlinear Acoustics (Westervelt) Contributions** node adds the nonlinear contribution (first term on the right-hand side) to the **Transient Pressure Acoustics Model**. That is, the linear acoustic wave equation transforms to the nonlinear Westervelt equation. To add the dissipation term (last term on the left-hand side) select, for

example, the **General dissipation** as the **Fluid model** in the Transient Pressure Acoustics Model.

	The resulting equation is nonlinear and therefore does not allow the application of the superposition principle. In particular, the Background Pressure Field (for Transient Models) cannot be defined together with the **Nonlinear Acoustics (Westervelt) Contributions** feature.

	The presence of the **Nonlinear Acoustics (Westervelt) Contributions** node changes the default solver settings for the proper treatment of nonlinearities. It is therefore required to **Reset Solver to Defaults** whenever the feature is added to or removed from the model.

	The frequency spectrum of a nonlinear wave model contains not only the center frequency f_0 but also the harmonics $N \cdot f_0$, $N = 1, 2, 3, \ldots$ generated. The **Maximum frequency to resolve** in the Transient Solver and Mesh Settings section should be specified to resolve a desired number of harmonics to achieve a certain precision. This is especially important for shock waves, where the number of harmonics to resolve can exceed 10.

	• For the theory background of this nonlinear feature, see Pressure Acoustics, Transient Equations.

NONLINEAR ACOUSTICS (WESTERVELT)

Specify the coefficient of nonlinearity. The available options are **From parameter of nonlinearity** (the default), **From ratio of specific heats (for gases)**, and **User defined**.

- For the option **From parameter of nonlinearity** enter the value of the **Parameter of nonlinearity**, B/A. The coefficient of nonlinearity is defined as follows

$$\beta = 1 + \frac{B}{2A}$$

The parameter of nonlinearity quantifies the effect of nonlinearity on the local speed of sound in the fluid. It is expressed as $c = c_0 + (B/2A)u$ in the first-order terms, where u is the acoustic particle velocity.

- For the option **From ratio of specific heats (for gases)** enter the **Ratio of specific heats**, γ. This option is valid for perfect gases under isentropic conditions. Here, $B/A = \gamma - 1$, which yields

$$\beta = \frac{\gamma + 1}{2}$$

- Choose the **User defined** option to specify β explicitly.

SHOCK-CAPTURING STABILIZATION

To display this section, click the **Show More Options** button ( ) and select **Stabilization**. Select **Enable q-Laplacian relaxation** to add some artificial nonlinear damping to the model (it is turned off per default). This is typically only necessary in highly nonlinear models. The dissipative term of physical origin will often balance the shock formation (remember to define the appropriate **Fluid model** in the Transient Pressure Acoustics Model). The effective diffusivity is tuned with the stabilization by adding an extra term that reads

$$-\delta\kappa\left(1 - \left|\frac{\partial}{\partial t}\nabla p\right|^{q-1}\right)$$

The highest artificial damping is achieved where the acoustic pressure increases or decreases the most rapidly. It reaches its maximal values where the pressure endures discontinuities, that is, where shocks arise. Thus, this technique provides a shock-capturing stabilization.

Specify the **q-Laplacian exponent** q and the **q-Laplacian factor** κ to get the desirable amount of artificial damping. Note that the damping must not be too high nor too low. The particular values of q and κ depend on the material and the input signal frequency. The two parameters that control the stabilization require manual tuning. A suggested approach is to use a simple 1D model to tune the parameters based on fluid material properties and frequency content. Use, for example, the *Nonlinear Acoustics — Modeling of the 1D Westervelt Equation* model from the application library to do so.

	Nonlinear Acoustics — Modeling of the 1D Westervelt Equation: Application Library path **Acoustics_Module/Nonlinear_Acoustics/nonlinear_acoustics_westervelt_1d**

If the chosen fluid model has no dissipation (**Linear elastic** (the default) or **Ideal Gas**), a default sound dissipation $\delta = 2 \cdot 10^{-5}$ m^2/s will be used in the artificial damping term.

Background Pressure Field (for Transient Models)

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

$$p_t = p_b + p_s$$

This feature sets up the equations in a so-called scattered field formulation where the dependent variable is the scattered field $p = p_s$. In a model where the background pressure field is not defined on all acoustic domains (or it is different), continuity is automatically applied in the total field p_t on interior boundaries between domains.

BACKGROUND PRESSURE FIELD

Select the **Pressure field type** as **Plane wave (monochromatic)** (the default), **Modulated Gaussian pulse**, or **User defined**.

- For **Plane wave (monochromatic)** enter a value for the **Pressure amplitude** p_0 of the wave, select the **Speed of sound** as **User defined** (enter a value for c) or **From material**, enter a **Wave direction** vector $\mathbf{e}_k$, enter the **Signal frequency** f_0 of the wave, and a possible **Phase** ϕ.

  These settings defined a background pressure field p_b as a plane propagating wave of the type:

$$p_b(\mathbf{x}, t) = p_0 \sin\left(2\pi f_0\left(t - \frac{\mathbf{x} \cdot \mathbf{e}_k}{c|\mathbf{e}_k|}\right) + \phi\right)$$

- For **Modulated Gaussian pulse**, enter a value for the **Pressure amplitude** p_0 of the wave, select the **Speed of sound** as **User defined** (enter a value for c) or **From material**, enter a **Wave direction** vector $\mathbf{e}_k$ and enter the **Signal emitting plane coordinate** $\mathbf{x}_0$. The latter two options define the initial location of the pulse and the propagation direction. When modeling scattering problems, the initial location should be placed outside the computational domain. Next define the frequency content of the Gaussian pulse. Select the **Gaussian pulse bandwidth** as **Full width at half maximum, power** (the default), **Full width at half maximum, amplitude**, **Full duration at half maximum, power**, **Full duration at half maximum, amplitude**, or **Standard deviation**.

For the first two options, enter a frequency bandwidth Δf and for the latter a time window Δt. Finally, enter the carrier **Signal frequency (for time-dependent studies)** f_0.

These settings defined a background pressure field p_b as a propagating modulated Gaussian pulse of the type:

$$p_b(\mathbf{x}, t) = p_0 G(\mathbf{x}, t) \sin\left(2\pi f_0\left(t - \frac{\mathbf{x} \cdot \mathbf{e}_k}{c|\mathbf{e}_k|}\right)\right)$$

where the standard deviation σ of the Gaussian $G(\mathbf{x},t)$ depends on the bandwidth option selected. The modulated Gaussian pulse option only gives contributions when a time dependent model is solved. It has the advantage of having a defined frequency content and can be used for limited bandwidth simulations. In combination with the Exterior Field Calculation (for Transient Models) and an FFT study, the broadband scattering characteristics of an object can be analyzed.

- For **User defined**, enter an analytical expression that can be a function of both space and time or use a solution from a previous study.

ADVANCED SETTINGS

To display this section, when the **Plane wave (monochromatic)** option is selected, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

Click to select the **Use ramp function** option (selected per default). With this option turned on, a ramp is automatically added to smoothly increase the background pressure field amplitude over the first period $T = 1/f_0$. This option ensures numerical stability.

Impedance

The **Impedance** node adds an impedance boundary condition with the option to select between several built-in impedance models that allow modeling frequency dependent impedance conditions in the time domain. The impedance condition is a generalization of the sound-hard and sound-soft boundary conditions, and in the time domain reads:

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

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

the acoustic input impedance is the ratio between the local pressure and local normal particle velocity. The **Impedance** boundary condition is a good approximation of a locally reacting surface — a surface for which the normal velocity at any point depends only on the pressure at that exact point.

In many practical applications the surface impedance is known (measured or simulated) in term of its real and imaginary part as function of frequency. To capture this behavior in the time domain the **Impedance** condition has built-in functionality for setting up a General local reacting (rational approximation) impedance through a rational approximation. Fitting of the frequency domain data can be performed using the Partial Fraction Fit function.

IMPEDANCE

See the details for the Impedance condition in The Pressure Acoustics, Time Explicit Interface.

For details on how to best fit data, see Using the Partial Fraction Fit for Acoustic Data.

Incident Pressure Field (for Transient Models)

The **Incident Pressure Field** node is a subnode to all nonreflecting boundary conditions (plane, cylindrical, and spherical wave radiation). From the **Physics** toolbar, add to Plane Wave Radiation, Spherical Wave Radiation, or Cylindrical Wave Radiation nodes. Three options exist for transient models: **Plane wave (monochromatic)** (the default), **Modulated Gaussian pulse**, and **User defined**.

Plane Wave (Monochromatic)

The same options apply as for the incident fields as for the Background Pressure Field (for Transient Models) feature.

Modulated Gaussian Pulse

The same options apply as for the incident fields as for the Background Pressure Field (for Transient Models) feature.

User Defined

For **User defined** enter an expression that defined the incident wave. The incident pressure field needs to be defined as a traveling wave of the form

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

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

The Gaussian Pulse Source Type Settings

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

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

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

where S is the source strength.

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

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

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

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

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

Normal Acceleration

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

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

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

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

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

NORMAL ACCELERATION

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

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

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

Normal Velocity

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

NORMAL VELOCITY

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

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

Normal Displacement

The **Normal Displacement** adds an inward normal displacement $d_n(t)$ or specify the displacement $\mathbf{d}_0(t)$ of the boundary. The part in the normal direction is used to define the boundary condition. The condition is very similar to the Normal Acceleration condition having the acceleration given by the double time derivative of the displacement $a_n(t) = \partial^2 d_n(t)/\partial t^2$. This feature represents an external source term. It can also be used to manually couple acoustics with a structural analysis for modeling acoustic-structure interaction.

NORMAL DISPLACEMENT

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

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

Exterior Field Calculation (for Transient Models)

The **Exterior Field Calculation** feature adds the equations necessary to postprocess the exterior field to the computational domain for open problems. The feature is the same as the Exterior Field Calculation in The Pressure Acoustics, Frequency Domain Interface. This feature will only generate variables and results that can be used in postprocessing when the results of a time-dependent simulation have been transformed to the frequency domain using a **Time to Frequency FFT** study.

For more detailed information about the FFT study see the Time-to-Frequency FFT section in the *COMSOL Multiphysics Reference Manual*.

The feature only operates on the scattered field variables and is thus especially well suited for analyzing the results of a scattering problem. Such a model can be set up by using the Background Pressure Field (for Transient Models) feature combined with a radiation condition or PMLs. In this way, the radiation pattern of, for example, a loudspeaker modeled in the time can still be analyzed when an FFT of the time domain solution is performed.

Lumped Speaker Boundary

The **Lumped Speaker Boundary** condition is used to model a loudspeaker (dust cap, cone, surround assembly, and motor) or another transducer using a lumped representation with a coupling to an **Electrical Circuit** interface. The mechanical and electric properties of the speaker can, for example, be described through a Thiele-Small parameter representation and associated lumped parameters.

In the time domain, nonlinear effects can be included in the lumped model through parameters that depend on the axial position x or velocity v of the speaker. These are, for example, the typically measured $BL(x)$, $C_{MS}(x)$, or $R_{MS}(v)$. Predefined variables exist for the axial velocity `actd.lsb1.v_ax` and an ODE is solved for the associated axial position `actd.lsb1.x_ax` (use the appropriate physics and feature tag). These variables can be directly used in expressions in the electric components in the **Electrical Circuit** model as they are globally defined.

The properties of a back volume can also be modeled using a lumped representation through either a simple equivalent acoustic compliance or a more general RCL circuit representation. On the other hand, if the air domains on both sides of the speaker are modeled explicitly (included in the model) then there are two possible setups:

1. If the speaker diaphragm is represented by an interior boundary (a "shell" with air domains on both sides); then use the Interior Lumped Speaker Boundary condition.
2. If the speaker diaphragm is represented by a domain (a "solid" with air domains on both sides); then use two **Lumped Speaker Boundary** features, one applied to the upper side and one to the lower side. Make sure the speaker axis is the same on both sides and that the back volume correction is disabled (the **No correction** option). Finally, in the **Electrical Circuit** add two **External I vs. U** nodes in series, when setting up the coupling.

For microtransducer applications it may be necessary to use the Lumped Speaker Boundary or Interior Lumped Speaker Boundary of The Thermoviscous Acoustics, Transient Interface.

Lumped Loudspeaker Driver Transient Analysis with Nonlinear Large-Signal Parameters: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver_transient**

SPEAKER GEOMETRY

Define how the **Speaker area** (projected) of the speaker is computed by selecting either **Selected boundaries** (the default) if all boundaries are present, or **Use symmetries** if the speaker surface is only partially represented.

When **Use symmetries** is selected, select the **Speaker area multiplication factor** A_{scale} as **Automatic** (the default, the model is analyzed for the presence of symmetry conditions); or **User defined** and enter a value (default is 1). The settings are required in order to compute the effective area of the speaker which is used to compute the acoustic load and radiated power.

Define the **Speaker axis direction** $\mathbf{e}_{\text{ax}}$ by selecting **Automatic** (the default) or **User defined**. For the automatic option, the axis is computed as the average of the surface normals; this option is valid if a full speaker surface is selected. The **User defined** option should be used if the speaker surface is only partially represented and is the only option when **Use symmetries** is selected.

BACK VOLUME CORRECTION

The **Lumped Speaker Boundary** does not model the acoustics on the back side of the speaker cone explicitly. The properties can be entered either through a volume giving a simple compliance effect, through a simple RCL model, or through a user defined acoustic impedance of the back volume.

Select the **Acoustic impedance of the back volume** as **Volume compliance** (the default), **RCL**, or **No correction**.

- For the **Volume compliance** enter a **Volume** V_{back} (SI unit: m^3). The volume represents an acoustic compliance $V_{\text{back}}/(\rho c^2)$.

- For **RCL** enter the values of the equivalent acoustic resistance R_{ac} (SI unit: kg/(m^4·s)), compliance C_{ac} (SI unit: m^4·s^2/kg), and inertance L_{ac} (SI unit: kg/m^4).
- For **No correction**, the correction impedance is 0.

When the **RCL** and the **Volume compliance** options are selected an ODE is solved relating force acting on the cone F_c and axial velocity v_{ax}. For the general RCL option

$$\frac{dF_c}{dt} = -S_D^2\left[R_{ac}\frac{dv_{ax}}{dt} + \frac{1}{C_{ac}}v_{ax} + L_{ac}\frac{d^2v_{ax}}{dt^2}\right]$$

where S_D is the (projected) speaker area.

INITIAL VALUE

Enter the initial value of the force F_c and axial speaker position x_{ax}, used when solving the associated ODEs. The default value is 0 for both and is typically correct, except if the simulation is started, for example, with an initial excursion of the driver.

CIRCUIT

This section gives information about the state of coupling/connection to an Electrical Circuit interface. If the **Interior Lumped Speaker Boundary** is not connected, the text **Connect to 'External I vs. U' in the Electrical Circuit interface** text will be displayed. Once connected, a reference with a tag to the associated **External I vs. U** node, in the Electrical Circuit interface, is displayed.

Interior Normal Acceleration

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

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

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

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

INTERIOR NORMAL ACCELERATION

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

- For **Acceleration**, enter values for the components of the **Acceleration** $\mathbf{a}_0$ (SI unit: m/s^2).
- For **Normal acceleration**, enter the value of the **Normal acceleration** a_n (SI unit: m/s^2) in the normal direction. The normal is depicted as a red arrow in the graphics window on the selected boundary.

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

Interior Normal Velocity

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

INTERIOR NORMAL VELOCITY

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

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

Interior Normal Displacement

The **Interior Normal Displacement** node adds a normal displacement on an interior boundary and ensures that the pressure is noncontinuous. Add this node from the **Interior Conditions** submenu. The pressure has a slit condition on this boundary. The

condition is very similar to the Interior Normal Acceleration condition having the acceleration given by the double time derivative of the displacement.

INTERIOR NORMAL DISPLACEMENT

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

- For **Displacement** enter values for the components of the **Displacement** $\mathbf{d}_0$ (SI unit: $\mathrm{m/s^2}$).
- For **Normal displacement** enter the value of the **Normal displacement** d_n (SI unit: $\mathrm{m/s^2}$) in the normal direction. The normal is depicted as a red arrow in the graphics window on the selected boundary.

Interior Impedance/Pair Impedance

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

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

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

On pair (interior) impedance conditions, the up and down, which refers to the up and down sides of an interior boundary, is replaced by a dst and src subscript in the equation display. These refer to the destination and source faces in the pair. In the pair feature the normal on the left-hand side in the equation refers to the destination normal direction $\mathbf{n}_\mathrm{dst}$. Z_n is the specific transfer impedance, which from a physical point, is the ratio between pressure drop and local normal particle velocity.

INTERIOR IMPEDANCE/PAIR IMPEDANCE

Enter the value of the **Specific transfer impedance** Z_n (SI unit: Pa·s/m). The default is 0 Pa·s/m.

Interior Lumped Speaker Boundary

The **Interior Lumped Speaker Boundary** is used to model a loudspeaker (dust cap, cone, and surround assembly) or another transducer using a lumped representation with a coupling to an **Electrical Circuit** interface. The mechanical and electric properties of the speaker can, for example, be described through a Thiele-Small representation and associated lumped parameters. Add this node from the **Interior Conditions** submenu. The interior version of the Lumped Speaker Boundary includes the effects of the fluid domains on both sides of the speaker boundary explicitly. For microtransducer applications, it may be necessary to use the Lumped Speaker Boundary or Interior Lumped Speaker Boundary of The Thermoviscous Acoustics, Transient Interface.

In the time domain, nonlinear effects can be included in the lumped model through parameters that depend on the axial position x or velocity v of the speaker. These are, for example, the typically measured $\mathrm{BL}(x)$, $\mathrm{C_{MS}}(x)$, or $\mathrm{R_{MS}}(v)$. Predefined variables exist for the axial velocity `actd.ilsb1.v_ax` and an ODE is solved for the associated axial position `actd.ilsb1.x_ax` (use the appropriate physics and feature tag). These variables can be directly used in expressions in the electric components in the **Electrical Circuit** model as they are globally defined.

Lumped Loudspeaker Driver Transient Analysis with Nonlinear Large-Signal Parameters: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver_transient**

SPEAKER GEOMETRY

Define how the **Speaker area** (projected) of the speaker is computed by selecting either **Selected boundaries** (the default) if all boundaries are present, or **Use symmetries** if the speaker surface is only partially represented.

When **Use symmetries** is selected, select the **Speaker area multiplication factor** A_{scale} as **Automatic** (the default, the model is analyzed for the presence of symmetry conditions); or **User defined** and enter a value (default is 1). The settings are required in order to compute the effective area of the speaker which is used to compute the acoustic load and radiated power.

Define the **Speaker axis direction** $\mathbf{e}_{\mathrm{ax}}$ by selecting **Automatic** (the default) or **User defined**. For the automatic option, the axis is computed as the average of the surface normals; this option is valid if a full speaker surface is selected. The **User defined** option should be used if the speaker surface is only partially represented and is the only option when **Use symmetries** is selected.

For the **Automatic** option select **Reverse automatic speaker axis direction** if necessary. The axis orientation can be visualized by plotting the axis components `actd.ilsb1.e_axx`, `actd.ilsb1.e_axy`, and `actd.ilsb1.e_axz`. (using the proper item and physics scope).

INITIAL VALUE

Enter the initial value of the force F_c and axial speaker position x_{ax}, used when solving the associated ODEs. The default value is 0 for both and is typically correct, except if the simulation is started, for example, with an initial excursion of the driver.

CIRCUIT

This section gives information about the state of coupling/connection to an Electrical Circuit interface. If the **Interior Lumped Speaker Boundary** is not connected, the text **Connect to 'External I vs. U' in the Electrical Circuit interface** text will be displayed. Once connected, a reference with a tag to the associated **External I vs. U** node, in the Electrical Circuit interface, is displayed.

The Pressure Acoustics, Boundary Mode Interface

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

The physics interface solves the Helmholtz eigenvalue equation on boundaries, searching for the out-of-plane wave numbers at a given frequency. The out-of-plane extension is added analytically to the governing equations. The analysis is also sometimes referred to as semianalytical finite elements or SAFE.

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

SETTINGS

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

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

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

EQUATION

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

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

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

SOUND PRESSURE LEVEL SETTINGS

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

DEPENDENT VARIABLES

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

DISCRETIZATION

From the list select the element order and type (Lagrange or serendipity) for the **Pressure**, the default is **Quadratic Lagrange**.

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

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

Initial Values

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

INITIAL VALUES

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

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

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

The Pressure Acoustics, Boundary Mode Interface has many of its feature nodes in common with the Pressure Acoustics, Frequency Domain interface. For some features only selected options exist. See Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface.

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

Impedance

Use the **Impedance** condition to specify a locally reacting/absorbing surface on the boundary of the waveguide where the *Boundary Mode* analysis is carried out.

Enter a value for the **Specific impedance**. The default is `1.2[kg/m^3]*343[m/s]`.

The Pressure Acoustics, Boundary Elements Interface

The **Pressure Acoustics, Boundary Elements (pabe)** interface ( ), found under the **Pressure Acoustics** branch ( ) when adding a physics interface, is used to compute the pressure variation for the propagation of acoustic waves in gases and liquids. It is suited for all frequency-domain simulations with harmonic variations of the pressure field. The formulation is based on the boundary element method (BEM) and the interface is available in 2D and 3D. The physics interface solves Helmholtz equation for constant-valued material properties and uses the pressure as the dependent variable.

The interface is fully multiphysics enabled and can be coupled seamlessly with the physics interfaces that are based on the finite element method (FEM). This includes coupling to vibrating structures with the Acoustic–Structure Boundary multiphysics coupling, to FEM acoustic domains, with the Acoustic FEM–BEM Boundary multiphysics coupling, to thermoviscous acoustic domains with the Acoustic–Thermoviscous Acoustic Boundary multiphysics coupling, and to poroelastic waves domains with the Acoustic–Porous Boundary multiphysics coupling. This approach allows modeling in a FEM-BEM framework, using the strength of each formulation adequately. The BEM-based interface is especially well suited for radiation and scattering problems.

The advantage of the boundary element method is that only boundaries need to be meshed and the degrees of freedom (DOFs) solved for are restricted to the boundaries. This introduces some clear ease-of-use for handling complex geometries. However, the BEM technique results in fully populated or dense matrices that need dedicated numerical methods. The BEM method is so to speak more expensive per DOF than the FEM method but has fewer DOFs. Assembling and solving these can be very demanding. This means that when solving acoustic models of small and medium size, The Pressure Acoustics, Frequency Domain Interface will often be faster than solving the same problem with the BEM interface. The challenge for the FEM interface is to set up open boundaries, for example, using PMLs, in an efficient way. When the geometries are complex or two structures are far apart, large air domains need to be meshed. This costs a lot on the computational side as the frequency is increased.

For acoustically large models (problems that contain many wavelengths, at high frequency or for large domains) the stabilized formulation option (see Stabilization)

ensures efficient convergence at the cost of some additional degrees of freedom. For low to medium frequencies (small to medium models), running without stabilization is more efficient. The stabilized formulation only gives a benefit in computing time for the acoustically large models.

	For good modeling strategies, meshing, solvers, postprocessing information, as well as tips and tricks, see the Modeling with the Pressure Acoustics Branch (BEM-Based Interface) section.

	• *Head and Torso HRTF Computation*. Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/head_torso_hrtf** • *Spherical Scatterer: BEM Benchmark*. Application Library path **Acoustics_Module/Verification_Examples/spherical_scatterer_bem_benchmark** • *Submarine Target Strength*. Application Library path **Acoustics_Module/Underwater_Acoustics/submarine_target_strength**

	For tutorials that uses a BEM-FEM multiphysics coupling, to model an acoustic problem using the hybrid method see: • The *Bessel Panel*. Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/bessel_panel** • The *Loudspeaker Driver in a Vented Enclosure*. Application Library path **Acoustics_Module/Electroacoustic_Transducers/vented_loudspeaker_enclosure**

The governing Helmholtz equation defined by the Pressure Acoustics, Boundary Element interface is given by:

$$-\frac{1}{\rho_c}\nabla^2 p_t - \frac{k_{eq}^2}{\rho_c} p_t = 0$$
$$p_t = p + p_b \qquad k_{eq}^2 = \left(\frac{\omega}{c_c}\right)^2 \tag{2-1}$$

where p_t is the total acoustic pressure, k_{eq} is the wave number, ρ_c is the density, and c_c is the speed of sound. The subscript "c" denotes that these can be complex-valued quantities in models with damping. The governing equations and boundary conditions

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

The boundary element method (BEM) used in the Pressure Acoustics, Boundary Element interface is based on the direct method with Costabels symmetric coupling. The adaptive cross approximation (ACA) fast summation method is used for solving the resulting linear system. This method uses partial assembly of the matrices where the effect of the matrix vector multiplication is computed. The default iterative solver used is GMRES. With the built-in multiphysics couplings, it is easy and seamless to set up problems that combine FEM- and BEM-based physics. When solving these coupled models, a hybrid solver is automatically generated with an appropriate preconditioner for the FEM part of the problem (direct or multigrid), if applicable.

More details about the BEM formulation can be found in The PDE, Boundary Elements Interface documentation in the *COMSOL Multiphysics Reference Manual*.

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

Then, from the **Physics** toolbar, add other nodes that implement boundary conditions. You can also right-click **Pressure Acoustics, Boundary Elements** to select physics features from the context menu. Infinite conditions like a symmetry plane or an infinite sound hard boundary are defined in the Symmetry/Infinite Boundary Condition section.

- Domain, Boundary, Edge, and Pair Nodes for the Pressure Acoustics, Boundary Elements Interface
- Theory Background for the Pressure Acoustics Branch

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

SETTINGS

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

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

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

EQUATION

Expand the **Equation** section to see the equations solved for with the **Equation form** specified. The default selection is **Equation form** is set to **Study controlled**. The available studies are selected under **Show equations assuming**. If **Frequency domain** is selected as the **Equation form** the **Frequency** of the study can be either taken **From solver** or **User defined**.

PHYSICS SYMBOLS

Select the **Enable physics symbols** checkbox to display the infinite boundary condition lines or planes in the geometry.

SOUND PRESSURE LEVEL SETTINGS

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

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

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

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

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

SYMMETRY/INFINITE BOUNDARY CONDITION

In this section, you can specify infinite boundary conditions for the boundary element problem. These are conditions that are not added on a boundary but apply to a line in 2D or a plane in 3D. The planes where the conditions are applied are visualized if the **Enable physics symbols** is turned on (the default) in the Physics Symbols section.

For pressure acoustics a symmetry condition is mathematically equivalent to a sound hard boundary, and an antisymmetry condition is equivalent to a sound soft boundary. In this way you can include a **Symmetry/Infinite sound hard boundary** or an **Antisymmetry/Infinite sound soft boundary** along "infinite lines" (in 2D) or "infinite planes" in (3D). Only one condition per Cartesian direction can be applied.

For the infinite conditions to be mathematically valid, it is important that all sources and BEM boundaries are located on the same side of (or on) the infinite conditions. The condition can, for example, not be used as an infinite baffle with sound radiating through a hole. In this case, the propagation domain is on a different side than the source. The Exclude Boundary feature can be used to exclude boundaries from the BEM problem for advanced symmetry configurations.

Choose an option from the **Condition for the x = x_0 plane**, **Condition for the y = y_0 plane**, and **Condition for the z = z_0 plane** lists (when applicable). In 2D, these are out-of-plane surfaces. Choose one of the following options:

- **Off**, for no symmetry (the default)
- **Symmetric/Infinite sound hard boundary**
- **Antisymmetric/Infinite sound soft boundary**

Then enter the value for the plane location x_0, y_0, or z_0 (the default is 0 m). This allows an offset of the infinite condition planes along the main coordinate axes.

There are important aspect on how results are visualized when symmetries are used:

- First, when setting the **Parameter Bounds** in the grid dataset. Say the condition represents a symmetry plane then visualize the solution in the entire domain. On the other hand if the condition represents an infinite sound hard boundary, like the ground or an infinite baffle, then only visualize the solution above that plane by setting the **Parameter Bounds** in the **Grid 3D** or **Grid 2D** dataset in **Results**.
- Secondly, evaluation on the grid dataset will also show results inside radiating/scattering objects, on the symmetry side. These are unphysical results and can be removed using a **Filter** subnode with a logic expression.

FAR-FIELD APPROXIMATION

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

These settings are used for matrix assembly and postprocessing. They allow characterization of interactions occurring in the boundary element method as near-field or far-field interactions. While the near-field interactions are represented explicitly, the far-field interactions can be represented in an approximate way. This approach results in considerable memory and performance improvements when used in combination with iterative solvers using matrix-free format or during postprocessing. The near-field part of the stiffness matrix is used as input by the Direct and Sparse Approximate Inverse preconditioners.

The **Use far-field approximation** checkbox is selected by default in order to accelerate the solution process. If the checkbox is cleared, the solution will be slightly more accurate but the computational time and memory consumption may become prohibitively high.

The **Approximation type** can be either **ACA+** or **ACA**. These alternatives correspond to two different versions of the adaptive-cross-approximation (ACA) method, which is a fast matrix multiplication method based on far-field approximations.

More details are found under Far-Field Approximation Settings in The PDE, Boundary Elements Interface documentation in the *COMSOL Multiphysics Reference Manual*.

CONDITION AT INFINITY

In this section, you specify the condition to apply at infinity for an unbounded problem. For the Helmholtz equation, solved here, choose to specify an **Outgoing wave** (the default) or an **Incoming wave**. In almost all cases the **Outgoing wave** option should be selected.

QUADRATURE

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

The quadrature settings control the details of the numerical integration performed when setting up the BEM problem.

Select **Thin gap handling** if narrow regions exist in the model. The **Thin gap handling** checkbox is used to increase the accuracy of the quadrature between almost-touching mesh element pairs, that is, pairs that are extremely close with respect to element size. The **Threshold distance (relative to mesh size)** controls the distance at which elements are considered almost-touching. A larger value should provide more accurate results, but at an increased computational cost. Under the **Integration order** settings, the field **Integration order, very close elements (thin gap)** becomes active (when **Manual** is selected), allowing a higher-order rule for such elements. In 3D, an additional variable transformation is applied to improve the accuracy even further.

The **Integration orders** are by default set to **Automatic**. This means that the quadrature integration order values will follow the element order selection in the **Discretization** section. Higher element orders automatically generate higher values for the quadrature integration orders. To see the used values or to set user-defined values, switch to **Manual**. Note that the values are reset to default when switching back and forth between the **Automatic** and **Manual** settings. The **Integration order, very close elements (thin gap)** is only editable when **Thin gap handling** is selected.

More details are found under Quadrature in The PDE, Boundary Elements Interface documentation in the *COMSOL Multiphysics Reference Manual.*

STABILIZATION

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog.

For acoustically large models (problems that contain many wavelengths, at high frequency or for large domains) enable the **Stabilized formulation** option to ensure efficient convergence at the cost of some additional degrees of freedom.

When **Stabilized formulation** is selected, a text field for the **Stabilization parameter** is enabled with the default value `sqrt(abs(pabe.k[m]))`. This is a parameter that should scale inversely with the wavelength. The default gives good performance in most cases.

If the **Stabilized formulation** is enabled after a nonstabilized model has been solved (maybe showing slow convergence) it is important to reset the solver default. A different solver configuration is used for the stabilized BEM formulation.

Note that interior boundary conditions like Interior Sound Hard Boundary (Wall) or Interior Normal Acceleration are not supported with the **Stabilized formulation**.

For a model that uses the **Stabilized formulation**, see the *Submarine Target Strength* tutorial. Application Library path **Acoustics_Module/Underwater_Acoustics/submarine_target_strength**

POSTPROCESSING SETTINGS

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

This setting controls the interpolation that is done close to boundaries (typically in the last mesh element) between variables on boundaries (these are the variables solved for in BEM based interfaces) and their value evaluated with the kernel in the domains. Numerically the kernel evaluation is prone to errors near the boundaries while the DOFs solved for are exact on the boundaries. Using an interpolation in the inner mesh element removes singularities in the results when postprocessing values in the domain near a boundary.

Enter an expression for the **Interpolation distance from boundary**. The entered value defines the distance beyond which the kernel evaluation is used. The default is `0.5*h` which gives reasonable results. This default corresponds to half the mesh size `h`. If the value is set to `0` no interpolation is done.

See also Postprocessing BEM Results in the Modeling with the Pressure Acoustics Branch (BEM-Based Interface) section.

DISCRETIZATION

From the **Dependent variable/Normal boundary flux** list, choose from predefined options for the boundary element discretization order for the dependent variable and the normal boundary flux. The predefined options represent the suitable combinations of element orders such as **Quadratic/Linear** (the default).

The settings under **Value types when using splitting of complex variables** are important for sensitivity and optimization computations. See the description of the built-in operators `fsens` and `fsensimag`.

DEPENDENT VARIABLES

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

Evaluation of the BEM results is best handled using the Grid 2D or Grid 3D datasets. These are automatically set up and are used in all tutorial models. Details are found in the Modeling with the Pressure Acoustics Branch (BEM-Based Interface) section.

To do a kernel evaluation of the BEM solution at a given coordinate use the `at3_spatial()` operator in 3D or the `at2_spatial()` operator in 2D. Both called with the `minc` argument, for mesh independent evaluations.

In a 3D model you can, for example, evaluate the sound pressure level in the point $(x,y,z) = (1\text{ m},0,0)$ by typing:
`at3_spatial(1[m],0,0,pabe.Lp,'minc').`

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

The Pressure Acoustics, Boundary Elements Interface has the following domain and boundary conditions (in alphabetic order):

- Background Pressure Field
- Impedance
- Initial Values
- Interior Normal Acceleration
- Interior Normal Displacement
- Interior Normal Velocity
- Interior Sound Hard Boundary (Wall)
- Normal Acceleration
- Normal Displacement
- Normal Velocity
- Pressure
- Pressure Acoustics
- Sound Hard Boundary (Wall)
- Sound Soft Boundary

Pressure Acoustics

The **Pressure Acoustics** node adds the equations for solving the Helmholtz equation, defined by Equation 2-1, in the frequency domain using the boundary element method. The material properties have to be constant valued for each **Pressure Acoustics** node added.

Each **Pressure Acoustics** node adds the necessary BEM equations to the adjacent boundaries of the selection (with the material properties defined). The domain features in the BEM interface can be applied to all types of domain selections: solid domains, finite voids, and the infinite void. Only the exterior boundaries of the selections as well as applied interior boundary conditions need to be meshed. *Finite voids* are selections of domains that have no solids in the geometry but are only defined by a closed set of boundaries. The *Infinite void* represents the exterior most infinite domain. This selection is the typical used for radiation or scattering problems. Remember that the materials defined in the **Materials** node need to be applied to the same selections.

See also Selections: Infinite Void and Finite Voids in the Modeling with the Pressure Acoustics Branch (BEM-Based Interface) section.

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

MODEL INPUTS

The values of the quiescent (or background) **Temperature** T (SI unit: K) and **Absolute pressure** p_A (SI unit: Pa) are entered in the **Model Inputs** section. The values of the model inputs can influence the material properties like, for example, the density and speed of sound, depending on their definition in the **Materials** node. In the BEM formulation these two values have to be constant valued for each **Pressure Acoustics** node added.

PRESSURE ACOUSTICS MODEL

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

conducting and viscous, or **Ideal gas**. The settings for the fluid models are the same as for Pressure Acoustics in The Pressure Acoustics, Frequency Domain Interface.

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

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

Background Pressure Field

Add a **Background Pressure Field** node to add a background pressure wave to model scattering problems solving for the scattered pressure field p_s. The total field p_t is the sum of the scattered field and the background pressure field p_b:

$$p_t = p_b + p_s$$

This feature sets up the equations in a so-called scattered field formulation where the dependent variable is the scattered field $p = p_s$. When added, the background pressure field is applied everywhere; it has a locked selection set to all.

In models solved with the boundary element method, it is important that the defined **Background Pressure Field** is an exact solution to the governing equation. If this is not the case, the results will be unphysical. This means that setting up a user-defined field needs careful consideration.

BACKGROUND PRESSURE FIELD

Select a **Background pressure field type**: **Plane wave** (the default), **Cylindrical Wave**, **Spherical Wave**, or **User defined**. Most of the options and settings here are the same as for the Background Pressure Field in The Pressure Acoustics, Frequency Domain Interface.

The only difference lies in the definition of the material properties for the background field. In the BEM interface define a **Wave number** either picked up from a Pressure Acoustics model in the physics tree (the default), for example, **Wave number (pabe/bpam1)**, or defined **From speed of sound**. This option ensures that the background field

is a solution to the governing equation. If several material models are added, they must have the same speed of sound, to ensure that the background pressure field is a solution to the underlying Helmholtz equation.

Initial Values

The **Initial Values** node adds an initial value for the pressure.

Sound Hard Boundary (Wall)

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

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

To apply a sound hard boundary condition to an infinite plane in the boundary element model, see the Symmetry/Infinite Boundary Condition functionality.

Normal Acceleration

The **Normal Acceleration** adds a source at an exterior boundary defined by the inward normal acceleration a_n or alternatively given through an acceleration $\mathbf{a}_0$:

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}\nabla p_t\right) = a_n = -\mathbf{n}\cdot\mathbf{a}_0$$

NORMAL ACCELERATION

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

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

Normal Velocity

The **Normal Velocity** adds a source at an exterior boundary defined by the inward normal velocity v_n or alternatively given through a velocity $\mathbf{v}_0$:

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

NORMAL VELOCITY

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

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

Normal Displacement

The **Normal Displacement** adds a source at an exterior boundary defined by the inward normal displacement d_n or alternatively given through a displacement $\mathbf{d}_0$:

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

NORMAL DISPLACEMENT

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

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

Sound Soft Boundary

The **Sound Soft Boundary** adds a boundary condition for a sound soft boundary, where the acoustic pressure vanishes: $p_t = 0$. It is an appropriate approximation for a liquid-gas interface and in some cases for external waveguide ports. To apply a sound soft condition to an infinite plane in the boundary element model, see the Symmetry/Infinite Boundary Condition functionality.

Pressure

The **Pressure** node creates a boundary condition that acts as a pressure source at the boundary, which means that a constant acoustic pressure p_0 is specified and maintained at the boundary: $p_t = p_0$. The pressure can be a complex number and thus includes phase information.

PRESSURE

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

Impedance

The **Impedance** node adds an impedance boundary condition with the option to select between several built-in impedance models and engineering relations. All the options are the same as for the Impedance condition of The Pressure Acoustics, Frequency Domain Interface. All the settings and models are described there.

Exclude Boundary

The **Exclude Boundary** feature is used to remove boundaries from the boundary element problem. The selected boundaries will neither contribute to the assembly of the BEM problem nor to the solution. The condition can be used to set up models where the BEM interface is used as an ideal infinite domain in an infinite baffle (infinite half space) setup when coupled to the *Pressure Acoustics, Frequency Domain*. In these models, all boundaries of the FEM problem need to be excluded, except for the ones in the multiphysics coupling.

For a tutorial that uses a BEM-FEM coupling and the **Exclude Boundary** feature to model an acoustic problem in an infinite baffle setup see the tutorial *Piezoelectric Tonpilz Transducer with a Prestressed Bolt*. Application Library path **Acoustics_Module/Piezoelectric_Devices/tonpilz_transducer_prestressed**

Interior Sound Hard Boundary (Wall)

The **Interior Sound Hard Boundary (Wall)** sets up a wall condition on an interior boundary. Add this node from the **Interior Conditions** submenu. The pressure is discontinuous across this boundary.

When postprocessing and plotting variables on boundaries with interior conditions, the boundary variables (with extension `_bnd`) give the average value of the two sides. Use the up (`_up`) and down (`_down`) variants to depict the values on both sides of the boundary.

See also Postprocessing BEM Results in the Modeling with the Pressure Acoustics Branch (BEM-Based Interface) section.

In the BEM formulation, interior conditions cannot intersect other (exterior) conditions. It is recommended that this type of model is set up by using FEM in the region around the interior condition and BEM outside of that region.

Interior Normal Acceleration

The **Interior Normal Acceleration** sets up an interior source that can be used to model a thin vibrating structure, for example, a loudspeaker diaphragm. Add this node from the **Interior Conditions** submenu.

The pressure is discontinuous across this boundary. Use the up and down version of the postprocessing variables to visualize quantities on the two sides of the boundary, for example, the up side total pressure is `pabe.p_t_up`.

INTERIOR NORMAL ACCELERATION

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

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

In the BEM formulation, interior conditions cannot intersect other (exterior) conditions. It is recommended that this type of model is set up by using FEM in the region around the interior condition and BEM outside of that region.

PHYSICS SYMBOLS

Select **Show physics symbols** to show the normal direction as a symbol (red arrow) in the Graphics window (selected per default).

Interior Normal Velocity

The **Interior Normal Velocity** sets up an interior source that can be used to model a thin vibrating structure, for example, a loudspeaker diaphragm. Add this node from the **Interior Conditions** submenu.

The pressure is discontinuous across this boundary. Use the up and down version of the postprocessing variables to visualize quantities on the two sides of the boundary, for example, the up side total pressure is `pabe.p_t_up`.

INTERIOR NORMAL VELOCITY

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

- For **Velocity** enter values for the components of the **Velocity** $\mathbf{v}_0$ (SI unit: m/s).
- For **Normal velocity** enter the value of the **Normal velocity** v_n (SI unit: m/s) in the normal direction.

In the BEM formulation, interior conditions cannot intersect other (exterior) conditions. It is recommended that this type of model is set up by using FEM in the region around the interior condition and BEM outside of that region.

PHYSICS SYMBOLS

Select **Show physics symbols** to show the normal direction as a symbol (red arrow) in the Graphics window (selected per default).

Interior Normal Displacement

The **Interior Normal Displacement** sets up an interior source that can be used to model a thin vibrating structure, for example, a loudspeaker diaphragm. Add this node from the **Interior Conditions** submenu.

The pressure is discontinuous across this boundary. Use the up and down version of the postprocessing variables to visualize quantities on the two sides of the boundary, for example, the up side total pressure is `pabe.p_t_up`.

INTERIOR NORMAL DISPLACEMENT

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

- For **Displacement** enter values for the components of the **Displacement** $\mathbf{d}_0$ (SI unit: m).
- For **Normal displacement** enter the value of the **Normal displacement** d_n (SI unit: m) in the normal direction.

In the BEM formulation, interior conditions cannot intersect other (exterior) conditions. It is recommended that this type of model is set up by using FEM in the region around the interior condition and BEM outside of that region.

PHYSICS SYMBOLS

Select **Show physics symbols** to show the normal direction as a symbol (red arrow) in the Graphics window (selected per default).

The Pressure Acoustics, Time Explicit Interface

The **Pressure Acoustics, Time Explicit (pate)** interface ( ), found under the **Acoustics** > **Pressure Acoustics** branch ( ) when adding a physics interface, is used to compute the pressure variation when modeling the propagation of acoustic waves in fluids at quiescent background conditions. The interface is used to solve large transient linear acoustic problems containing many wavelengths. It is suited for time-dependent simulations with arbitrary time-dependent sources and fields. The interface includes a Background Acoustic Field option for modeling of scattering problems. Absorbing layers are used to set up effective nonreflecting-like boundary conditions. The exterior field can be calculated by combining the *Exterior Field Calculation* feature with a *Time to Frequency FFT* study step. The interface exists in 2D, 2D axisymmetry, and 3D. Losses at boundaries can be modeled with the Impedance condition. The interface has built-in options that allow setting up frequency dependent impedance conditions in the time domain, by using the very general and flexible General local reacting (rational approximation) impedance option. Losses in porous domains materials can be modeled with the Poroacoustics feature.

The interface is based on the discontinuous Galerkin method (dG-FEM) and uses a time-explicit solver. The method is very memory lean and is well suited for cluster computing. The interface has dedicated support for running on a GPU using an accelerated formulation. Application areas include the transient propagation of audio pulses in room acoustics or modeling scattering phenomena involving large objects relative to the wavelength.

If the acoustic waves propagate in a background flow, for example, inside a flowmeter, then use The Convected Wave Equation, Time Explicit Interface.

For modeling vibroacoustic problems that involve acoustic structure interaction (ASI) the Pressure Acoustics, Time Explicit interface can be combined with The Elastic Waves, Time Explicit Interface using the Acoustic–Structure Boundary, Time Explicit multiphysics coupling.

The interface solves the linearized Euler equations assuming an adiabatic equation of state. The dependent variables are the acoustic pressure and the acoustic velocity perturbations. Bulk losses (volume attenuation) can be included in the interface using

the classical expression for thermal and viscous attenuation or a general dissipation term.

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

For good modeling strategies, meshing, solvers, postprocessing information, acoustics specific plots, as well as tips and tricks, see the Modeling with the Pressure Acoustics Branch (DG-FEM-Based Interface) section.

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

- *Wave-Based Time-Domain Room Acoustics with Frequency-Dependent Impedance*. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/wave_based_room**
- For a tutorial model that uses the interface to model a scattering problem, see the *Submarine Scattering: Time-Domain Simulation and FFT* tutorial in the COMSOL Application Gallery: https://www.comsol.com/model/submarine-scattering-time-domain-simulation-and-fft-55921

SETTINGS

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

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

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

FILTER PARAMETERS FOR ABSORBING LAYERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options**. In the **Filter Parameters for Absorbing Layers** section you can change and control the values set for the filter used in the Absorbing Layers. The values of the filter parameters defined here are used in all absorbing layers added to the model and they override the value of filter parameters enabled in the material model (Pressure Acoustics, Time Explicit Model). The default values of the filter parameters α, η_c, and s are set to 0.1, 0.01, and 2, respectively. Inside the absorbing layer it is important to use a filter that is not too aggressive since this will result in spurious reflections.

For general information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

SOUND PRESSURE LEVEL SETTINGS

The settings selected here are only used if the transient solution solved is transformed into the frequency domain using the **Time to Frequency FFT** study. The zero level on the dB scale varies with the type of fluid. That value is a reference pressure that corresponds to 0 dB. This variable occurs in calculations of the sound pressure level L_p based on the root mean square (rms) pressure p_{rms}, such that

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

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

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

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

TRANSIENT MESH SETTINGS

Enter a value for the **Maximum frequency to resolve** f_{max} (the default is `1000[Hz]`). This value is used to set up the physics-controlled mesh for the Pressure Acoustics, Time Explicit interface.

MODEL EQUATIONS AND SOLVER SETTINGS

Select the **Use accelerated solver formulation** for an option where the system data is extracted and cashed for efficient evaluation on either on GPU or CPU. With this option selected, the number of available boundary conditions and features is limited.

Select **Compute residual on GPU** (the default) so that the accelerated solver runs on GPU. When selected, the option **Use single precision on GPU** is active and selected per default. Single precision is then used on the GPU. If the **Use single precision on GPU** is not selected, double precision is used. Single precision is often adequate for wave propagation problems and gives the best speedup and performance.

If the **Compute residual on GPU** is not selected, the residuals are computed on CPU, but still extracted and cashed, which will result in solver speedup, compared to the default formulation. If the solved problem can fit in the memory of the GPU, this results in the most significant speedup.

If changes are made to the settings in this section, make sure to **Reset Solver to Default** on the study level.

The available features, when **Use accelerated solver formulation** is selected, are:

- Pressure Acoustics, Time Explicit Model
- Initial Values
- Sound Hard Boundary (Wall)
- Pressure
- Symmetry
- Normal Velocity
- Impedance

The **Use accelerated solver formulation** is also supported in **Absorbing Layer** domains, for modeling open problems. When enabled the source definition in Pressure and Normal Velocity also needs to redefined. See the features for details.

See also Hardware Acceleration in the *COMSOL Multiphysics Reference Manual*.

DISCRETIZATION

In this section you can select the discretization for the **Acoustic pressure** and **Acoustic velocity**. Per default both are set to **Quartic** (4th order). Using quartic elements

together with a mesh size equal to approximately half the wavelength to be resolved, leads to the best performance when using the DG method. For further details see the Meshing, Discretization, and Solvers section.

When solving **Pressure Acoustics, Time Explicit**, it is important to have consistent settings for the Geometry Shape Function and the Discretization of the physics. The **Automatic** setting for the **Geometry shape function** (in the Curved Mesh Elements section on the Components node's settings) may results in a linear geometry representation, if other physics are present in the model. This can lead to numerical errors when solving as the default is to use fourth-order (**Quartic**) spatial discretization of the dependent variables. To remedy this change the **Geometry shape function** to **Quadratic Lagrange**.

If the settings in a model are inconsistent, a warning is given in the solver with the text:

```
The geometry shape function order is not consistent with
the discretization of the dependent variables.
Consider increasing the geometry shape function order to
Quadratic Lagrange in the Curved Mesh Element section in
the Component node's settings.
```

DEPENDENT VARIABLES

The dependent variables are the **Acoustic pressure**, and the **Acoustic velocity**. The names can be changed, but the names of fields and dependent variables must be unique within a model. The name for the **Acoustic velocity**, **components** can also be selected individually.

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

The Pressure Acoustics, Time Explicit Interface has the following domain and boundary conditions listed in alphabetic order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users). Continuity in velocity and pressure is the default condition on interior boundaries. If a boundary separates two different materials the

Material Discontinuity or Continuity (the pair version of material discontinuity) condition has to be applied to ensure the continuity in velocity and pressure.

- Background Acoustic Field
- Continuity
- Continuity of Total Fields
- Exterior Field Calculation
- General Flux/Source
- General Interior Flux
- Heat Source
- Impedance
- Interior Normal Velocity
- Interior Sound Hard Boundary (Wall)
- Initial Values
- Mass Source
- Material Discontinuity
- Normal Velocity
- Poroacoustics
- Pressure
- Pressure Acoustics, Time Explicit Model
- Sound Hard Boundary (Wall)
- Sound Soft Boundary
- Symmetry
- Volume Force Source

Pressure Acoustics, Time Explicit Model

The **Pressure Acoustics, Time Explicit Model** node adds the equations to model the transient propagation of linear acoustics waves based on the dG-FEM time explicit method. For the time explicit method used, the governing equations are formulated as a first order system, in terms of the linearized continuity equation and the linearized momentum equation, as

$$\begin{aligned} \frac{1}{\rho c^2}\frac{\partial p_t}{\partial t} + \nabla \cdot \mathbf{u}_t &= Q_m \\ \rho \frac{\partial \mathbf{u}_t}{\partial t} + \nabla \cdot (p_t \mathbf{I}) &= \mathbf{q}_d \end{aligned} \tag{2-2}$$

where t is time, p_t is the total acoustic pressure, $\mathbf{u}_t$ is the total acoustic velocity, ρ is the fluid density, c is the speed of sound, and $\mathbf{I}$ is the unit matrix. Domain sources like a Mass Source, Heat Source, or Volume Force Source can be defined through the right hand sides Q_m and $\mathbf{q}_d$. These two source terms represent the same quantities as the Monopole Domain Source and the Dipole Domain Source in The Pressure Acoustics, Frequency Domain Interface and The Pressure Acoustics, Transient Interface.

The dependent variables are the acoustic pressure p and the acoustic velocity $\mathbf{u}$ (sometimes called particle velocity or perturbation velocity). The equations are formulated in the total fields p_t and $\mathbf{u}_t$ which is the sum of the scattered field (the dependent variable solved for) and a possible background field p_b and $\mathbf{u}_b$. The background field can be set up using the Background Acoustic Field feature.

The equations solved are the linearized Euler equations in a quiescent setting. The equations are easily combined to generate the scalar wave equation solved in The Pressure Acoustics, Transient Interface as discussed in Theory Background for the Pressure Acoustics Branch.

In the formulation of the wave equation, the speed of sound c and density ρ may in general be space dependent but only slowly varying in time, that is, at a time scale much slower than the variations in the acoustic signal.

The equations solved in the *Pressure Acoustics, Time Explicit* interface are closely related to the equations solved in The Convected Wave Equation, Time Explicit Interface. There is no background flow option in the *Pressure Acoustics, Time Explicit interface* but, on the other hand, it uses a scattered field formulation that allows solving scattering problems. For nonlinear acoustic problems The Nonlinear Pressure Acoustics, Time Explicit Interface should be used.

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

PRESSURE ACOUSTICS MODEL

Select a **Fluid model**: **Linear elastic** (the default), **Viscous**, **Thermally conducting**, **Thermally conducting and viscous**, **General dissipation**, or **Ideal gas**.

- If **Linear elastic** is selected, enter the speed of sound c and the density ρ.
- If **Viscous**, **Thermally conducting**, or **Thermally conducting and viscous** is selected, enter the fluid properties (see the Transient Pressure Acoustics Model for details). These three options will define the sound diffusivity δ through the classical material properties. The options are equivalent to defining the classical thermoviscous attenuation factor α_{tv} in the frequency domain.
- If **General dissipation** is selected, enter the speed of sound c, the density ρ, and the sound diffusivity δ. This option can be used for modeling fluids with measured damping properties. The relation between the sound diffusivity δ and the equivalent

(plane wave) attenuation coefficient α, at a given frequency f, is given by the expression

$$\alpha = \frac{\delta\omega^2}{2c^3}$$

where $\omega = 2\pi f$ and c is the speed of sound.

- If **Ideal gas** is selected, enter the combination of material properties defining the gas.

For all options, the default is to use the material property values from **From material**, select **User defined** from the list to enter a user-defined value or expression in the text field that appears. For numerical stability reasons it is recommended to use physical values of attenuation properties.

Selecting any of the dissipations models will modify the governing Equation 2-2 by adding a right hand side to the momentum equation defined as

$$\mathbf{q}_d = \rho\delta\nabla\cdot(\nabla\mathbf{u}) = \rho\delta\Delta\mathbf{u} \tag{2-3}$$

This term has a small cost on the computation time when solving the model. The term has to be evaluated at every time-step taken by the solver.

In transient models, when space dependent material data is given by an interpolation function, significant performance gains can be achieved with the following setup of the **Interpolation** function:

- Select **Use spatial coordinates as arguments**
- Set the **Frame** to **Mesh**
- Select **Reinterpolate interpolation data on computational mesh**

No arguments should be used when calling the interpolation function, for example, use `int` and not `int(x,y,z)`.

See the Interpolation section in the *COMSOL Multiphysics Reference Manual*.

Note that for the time explicit formulation, the material data has to change smoothly. For abrupt changes the Material Discontinuity condition has to be used on an interior boundary.

LAX–FRIEDRICHS FLUX PARAMETERS

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog. In this section, you specify the value of the *Lax–Friedrichs flux parameter* τ_{LF} (default value: 0.2). This value controls the numerical flux between the elements (nodal discontinuous Lagrange elements) used with the discontinuous Galerkin (DG) method. The numerical flux defines how adjacent elements are connected and how continuous p and $\mathbf{u}$ are. Different definitions of the numerical flux lead to different variants of the DG method. The flux implemented here is the so-called global Lax–Friedrichs numerical flux. The value of the parameter τ_{LF} should be between 0 and 0.5. For $\tau_{LF} = 0$ a so-called central flux is obtained. Setting $\tau_{LF} = 0.5$ gives a maximally dissipative global Lax–Friedrichs flux.

For general information about the numerical flux see the Numerical Flux section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

FILTER PARAMETERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. By default, the filter parameters α, η_c, and s are not active. Select the **Activate** checkbox to activate the filter. The filter provides higher-order smoothing for the DG formulation. Inside absorbing layers the settings given here are overridden by the Filter Parameters for Absorbing Layers.

For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

Background Acoustic Field

Add a **Background Acoustic Field** node to model a background/incident pressure wave to study the scattered pressure fields p_s and $\mathbf{u}_s$. The scattered fields are defined as the difference between the total acoustic fields p_t and $\mathbf{u}_t$ and the background acoustic field p_b and $\mathbf{u}_b$:

$$p_t = p_b + p_s$$
$$\mathbf{u}_t = \mathbf{u}_b + \mathbf{u}_s$$

This feature sets up the equations in a so-called scattered field formulation where the dependent variable is the scattered field ($p = p_s$ and $\mathbf{u} = \mathbf{u}_s$). In a model where the background acoustic field is not defined on all acoustic domains (or it is different) continuity is automatically applied in the total fields p_t and $\mathbf{u}_t$ on interior boundaries between domains.

BACKGROUND ACOUSTIC FIELD

Select the **Pressure field type** as **Plane wave (monochromatic)** (the default) or **User defined**.

- For **Plane wave (monochromatic)** enter a value for the **Pressure amplitude** p_0 of the wave, select the **Speed of sound** and **Density** as **User defined** (enter a value for c or ρ) or **From material**, enter a **Wave direction** vector $\mathbf{e}_k$, enter the **Signal frequency** f_0 of the wave, and a possible **Phase** ϕ.

  These settings defined a background acoustic field as a plane propagating wave of the type:

$$p_b(\mathbf{x}, t) = p_0 \sin\left(2\pi f_0\left(t - \frac{\mathbf{x} \cdot \mathbf{e}_k}{c|\mathbf{e}_k|}\right) + \phi\right) \qquad \mathbf{u}_b(\mathbf{x}, t) = \frac{p_b}{\rho c}\frac{\mathbf{e}_k}{|\mathbf{e}_k|}$$

- For **Modulated Gaussian pulse** enter a value for the **Pressure amplitude** p_0 of the wave, select the **Speed of sound** as **User defined** (enter a value for c) or **From material**, enter a **Wave direction** vector $\mathbf{e}_k$ and enter the **Signal emitting plane coordinate** $\mathbf{x}_0$. The latter two options define the initial location of the pulse and the propagation direction. When modeling scattering problems the initial location should outside the computational domain. Next define the frequency content of the Gaussian pulse. Select the **Gaussian pulse bandwidth** as **Full width at half maximum, power** (the default), **Full width at half maximum, amplitude**, **Full duration at half maximum, power**, **Full duration at half maximum, amplitude**, or **Standard deviation**. For the first two options enter a frequency bandwidth Δf and for the latter a time window Δt. Finally, enter the carrier **Signal frequency (for time-dependent studies)** f_0.

  These settings defined a background pressure field p_b as a propagating modulated Gaussian pulse of the type:

$$p_b(\mathbf{x}, t) = p_0 G(\mathbf{x}, t) \sin\left(2\pi f_0\left(t - \frac{\mathbf{x} \cdot \mathbf{e}_k}{c|\mathbf{e}_k|}\right)\right) \qquad \mathbf{u}_b(\mathbf{x}, t) = \frac{p_b}{\rho c}\frac{\mathbf{e}_k}{|\mathbf{e}_k|}$$

  where the standard deviation σ of the Gaussian $G(\mathbf{x},t)$ depends on the bandwidth option selected. The modulated Gaussian pulse option only gives contributions when a time dependent model is solved. It has the advantage of having a defined

frequency content and can be used for limited bandwidth simulations. In combination with the Exterior Field Calculation and an FFT study the broadband scattering characteristics of an object can be analyzed.

- For **User defined** enter an analytical expression that can be function of both space and time or use a solution from a previous study.

ADVANCED SETTINGS

To display this section, when the **Plane wave (monochromatic)** option is selected, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

Click to select the **Use ramp function** option (selected per default). With this option turned on, a ramp is automatically added to smoothly increase the background pressure field amplitude over the first period $T = 1/f_0$. This option ensures numerical stability.

Poroacoustics

The **Poroacoustics** feature allow modeling of porous materials in the time domain, using an equivalent fluid model. The properties of the equivalent fluid, representing the porous material, are known (measured or simulated) in term of its real and imaginary part of the (isentropic) compressibility $\beta_s(f)$ and density $\rho(f)$, both functions of frequency. To capture the behavior in the time domain, the **Poroacoustics** feature has built-in functionality for setting up and defining the compressibility and density expressions through a rational approximation (or rational expansion) of the frequency dependent data. The approximation has an analytical inverse Fourier transform and can thus be used to define the same frequency dependency in the time domain, by setting up a system of memory equations. These equations are automatically defined by the feature and solved in the porous domain. Fitting of the frequency domain data can be performed using the Partial Fraction Fit function.

	Porous Absorber with Local and Extended Reacting Approximations for Time-Domain Modeling. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/porous_absorber_time_domain**

	For more information see Partial Fraction Fit under User-Defined Functions in the *COMSOL Multiphysics Reference Manual*.

POROACOUSTICS MODEL

The only option for the **Poroacoustics model** is **User defined**.

ISENTROPIC COMPRESSIBILITY

Select the **Partial fraction** fit as **User defined** (the default) or **From function**.

- For the **User defined** option enter all the necessary parameters that are present in the rational approximation (rational expansion), that is, the **Frequency-independent (asymptotic) compressibility** β_∞ (SI unit: $1/\text{Pa}$), enter the **Real residues and poles** (R and ξ) into the table, and enter the **Complex residues and poles** (Q and ζ) into the table (note that the conjugate terms are automatically added). The quantities can also be loaded from a file (remember to include the number column). Select if the rational approximation function is scaled for frequency data or angular frequency data, by selecting the **Equation scaled for frequency data** option (selected per default).
- For the **From function** option select the **Reference**, which is a Partial Fraction Fit function defined in the model, then click the **Import** icon ( ) to copy the fitted data to the compressibility field, and the residues and poles tables. To go to the function source click the **Go to Source** icon ( ).

DENSITY

Select the **Partial fraction** fit as **User defined** (the default) or **From function**.

- For the **User defined** option enter all the necessary parameters that are present in the rational approximation (rational expansion), that is, the **Frequency-independent (asymptotic) density** ρ_∞ (SI unit: kg/m^3), enter the **Real residues and poles** (R and ξ) into the table, and enter the **Complex residues and poles** (Q and ζ) into the table (note that the conjugate terms are automatically added). The quantities can also be loaded from a file (remember to include the number column). Select if the rational approximation function is scaled for frequency data or angular frequency data, by selecting the **Equation scaled for frequency data** option (selected per default).
- For the **From function** option select the **Reference**, which is a Partial Fraction Fit function defined in the model, then click the **Import** icon ( ) to copy the fitted

data to the density field, and the residues and poles tables. To go to the function source click the **Go to Source** icon ( ).

The necessary data, the complex-valued and frequency-dependent data for the compressibility and the density, can be generated from a small frequency domain model that uses the Poroacoustics in the The Pressure Acoustics, Frequency Domain Interface.

Define a domain with the desired Poroacoustics model, add the necessary material properties, and solve the model for the range of frequencies to be resolved (no sources are needed). Then add an **Evaluation Group** and a **Point Evaluation** (select a point in the porous domain). Evaluate `real(acpr.rho_c)` and `imag(acpr.rho_c)` for the density; and `real(1/acpr.K_eq)` and `imag(1/acpr.K_eq)` for the compressibility.

The Partial Fraction Fit function can point directly to the **Results table** or the data can be stored in a `.txt` file and referred to.

Initial Values

The **Initial Values** node adds initial values for the acoustic pressure and the acoustic velocity that can serve as an initial state for the solver. If more than one initial value is needed, from the **Physics** toolbar click to add more **Initial Values** nodes.

INITIAL VALUES

Enter a value or expression for the initial values for the **Pressure** p (SI unit: Pa) and the **Acoustic velocity**, $\mathbf{u}$ (SI unit: m/s).

Mass Source

The **Mass Source** sets up the right-hand side of the continuity equation, in Equation 2-2, in terms of a mass rate of change. This is a monopole-like source that acts uniformly in all directions.

$$Q_m = \frac{1}{\rho} M$$

MASS SOURCE

Enter the **Mass source** M (SI unit: kg/(m^3·s)).

Heat Source

The **Heat Source** sets up the right-hand side of the continuity equation, in Equation 2-2, in terms of a heat density source. This is a monopole-like source that acts uniformly in all directions.

$$Q_m = \frac{\alpha_p}{\rho C_p} Q_{heat}$$

where α_p is the (isobaric) coefficient of thermal expansion, ρ is the density, C_p is the heat capacity at constant pressure, and Q_{heat} is the heat source.

MODEL INPUT

Enter the ambient **Temperature** T, the default is `293.15[K]`.

HEAT SOURCE

Enter the **Heat Source** Q_{heat} (SI unit: W/m^3).

FLUID PROPERTIES

Enter the **Heat capacity at constant pressure** C_p and the **Ratio of specific heats** γ either selected **From material** or **User defined**. The ratio of specific heats is used to calculate the coefficient of thermal expansion through the thermodynamic relation

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

Volume Force Source

The **Volume Force Source** sets up the right-hand side of the momentum equation, in Equation 2-2, in terms of a volumetric force. This is a dipole-like source that acts differently in two directions.

$$\mathbf{q}_d = \mathbf{F}$$

VOLUME FORCE SOURCE

Enter the **Volume force source** $\mathbf{F}$ (SI unit: N/m^3).

Sound Hard Boundary (Wall)

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

$$-\mathbf{n} \cdot \mathbf{u} = 0$$

Note that this condition is not identical to the Symmetry condition. The condition is also available from the **Pairs** submenu, as an option at interfaces between parts in an assembly. This effectively introduces a condition similar to the Interior Sound Hard Boundary (Wall) but used on an assembly.

Continuity of Total Fields

The **Continuity of Total Fields** feature is a default feature. The feature automatically ensures continuity of the total acoustic fields when necessary. The feature becomes active when a Background Acoustic Field feature is applied on only parts of the model domains. Note that if two different materials are present in the model the Material Discontinuity feature needs to be applied.

Sound Soft Boundary

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

Pressure

The **Pressure** node creates a boundary condition that acts as a pressure source at the boundary. The total acoustic pressure is given at the boundary $p_t = p_0(t)$.

PRESSURE

Enter an expression for the **Pressure** $p_0(\mathbf{x}, t)$ (SI unit: Pa). The input can be any analytical function or interpolation data that can be function of both time and space.

If the **Use accelerated solver formulation** option is selected in Model Equations and Solver Settings section, the source has to be split into its spatial and temporal

components such that $p_t = p_0(\mathbf{x})\, l(t)$. Enter an expression for the (space-dependent) **Pressure** $p_0(\mathbf{x})$ and the time-dependent **Load factor** $l(t)$.

Symmetry

The **Symmetry** node adds a boundary condition imposing symmetry in the pressure and velocity field. Use this condition to reduce the size of a model by cutting it in half where there are known symmetries. The condition implies symmetry in the acoustic fields

$$\mathbf{n} \cdot \mathbf{u}_t = 0$$

Normal Velocity

The **Normal Velocity** condition adds a source specified in terms of an inward normal velocity $v_n(t)$ or velocity $\mathbf{v}_0(t)$ of the boundary. The component in the normal direction is used to define the boundary condition. This feature represents an external source term like a vibrating surface. The condition is also available from the **Pairs** submenu, as an option at interfaces between parts in an assembly. This effectively introduces a condition similar to the Interior Normal Velocity but used on an assembly.

NORMAL VELOCITY

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

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

If the **Use accelerated solver formulation** option is selected in Model Equations and Solver Settings section, the source has to be split into its spatial and temporal components so that the inward velocity is $-v_n(\mathbf{x})\, l(t)$. Enter an expression for the (space-dependent) **Inward velocity** $v_n(\mathbf{x})$ (or the **Velocity**) and the time-dependent **Load factor** $l(t)$.

Impedance

The **Impedance** condition adds a boundary condition with the option to select between several built-in impedance models that allow modeling frequency dependent impedance conditions in the time domain. The impedance condition defines a relation between the local total acoustic pressure p_t and the normal acoustic velocity $\mathbf{n}\cdot\mathbf{u}_t$, that

is, the (normal) specific acoustic impedance Z_n (SI unit Pa·s/m). The condition specifies the normal velocity

$$\mathbf{n} \cdot \mathbf{u}_t = -\frac{p_t}{Z_n}$$

The **Impedance** boundary condition is a good approximation of a locally reacting surface — a surface for which the normal velocity at any point depends only on the pressure at that exact point. This condition can be used to model the properties of artificial boundaries like a wall treatment in room acoustics. When the impedance is set equal to the characteristic specific impedance of a propagating plane wave ρc, the condition represents the simplest nonreflecting boundary condition. This is also the default value of the impedance when the condition is added.

In many practical applications the surface impedance is known (measured or simulated) in term of its real and imaginary part as function of frequency. To capture this behavior in the time domain the **Impedance** condition has built-in functionality for setting up a General local reacting (rational approximation) impedance through a rational approximation. Fitting of the frequency domain data can be performed using the Partial Fraction Fit function.

	The acoustic impedance condition (with the default value ρc) should be used at the outer boundary when Absorbing Layers are used in a model.

	Wave-Based Time-Domain Room Acoustics with Frequency-Dependent Impedance. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/wave_based_room**

	For more information see Partial Fraction Fit under User-Defined Functions in the *COMSOL Multiphysics Reference Manual*.

	For details on how to best fit data, see Using the Partial Fraction Fit for Acoustic Data.

IMPEDANCE

Select the **Impedance model** as **User defined (resistive)** (the default), **Serial coupling RCL**, or **General local reacting (rational approximation)**. The different options are as follows:

User defined (resistive)

For the **User defined (resistive)** option represents a purely resistive constant impedance. The values entered here can only be real valued and constant in time.

Enter a value or expression for the **Specific impedance** Z_n (SI unit: Pa·s/m). The default expression is `pate.c*pate.rho` which is the characteristic specific impedance ρc of a propagating plane wave.

Serial coupling RCL

The **Serial coupling RCL** option defines a simple spring-mass-damper system in the form of a serially coupled RCL circuit. This option can be used to approximate many systems that exhibit a simple damped resonant characteristic. The RCL condition will exhibit a simple frequency dependent behavior.

Enter values for the **Equivalent acoustic resistance** R_{ac} (SI unit: kg/(m^4·s)), the **Equivalent acoustic compliance** C_{ac} (SI unit: m^4·s^2/kg), and the **Equivalent acoustic inertance** L_{ac} (SI unit: kg/m^4).

General local reacting (rational approximation)

The **General local reacting (rational approximation)** option offers the most general and flexible form for defining a frequency dependent impedance condition in the time domain. The condition relies on a rational approximation (or rational expansion) of the frequency dependent admittance $Y(f)$. This approximation has an analytical inverse Fourier transform and can thus be used to define the same frequency dependency in the time domain, by setting up a system of memory ODEs. These ODEs are automatically defined by the feature and solved on the impedance boundary. Approximating the frequency-dependent admittance data can be performed by using the built-in Partial Fraction Fit function.

Select the **Partial fraction** fit as **User defined** (the default) or **From function**.

- For the **User defined** option, enter all the necessary parameters that are present in the rational approximation (rational expansion), that is, the **Frequency-independent (asymptotic) admittance** Y_∞ (SI unit: m^2·s/kg), enter the **Real residues and poles** (R and ξ) into the table, and enter the **Complex residues and poles** (Q and ζ) into the table (note that the conjugate terms are automatically added). The quantities can also be loaded from a file (remember to include the number column). Select if the

rational approximation function is scaled for frequency data or angular frequency data by selecting the **Equation scaled for frequency data** option (selected per default).

- For the **From function** option select the **Reference**, which is a Partial Fraction Fit function defined in the model, then click the **Import** icon ( ) to copy the fitted data to the admittance field, and the residues and poles tables. To go to the function source click the **Go to Source** icon ( ).

The frequency-dependent admittance and thus its partial fraction approximation has to fulfill the following conditions for the results to be physically possible:

- Causality, $Y(f)$ is analytic and nonzero in $\Im(\omega) > 0$;
- Reality, $\overline{Y}(f) = Y(-f)$, that is, $\Re(Y(f))$ is even and $\Im(Y(f))$ is odd;
- Passivity, $\Re(Y(f)) > 0$ for all real f.

For more details see Using the Partial Fraction Fit for Acoustic Data.

The frequency-dependent admittance data, fitted by the **Partial Fraction Fit** function, can in general be extracted from a submodel of the boundary, for example, a model of a porous layer, solved in the frequency domain. Specifically the data can, for a subset of wall/boundary configuration, be computed using the *Acoustic Treatment Boundary Calculator* application in the Acoustics Module's Application Library.

Exterior Field Calculation

The **Exterior Field Calculation** feature adds the equations necessary to postprocess the exterior field to the computational domain for open problems. The feature is the same as the Exterior Field Calculation in The Pressure Acoustics, Frequency Domain Interface. This feature will only generate variables and results that can be used in postprocessing when the results of a time-dependent simulation have been transformed to the frequency domain using a **Time to Frequency FFT** study.

For more detailed information about the FFT study see the Time-to-Frequency FFT section in the *COMSOL Multiphysics Reference Manual*.

The feature only operates on the scattered field variables and is thus especially well suited for analyzing the results of a scattering problem. Such a model can be set up by using the Background Acoustic Field feature combined with the Absorbing Layers. Since the time explicit formulation is very memory lean, scattering problems that cannot be solved in the frequency domain can be handled with a time domain approach and an FFT.

	Submarine Scattering: Time Domain Simulation and FFT: In the COMSOL Application Gallery https://www.comsol.com/model/submarine-scattering-time-domain-simulation-and-fft-55921

Interior Sound Hard Boundary (Wall)

The **Interior Sound Hard Boundary (Wall)** node, found under the **Interior Conditions** submenu, adds the boundary condition for a sound hard boundary or wall on interior boundaries. The condition reads

$$\mathbf{u}_{\mathrm{t,up}} \cdot \mathbf{n} = 0$$
$$\mathbf{u}_{\mathrm{t,down}} \cdot \mathbf{n} = 0$$

The condition ensures that the total normal velocity is zero on both the up and down side of the boundary, while applying a slit on the pressure (the pressure is discontinuous across the boundary).

Interior Normal Velocity

The **Interior Normal Velocity** node, found under the **Interior Conditions** submenu, adds the boundary condition for a moving interior boundary. This can, for example, represent a thin vibrating structure or s speaker membrane. The condition reads

$$\mathbf{u}_{\mathrm{t,up}} \cdot \mathbf{n} = \mathbf{v}_0(t) \cdot \mathbf{n} = -v_{\mathrm{n}}(t)$$
$$\mathbf{u}_{\mathrm{t,down}} \cdot \mathbf{n} = \mathbf{v}_0(t) \cdot \mathbf{n} = -v_{\mathrm{n}}(t)$$

The condition ensures that the total normal velocity is defined on both the up and down side of the boundary, while applying a slit on the pressure (the pressure is discontinuous across the boundary).

NORMAL VELOCITY

Select the **Type**: **Velocity** (default) or **Normal velocity** and specify either the **Velocity** (vector) $\mathbf{v}_0$ or the normal component through the **Normal velocity** v_n.

Interior Impedance

The **Interior Impedance** node, found under the **Interior Conditions** submenu, adds an interior boundary condition to prescribe a transfer impedance between two domains. The condition defined a relation between the local total acoustic pressure jump $p_{t,up} - p_{t,down}$ (difference between upside and downside of the boundary) and the normal acoustic velocity $\mathbf{n} \cdot \mathbf{u}_t$, that is, the specific acoustic transfer impedance Z_n. The condition also prescribes continuity in the normal velocity across the boundary. The condition reads

$$\mathbf{n} \cdot \mathbf{u}_t = -\frac{p_{t,down} - p_{t,up}}{Z_n}$$

This condition can be used to model the properties of artificial boundaries by adding constant resistive losses (no reactive components can be added).

A pair version of this feature the **Pair Impedance** exists for use in geometry assemblies on pair. In this case, the pressure jump is computed from the source (`src`) to the destination (`dst`).

INTERIOR IMPEDANCE

Enter a value or expression for the **Specific transfer impedance** Z_n (SI unit: Pa·s/m). The default expression is `pate.c*pate.rho` which is the characteristic specific impedance ρc of a propagating plane wave.

Material Discontinuity

The **Material Discontinuity** node, found under the **Interior Conditions** submenu, adds an interior boundary condition to handle jumps in material properties. As the interface is based on the discontinuous Galerkin method special handling is necessary when a jump exists in the specific characteristic impedance $Z_0 = \rho c$ of the medium.

The condition implements continuity in total pressure and in the normal velocity over the interior boundary:

$$p_{\text{t, up}} = p_{\text{t, down}}$$
$$\mathbf{n} \cdot \mathbf{u}_{\text{t, up}} = \mathbf{n} \cdot \mathbf{u}_{\text{t, down}}$$

Subscripts “up” and “down” in the equation refer to the up and down sides of the interior boundary, relative to the normal $\mathbf{n}$.

Continuity

The **Continuity** node, exists as an option at interfaces between parts in an assembly, from the **Pairs** submenu. The condition is similar to the Material Discontinuity condition that applies to interior boundaries. The difference is that the **Continuity** condition should also be applied when the same material exists on both sides of a pair. The advantage of using a pair feature is that the mesh does not need to be conforming on the two sides of the interface (the two parts pf the assembly). This is especially advantageous for the time explicit discontinuous Galerkin method as the time step depends on mesh size and local speed of sound.

The condition implements continuity in total pressure and in the normal velocity over the pair:

$$p_{\text{t, src}} = p_{\text{t, dst}}$$
$$\mathbf{n}_{\text{src}} \cdot \mathbf{u}_{\text{t, src}} = -\mathbf{n}_{\text{dst}} \cdot \mathbf{u}_{\text{t, dst}}$$

Subscripts “src” and “dst” in the equation refer to the source and destination in the pair; that is, the two sides in the pair.

For more details on using assemblies and the pair features see Assemblies and Pair Conditions in the Modeling with the Pressure Acoustics Branch (DG-FEM-Based Interface) section.

General Flux/Source

The **General Flux/Source** node, found under the **More** submenu, adds the most general boundary condition for the Pressure Acoustics, Time Explicit model, as it is formulated (in the conservative form) for the discontinuous Galerkin method (dG). The condition defines the normal flux g_{p} and $\mathbf{g}_{\text{u}}$ at an exterior boundary, given by

$$-\mathbf{n} \cdot \Gamma_p = \mathbf{n} \cdot \mathbf{u} = g_p$$

$$-\mathbf{n} \cdot \Gamma_u = \begin{bmatrix} n_1 p \\ n_2 p \\ n_3 p \end{bmatrix} = \mathbf{g}_u$$

where $\mathbf{u} = (u, v, w)$ and $\mathbf{n} = (n_x, n_y, n_z)$ in 3D and 2D, or $\mathbf{n} = (n_r, n_\varphi, n_z)$ in 2D axisymmetric. Note that this condition is expressed in terms of the dependent variables and not the total fields. Care should be taken when using this advanced condition as the method is sensitive to proper formulation of boundary conditions. Only one characteristic can enter the domain at any boundary at the time, meaning that it is easy to overspecify the problem. Use the mesh normals (`nxmesh`, `nymesh`, and `nzmesh`; or `nrmesh`, `nphimesh`, and `nzmesh`) in the expression you define.

GENERAL FLUX/SOURCE

Enter the expression for the components of the **Flux vector** g_p and $\mathbf{g}_u$.

General Interior Flux

The **General Interior Flux** node, found under the **More** submenu, adds the most general interior boundary condition for the Pressure Acoustics, Time Explicit model, as it is formulated (in the conservative form) for the discontinuous Galerkin method (dG). The condition defines the normal flux g_p and $\mathbf{g}_u$ at an interior boundary. It should include any necessary numerical flux (as it overrides the Lax–Friedrichs flux). Care should be taken when using this advanced condition as the method is sensitive to proper formulation of boundary conditions.

GENERAL INTERIOR FLUX

Enter the expression for the components of the **Flux vector** g_p and $\mathbf{g}_u$.

The Pressure Acoustics, Asymptotic Scattering Interface

The **Pressure Acoustics, Asymptotic Scattering (paas)** interface ( ), found under the **Acoustics** > **Pressure Acoustics** branch ( ) when adding a physics interface, is used to model scattering problems at high frequencies using the Kirchhoff–Helmholtz integral formulation. The acoustic field is assumed to be locally plane such that the scattered field can be expressed analytically. The scattering object has to be acoustically large; the wavelength should be much smaller than important geometric features as well as the radius of curvature of important features, see Ref. 70, 71, and 72. The scattering analysis is only valid for open domains. The harmonic variation of all fields and sources is given by $e^{i\omega t}$ using the $+i\omega$ convention.

The scattering object surface can be treated as perfectly reflecting or having absorbing properties by defining a surface normal impedance, a reflection coefficient, or an absorption coefficient. The latter two can depend on the angle of incidence. The interface can model scattering from spherical and plane waves in 3D and 2D axisymmetry, as well as plane and cylindrical waves in 2D. The interface has built-in functionality to compute the visibility factor.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Pressure Acoustics**, **Background Pressure Field**, **Scattering Object** (with **Wall** subfeature), and **Exterior Field Calculation**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and source. You can also right-click **Pressure Acoustics, Asymptotic Scattering** to select physics features from the context menu.

	Domain and Boundary Nodes for the Pressure Acoustics, Asymptotic Scattering Interface

	Submarine High-Frequency Asymptotic Scattering. Application Library path **Acoustics_Module/Underwater_Acoustics/ submarine_asymptotic_scattering**

SETTINGS

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

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

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

SOUND PRESSURE LEVEL SETTINGS

The settings selected here are only used if the transient solution solved is transformed into the frequency domain using the **Time to Frequency FFT** study. The zero level on the dB scale varies with the type of fluid. That value is a reference pressure that corresponds to 0 dB. This variable occurs in calculations of the sound pressure level L_{p} based on the root mean square (rms) pressure p_{rms}, such that

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

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

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

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

VISIBILITY

The high frequency assumption used in the asymptotic scattering interface requires the computation of the visibility. That is the surface of the scattering object "illuminated" by the background (incident) acoustic field. Select the **Method** to compute the visibility either **Angle of incidence** (the default) or **Hemicube**.

- **Angle of incidence**: COMSOL Multiphysics evaluates the angle of incidence between the background field and the surfaces directly, without considering which face elements are obstructed by others. This means that shadowing effects (that is,

surface elements being obstructed in nonconvex cases) are not taken into account. Elements facing away from each other are, however, excluded from the integrals. The **Angle of incidence** method is fast and accurate for simple geometries with no or little shadowing.

- **Hemicube**: The more sophisticated and general hemicube method uses a z-buffered projection on the sides of a hemicube (with generalizations to 2D and 1D) to account for shadowing effects. Think of it as rendering digital images of the geometry in five different directions (in 3D; in 2D only three directions are needed), and counting the pixels in each mesh element to evaluate its view factor.

  Its accuracy can be influenced by setting the **Radiation resolution** of the virtual snapshots. The number of z-buffer pixels on each side of the 3D hemicube equals the specified resolution squared. Thus the time required to evaluate the irradiation increases quadratically with resolution. In 2D, the number of z-buffer pixels is proportional to the resolution property, and thus the time is, as well. Visualizing results will be slower when the **Hemicube** option is selected.

DISCRETIZATION

In this section, you can select the discretization or element order used for resolving the pressure and its normal derivative at the scattering surfaces. No dependent variable is solved by the interface, but the representation of curved surfaces is influenced by this setting as well as the number of elements needed to resolve the problem.

Domain and Boundary Nodes for the Pressure Acoustics, Asymptotic Scattering Interface

The Pressure Acoustics, Asymptotic Scattering Interface has the following domain and boundary conditions listed in alphabetic order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

- Background Pressure Field
- Exterior Field Calculation
- Pressure Acoustics
- Scattering Object
- Wall

Pressure Acoustics

The **Pressure Acoustics** node defines the properties of the fluid surrounding the scattering object. This is a default feature. The selection of the feature contains all domains and voids. Select a **Fluid Model** following the same options as the Pressure Acoustics node in The Pressure Acoustics, Frequency Domain Interface.

Background Pressure Field

The **Background Pressure Field** node defines the background or incident pressure field that is scattered off the Scattering Object. The **Pressure field type** can be selected as **Plane wave** or **Spherical wave**. Set the wave amplitude and direction, see the Background Pressure Field node in The Pressure Acoustics, Frequency Domain Interface.

Several fields can be defined for computing the scattering from a complex incident sound field. The surface properties defined on the Wall will take the effects of each background field into account.

Scattering Object

The **Scattering Object** node defines the domains and/or finite voids that make up the scattering object(s). This is a so-called singleton feature that is only added once (per default) to the model. Note that secondary scattering effects are not taken into account if several not connected objects are selected, that is, the scattered field of one object is not scattered of the other objects. The properties of the scattering object surfaces are defined in the Wall subnode.

Wall

The **Wall** subnode defines the surface properties of the scattering object. Several **Wall** subfeatures can be added to represent different properties of the boundaries of the surface. Select the **Type** as **Sound hard wall** (the default), **Reflection coefficient**, **Normal impedance**, or **Absorption coefficient**.

- For **Sound hard wall**, there is perfect specular reflection with no absorption of energy at the boundary. The condition corresponds to a reflection coefficient $R = 1$.
- For **Reflection coefficient**, enter a value for R (SI unit 1). The reflection coefficient is a complex valued quantity that can depend on the angle of incidence θ_i. The angle is defined through the built-in variable `paas.theta`.

- For **Normal impedance**, enter a value for the normal impedance Z_n (SI unit: Pas/m). The reflection coefficient of the surface is computed according to the classical plane wave reflection expression

$$R = \frac{Z_n \cos(\theta_i) - Z}{Z_n \cos(\theta_i) + Z}$$

where $Z = \rho c$ is the specific characteristic acoustic impedance of the medium and θ_i is the (local) angle of incidence. This corresponds to a locally reacting surface. The normal impedance is typically complex valued.

- For **Absorption coefficient**, enter a value for the (normal) absorption coefficient α_n (SI unit: 1) and the phase ϕ (SI unit: rad). Both can depend on the angle of incidence in the same way as the reflection coefficient. The reflection coefficient of the surface is computed according to

$$R = \sqrt{1-\alpha}\ e^{i\phi}$$

Angle dependency of the absorption or reflection coefficient can be introduced using the angle if incidence variable `paas.theta` or the cosine to the angle of incidence `paas.costheta`.

Exterior Field Calculation

The **Exterior Field Calculation** node is the feature that sets up the Kirchhoff–Helmholtz integral to perform the postprocessing of the scattered field. It is per default added to all the scattering boundaries. The selection is not locked so it can be modified but it should in general coincide with all the Wall boundaries of the Scattering Object. Only one **Exterior Field Calculation** feature can be added to the interface.

Enter a name for the **Exterior field variable** name and select the **Type of integral**. See the Exterior Field Calculation node in The Pressure Acoustics, Frequency Domain Interface. Note that no symmetry options exist for this interface. The **Advanced Settings** section is also not present in this physics interface.

The Pressure Acoustics, Kirchhoff–Helmholtz Interface

The **Pressure Acoustics, Kirchhoff–Helmholtz (pakh)** interface ( ), found under the **Acoustics** > **Pressure Acoustics** branch ( ) when adding a physics interface, is used to model radiation problems at high frequencies using the Kirchhoff–Helmholtz integral formulation. The acoustic field is assumed to be locally plane. This technique is also often referred to as high frequency BEM, HFBEM, or simply HFB. The method is often used for computing the radiated acoustic field from vibrating structures at high frequencies, without the need to model the surrounding fluid. The method is valid as long as the acoustic wavelength in the fluid is smaller then the structure and the structural modes. The method is only applicable for open radiation problems. For a flat vibrating surface the method reduces to solving the Rayleigh integral. A user-defined option gives access to the full Kirchhoff–Helmholtz integral formulation for defining both pressure and its normal gradient. The harmonic variation of all fields and sources is given by $e^{i\omega t}$ using the $+i\omega$ convention.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Pressure Acoustics**, **Radiating Object** (with **Wall** subfeature), and **Exterior Field Calculation**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and source. You can also right-click **Pressure Acoustics, Asymptotic Scattering** to select physics features from the context menu.

Domain and Boundary Nodes for the Pressure Acoustics, Asymptotic Scattering Interface

SETTINGS

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

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

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

SOUND PRESSURE LEVEL SETTINGS

The settings selected here are only used if the transient solution solved is transformed into the frequency domain using the **Time to Frequency FFT** study. The zero level on the dB scale varies with the type of fluid. That value is a reference pressure that corresponds to 0 dB. This variable occurs in calculations of the sound pressure level L_p based on the root mean square (rms) pressure p_{rms}, such that

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

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

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

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

DISCRETIZATION

In this section, you can select the discretization or element order used for resolving the pressure and its normal derivative at the scattering surfaces. No dependent variable is solved by the interface, but the representation of curved surfaces is influenced by this setting as well as the number of elements needed to resolve the problem.

Domain and Boundary Nodes for the Pressure Acoustics, Asymptotic Scattering Interface

The Pressure Acoustics, Kirchhoff–Helmholtz Interface has the following domain and boundary conditions listed in alphabetic order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

- Exterior Field Calculation
- Pressure Acoustics
- Radiating Object
- Wall

Pressure Acoustics

The **Pressure Acoustics** node defines the properties of the fluid surrounding the scattering object. This is a default feature. The selection of the feature contains all domains and voids. Select a **Fluid Model** following the same options as the Pressure Acoustics node in The Pressure Acoustics, Frequency Domain Interface.

Radiating Object

The **Radiating Object** node defines the domains and/or finite voids that make up the radiating object(s). This is a so-called singleton feature that is only added once (per default) to the model. Note that scattering effects are not taken into account. The sources associated with the radiating object (pressure or velocity) are defined in the Wall subnode.

Wall

The **Wall** subnode defines the surface properties of the radiating object. That is, how the surface sources are defined. Several **Wall** subfeatures can be added to represent different properties of the boundaries of the surface. Note that the term "wall" here is very generic as it can also represent, for example, an acoustic source boundary. Select the **Type** as **Velocity** (the default), **Normal velocity**, or **User defined**.

- For **Velocity**, enter the components of the vibrating surface velocity $\mathbf{v}$. This could be the velocity that results from modeling a vibrating surface like the hull of a ship or and engine. For example, if the vibrating object is modeled by a Solid Mechanics interface, then enter the velocities as `solid.u_tX`, `solid.u_tY`, and `solid.u_tZ`, respectively.
- For **Normal velocity**, enter the normal velocity v_n (SI unit: m/s). This is the normal velocity pointing into the infinite fluid domain. This can be a constant, an expression that depends on space coordinates, or it can be taken from the solution of another model.
- For **User defined**, enter the pressure p (SI unit: Pa) and t the normal velocity v_n (SI unit: m/s) defined at the given boundary. This option gives full user control of the input to the Kirchhoff–Helmholtz integral that is set up by the Exterior Field Calculation feature.

Exterior Field Calculation

The **Exterior Field Calculation** node is the feature that sets up the Kirchhoff–Helmholtz integral to perform the postprocessing of the radiated field under the high frequency assumption. It is per default added to all the source boundaries. The selection is not locked, so it can be modified but it should in general coincide with all the Wall boundaries of the Radiating Object. Only one **Exterior Field Calculation** feature can be added to the interface.

For the settings see the Exterior Field Calculation node in The Pressure Acoustics, Frequency Domain Interface. Note that the **Advanced Settings** section is not present in this physics interface.

Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces)

This section contains good practices and modeling tips and tricks for the finite element method (FEM) based interfaces of the Pressure Acoustics Branch: The Pressure Acoustics, Frequency Domain Interface, The Pressure Acoustics, Transient Interface, and The Pressure Acoustics, Boundary Mode Interface. In this section:

- Physics-Controlled Mesh for Pressure Acoustics
- Meshing: Resolving the Waves in Space
- Lagrange and Serendipity Shape Functions
- Time Stepping: Resolving the Waves in Time
- Frequency Domain, Modal and Adaptive Frequency Sweep (AWE)
- Solving Large Acoustics Problems Using Iterative Solvers
- Perfectly Matched Layers (PMLs)
- Postprocessing Variables
- Evaluating the Acoustic Field in the Exterior: Near- and Far-Field
- Dedicated Acoustics Plots for Postprocessing
- About the Material Databases for the Acoustics Module
- Specifying Frequencies: Logarithmic and ISO Preferred

Physics-Controlled Mesh for Pressure Acoustics

When modeling acoustic problems using the Pressure Acoustics, Frequency Domain or the Pressure Acoustics, Transient interfaces the computational mesh can be automatically set up using the **Physics-controlled mesh** functionality. For these two physics, the generated computational mesh will follow best practices. The mesh will automatically resolve the wave, perfectly matched layer (PML) domains will have an appropriate mapped mesh, a boundary layer mesh is added for the exterior field, and periodic boundaries will be set up with copy mesh operations.

- First, select the **Sequence Type** as **Physics-controlled mesh** (the default) or **User-controlled mesh**, see Figure 2-8. When the user-controlled option is selected the mesh sequence nodes, generated by the physics, are shown and can be modified.

For a fully user-defined mesh proceed and directly right-click the **Mesh** node, in the **Model Builder** tree, and add the first desired mesh element. The physics that contribute to the physics-controlled mesh generation can be selected in the **Contributor** list.

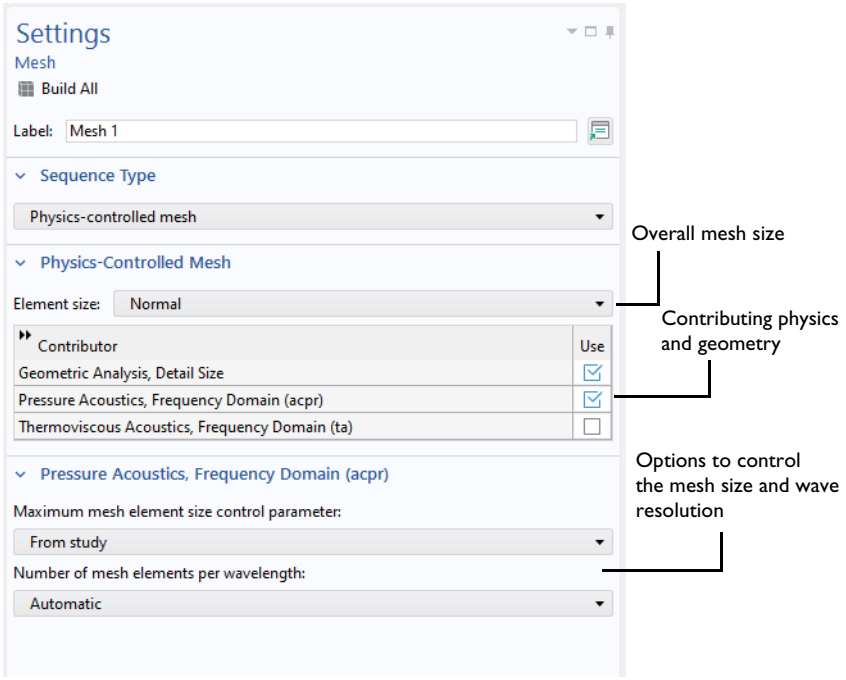


Figure 2-8: User interface of the Physics-controlled mesh options for Pressure Acoustics as seen on the Mesh node.

Resolving the wave in space (see also Meshing: Resolving the Waves in Space) requires and input of the maximum frequency solved for in the model. The way the frequency is picked up and how many elements are used to resolve the wave, can be controlled in the physics-controlled mesh user interface, see Figure 2-8.

- Select the **Maximum mesh element size control parameter** as **From study** (the default) or **Frequency**. For the default **From study** option, the maximum frequency will, for frequency domain problems, be picked up from the **Frequencies** list in the frequency domain study step; while for time domain problems the value is picked up from the **Maximum frequency to resolve** setting in the Transient Solver and Mesh Settings section. In the frequency domain case, enter frequency values in the study step before generating the mesh. Alternatively, select the **Frequency** option and manually

enter the **Maximum frequency**. The manual input can be a good option in models with several studies to ensure that the mesh is the same for all.

- Select the **Number of mesh elements per wavelength** as **Automatic** (the default) or **User defined**. The **Automatic** option gives 5 elements per wavelength.

The mesh also needs to resolve the geometry details in the model. This is done automatically by having the **Geometric Analysis, Detail Size** contribution selected in the **Physics-Controlled Mesh** settings, see Figure 2-8.

The overall refinement such as minimum mesh size, maximum element growth rate, curvature factor, and the resolution of narrow region values are controlled by the **Element size** option. The default is to use the **Normal** setting.

If a model contains a **Parametric Sweep**, the physics-controlled mesh will be regenerated for each new parameter value. This is of course important, if the parameter changes the geometry or the minimum wavelength in the model. For example, if a sweep is done over a number of frequency bands, a proper mesh should be regenerated for each band.

In cases where the parameter has no influence on the mesh, remeshing will still take place, and may slow down the solution process. In this case it is advised to instead use an **Auxiliary Sweep** (an inner sweep) found in the **Study Extensions** section.

- *Acoustic Scattering off an Ellipsoid*. Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/acoustic_scattering**
- *Absorptive Muffler*. Application Library path **Acoustics_Module/Automotive/absorptive_muffler**

See Meshing and The Mesh Node in the *COMSOL Multiphysics Reference Manual*

Meshing: Resolving the Waves in Space

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

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

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

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

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

RESOLVING WAVES WITH LAGRANGE AND SERENDIPITY ELEMENTS

When creating an unstructured mesh for use with the default 2nd-order Lagrange elements or 2nd-order serendipity elements (see Lagrange and Serendipity Shape Functions), set the maximum element size $h_{\max}$ to about $\lambda/5$ or smaller. Note that this choice is the default for the Physics-Controlled Mesh for Pressure Acoustics. Because all elements in the constructed mesh are smaller than $h_{\max}$, the limit is set larger than the actual required element size. Note that for specific engineering purposes it is sometimes possible to use a coarser mesh. This is, for example, the case if only the sound pressure level (SPL) is of interest as a result, and not the exact phase of the pressure (correct balance between real and imaginary part of the pressure variable in the frequency domain). In general a mesh sensitivity analysis should be carried out to investigate the sensitivity on the solutions parameters of interest. The mesh should also resolve important geometric features and possible gradients in the material parameters and model inputs. Geometric features are, for example, curved surfaces or narrow gaps which need to be resolved adequately. Another important example is if a thin structure is present (possibly modeled as a boundary using an interior condition). In this case, it

is important to have a fine mesh near the edge of the structure to resolve gradients in the pressure.

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

Meshing in the *COMSOL Multiphysics Reference Manual*

MESHING THE PERFECTLY MATCHED LAYER (PML)

When using a perfectly matched layer (PML) to truncate the computational domain, it is good practice to use a structured mesh inside the PML region. In 3D models, use a **Swept** mesh inside the PML and in 2D models use a **Mapped** mesh. Use at least 5 elements in the thickness when using rational PML scaling and 8 elements when using the default polynomial scaling in the PML. The automatic Physics-Controlled Mesh for Pressure Acoustics sets the suggested number of layers. To verify the mesh, it is good practice to check using a mesh convergence by adding more layers to the PML.

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

Structured Meshes in the *COMSOL Multiphysics Reference Manual*

Lagrange and Serendipity Shape Functions

In most of the physics interfaces in the Acoustics Module and specifically in the Pressure Acoustics interfaces, you can choose between two families of shape functions: *Lagrange* and *serendipity*. The current default is to use Lagrange shape functions.

When using a structured mesh, it may be advantageous to switch to the serendipity elements as they generate significantly fewer degrees of freedom (DOFs). The accuracy

is in most cases almost as good as for the Lagrange elements. The Lagrange elements are however less sensitive to strong mesh distortions.

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

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

In the *COMSOL Multiphysics Reference Manual*:

- The Lagrange Element (shlag)
- The Nodal Serendipity Element (shnserp)

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

Time Stepping: Resolving the Waves in Time

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

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

The logic for the automatic choice made is as follows. The mesh resolution imposes a restriction on the time-step size Δt taken by the solver. The relationship between mesh

size and time-step size is closely related to the Courant number given by the Courant–Friedrichs–Lewy (CFL) condition (Ref. 34), which is defined as

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

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

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

Time Dependent in the *COMSOL Multiphysics Reference Manual.*

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

The condition on the Courant number is, for a time-implicit method, not necessary for stability reasons of the method, but it is used to give a good estimate for accuracy. Basically, you want to ensure the same numerical resolution in time and space.

DEFINING A GOOD TIME STEP FOR A MODEL WITH PROPER SPATIAL RESOLUTION

When the excitation contains all the frequencies the mesh can resolve, there is no point in using an automatic time-step control which can be provided by the time-dependent solver (the **Free** option). The tolerances in the automatic error control are difficult to tune when there is weak but important high-frequency content. Instead, you can use your knowledge of the typical mesh size, speed of sound, and Courant number to calculate and prescribe a fixed time step. This is exactly the default behavior when the **Fixed (preferred)** method is chosen in the **Transient Solver and Mesh Settings** section. The **Free** option corresponds to the automatic time-step control but with some tighter

controls of the allowed time steps. This latter option is still in general not recommended as the fixed time stepping option typically yields much better results (and is faster).

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

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

Assuming that N is between 5 and 6 and the Courant number is roughly 0.1, these values give a good margin of safety. To check that the accuracy is acceptable, it is recommended that you run a short sequence of typical excitations with progressively smaller time steps (larger $f_{\max}$) and check the convergence.

If a **Multiphysics** coupling is used to couple, for example, acoustics with structural interfaces, the correct time-stepping method will be used and it will be based on the settings in the acoustics interface.

When several acoustics interfaces are present in a model, the logic for the time step is to use the minimal time step dictated by the contributing physics.

Some physics, like *Electrostatics* (present when modeling piezoelectric problems), suggest a BDF time stepping method to be used. If this physics is present, the BDF method will be used but the maximal time steps will be taken from the acoustics suggestions. In this case go to the **Time-Dependent Solver** node and change the **Method** in the **Time Stepping** section to **Generalized alpha**. All other settings will be correct.

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

Frequency Domain, Modal and Adaptive Frequency Sweep (AWE)

When solving models in the frequency domain and performing a large frequency sweep, it can sometimes be useful to use the **Frequency Domain, Modal** study or the

Adaptive Frequency Sweep study steps. Both approaches can significantly speed up the solution time when sweeping over many frequencies.

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

- The modal solver requires that the eigenfrequencies and modes can be easily found for up to twice the maximal study frequency. If a model contains many resonances in the desired frequency range, this may not be a good idea. It is often necessary to include the DC (static) pressure mode also to get the correct low frequency behavior.
- Note that since the modal solver assumes linear perturbation, all sources should be declared using the `linper()` operator.

For the **Adaptive Frequency Sweep**:

- This approach relies on the asymptotic waveform expansion or AWE approach. The **Adaptive Frequency Sweep** study is a special version of the regular **Frequency Domain** study.
- For this study add the AWE Expression feature to define the necessary AWE expression/metric for the solver.
- The AWE option can also be added under the **Study Extensions** section in a regular **Frequency Domain** study. In this case, it is necessary to enter an **AWE expression** manually. This is an expression that represents the response of the system. It needs to be a global expression. This can for example be the sound pressure level (SPL) evaluated using an integration or average operator. Integration in a point will give the SPL in that point.

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

See also the sections Frequency Domain Study and Frequency Domain, Modal and Time-Dependent, Modal Studies in the Study Types chapter.

Solving Large Acoustics Problems Using Iterative Solvers

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

- Direct Solver Tuning
- Automatically Generated Iterative Solver Suggestions
- Complex Shifted Laplacian for Very Large Frequency Domain Models
- Domain Decomposition for Helmholtz on Clusters
- Efficient Iterative Solver with Cubic Discretization and Two Multigrid Levels
- Manual Setup of GMG Solver Suggestions and Theory

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

DIRECT SOLVER TUNING

When using the default direct solver MUMPS, one option to reduce computation time and decrease the memory consumption, is to enable the option **Block low rank factorization** on the **Direct** solver node. This can for many pressure acoustics problems reduce both memory and computation time by about 20% to 25%. This option is not default, as it is not efficient for all types and combinations of boundary conditions. For example, it is not suited for models with nonlocal couplings, like models with periodic conditions.

For further details see the Direct section in the *COMSOL Multiphysics Reference Manual*.

For an example that uses the **Block low rank factorization** see: *Test Bench Car Interior*. Application Library path **Acoustics_Module/Automotive/test_bench_car_interior**

AUTOMATICALLY GENERATED ITERATIVE SOLVER SUGGESTIONS

If the direct solver runs out of memory, a simple first approach is to enable and use one of the auto generated iterative solver suggestions. A good starting point for this is to right-click the study node and select **Show Default Solver**, then expand the **Solver Configuration** tree under **Stationary Solver** or **Time-Dependent Solver**. Predefined iterative solver suggestions are automatically generated. Per default, for a pure *Pressure Acoustics, Frequency Domain* model, a direct solver is used and four iterative solvers are suggested (disabled). To turn on one of these, right-click the solver and select **Enable** (or press F4). The four suggestions are:

1. *Suggested Iterative Solver (GMRES with GMG)*: uses the GMRES iterative solver with a geometric multigrid (GMG) preconditioner. This method is typically faster than the direct solver and uses less memory for medium to large 3D models. For details, see Manual Setup of GMG Solver Suggestions and Theory.
2. *Suggested Iterative Solver (FGMRES with GMG)*: uses the FGMRES iterative solver with a geometric multigrid (GMG) preconditioner. This method is more robust than GMRES, especially for problems that exhibit sharp resonances. If the GMRES suggestion does not converge try the FGMRES suggestion instead. For details, see Manual Setup of GMG Solver Suggestions and Theory.
3. *Suggested Iterative Solver (Shifted Laplace)*: For increasing frequencies, the first two suggested iterative solvers, described above, will eventually stop converging. One solution is to use the complex shifted Laplacian (CSL or SL) method for the multigrid preconditioner. The SL method will in general speed up convergence for larger models. For details, see Complex Shifted Laplacian for Very Large Frequency Domain Models.
4. *Suggested Iterative Solver (Domain Decomposition)*: This last suggestion is for solving very large models that need to run on a cluster (using a distributed architecture). The performance of the method will be best when used on several nodes. For details, see Domain Decomposition for Helmholtz on Clusters.

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

For models that run with a very fine frequency step (with a linear frequency distribution) it can be considered to change the **Reuse solution from previous step** option to **Auto** (or **Yes**). This will help convergence by providing a good initial guess for the iterative solvers.

However, in most cases for models that have a moderate spacing in the solved frequencies (and where frequencies are given on a logarithmic axis), the default option **Off** should be used. This setting will give the best performance in almost all cases.

For an example that solves a model using an iterative solver see: *Headphone on an Artificial Ear*. Application Library path **Acoustics_Module/Electroacoustic_Transducers/headphone_artificial_ear**

COMPLEX SHIFTED LAPLACIAN FOR VERY LARGE FREQUENCY DOMAIN MODELS

When the frequency in the solved problem increases, the first two suggested iterative solvers, described above, will eventually stop converging. One solution is to use the complex shifted Laplacian (CSL or SL) method for the multigrid preconditioner. The SL method will in general speed up convergence for larger models. One exception, where the SL method will not help, is in models solved near resonances with low damping from the boundaries. Here, the default GMG suggestion described above will most likely work better. For a pure Pressure Acoustics model the SL solver suggestion is set up automatically with appropriate settings.

Manual Setup of the SL Solver

To set up the SL method manually, expand the solver tree and select the **Multigrid** node under the first iterative suggestion (enable the suggestion). Select the **Prefer the free matrix format** checkbox and then select the **Shifted Laplace contribution** checkbox. Now select the **Physics** (a pressure acoustics interface present in the list) and select the **Manual** option for the **Add weak contributions**. In the **Weak expression** field, enter the following term:

```
shift*acpr.p_t*test(p)*(-i*abs(acpr.k[m])^1.5)*acpr.delta/
acpr.rho_c
```

where `shift` is a parameter between –1 and 1 (start with 0.5). This parameter acts as a relaxation factor. If the physics interface tag is different from `acpr`, update the expression to match the physics interface tag. The added weak term does not modify

the solution of the original problem. It is only used on the multigrid levels to help with convergence.

The same should be done by selecting the **Shifted Laplace contribution on multigrid levels** checkbox. Then select the **Manual** option and add the same term as **Weak expression**.

The first option, **Shifted Laplace contribution**, adds the SL contribution only on the fine level used for the smoothers. The second option, **Shifted Laplace contribution on multigrid levels**, adds the SL contributions also on all multigrid levels.

To further improve the method at very high frequencies the **Hierarchy generation method** option (on the multigrid mode) can be changed from the default **Lower element order first (any)** to **Coarsen mesh**. Then enter an appropriate **Mesh coarsening factor**. For acoustics models meshed with λ/N the coarsening can be up to about N. Typically use $N = 5$ and set a coarsening factor between 2 and 4.

The default iterative solver is GMRES which can become memory consuming for large problems. When the problem requires many iterations to be solved, it is typically necessary to increase the **Number of iterates before restart** to say 1000. In this case the iterative solver can be changed to TFQMR. This solver will typically require more iterations to reach convergence, but it is more memory lean.

Another memory saving option is to use the **Block low rank factorization** option for the **Direct** solver (when using MUMPS) under the **Coarse Solver** node. Expand the multigrid solver node to locate it.

For further details see Complex Shifted Laplacian for Large Helmholtz Problems section in the *COMSOL Multiphysics Reference Manual*.

For an example that solves a pressure acoustics model using the shifted Laplacian solver. see the COMSOL Application Library model: *Car Cabin Acoustics — Frequency-Domain Analysis*

DOMAIN DECOMPOSITION FOR HELMHOLTZ ON CLUSTERS

For very large models that can be solved on clusters with many nodes, you can set up a special version of domain decomposition for Helmholtz equation. The method can

be set up to use multigrid with the complex shifted Laplacian method as well as absorbing boundaries for the domains.

For further details see Complex Shifted Laplacian for Large Helmholtz Problems section in the *COMSOL Multiphysics Reference Manual*.

EFFICIENT ITERATIVE SOLVER WITH CUBIC DISCRETIZATION AND TWO MULTIGRID LEVELS

Solving very large pressure acoustics models in the frequency domain can be challenging. Several iterative solver suggestions exist for the default physics setups and the default quadratic Lagrange discretization (as described above). For very large models it can be advantageous to switch from quadratic to **Cubic Lagrange** discretization. This will allow for the use of a coarser mesh and the use of an optimized two-level multigrid solver approach. The following steps are necessary to set up this solver:

1. Add an **Iterative** solver
   - **a** Select the **TFQMR** solver
   - **b** Use **Right** preconditioning
   - **c** Set the **Maximum number of iterations** to 1e6
2. Add the **Multigrid** preconditioner
   - **a** Use **Lower element order first (any)**
   - **b** Set the **Number of multigrid levels** to 2
   - **c** Set the **Mesh coarsening factor** to 2
   - **d** In the **Equations** section enable **Shifted Laplace contributions on multigrid levels**
   - **e** Select the pressure acoustics interface and set **Add weak contributions** to **Automatic**
   - **f** Change the **Shift coefficient** to 5
3. In the **Presmoother > SOR**, use the **SOR** solver, set **Number of iterations** to 1, and the **Relaxation factor** to 0.5
4. In the **Postsmoother > SOR**, use the **SORU** solver, set **Number of iterations** to 1, and the **Relaxation factor** to 0.5
5. Use a Direct solver for the **Coarse Solver**, use **MUMPS**, and enable **Block low rank factorization**

With this solver and using **Cubic Lagrange** discretization the mesh can be set such that the **Maximum mesh element size** is only 0.45 λ, where λ is the wavelength. If a PML is present in the model use 4 mesh layers, and the **Polynomial** scaling.

For an example that solves a pressure acoustics model using cubic discretization and two multigrid levels see the COMSOL Application Gallery model: Car Cabin Acoustics — Iterative Solver Suggestion for Cubic Elements

MANUAL SETUP OF GMG SOLVER SUGGESTIONS AND THEORY

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

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

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

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

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

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

Using GMRES or FGMRES as an outer iteration and smoother removes the requirements on the coarsest mesh. When GMRES is used as a smoother for the multigrid preconditioner, FGMRES must be used for the outer iterations because such a preconditioner is not constant (see Ref. 35).

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

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

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

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

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

Perfectly Matched Layers (PMLs)

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

A **Perfectly Matched Layers** ( ) is added to the model in the **Definitions** node in the component where the physics is solved. In the frequency domain, the PMLs can be used for the Pressure Acoustics, Acoustic-Structure Interaction, Aeroacoustics, and Thermoviscous Acoustics interfaces. In the time domain the PMLs only exist for the Pressure Acoustics, Transient interface. Note that for Pressure Acoustics, Frequency Domain models, the Perfectly Matched Boundary is an alternative to the PML for radiation and scattering problems. This boundary conditions applies the PML automatically using the extra dimension machinery of COMSOL Multiphysics.

In this section:

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

- Perfectly Matched Layers in an Eigenfrequency Analysis

GEOMETRY OF THE PML LAYER

When creating the geometry for your model, it is advantageous to use the **Layers** feature in the geometry to create the PML domains. This ensures that the geometry is suited for a structured mesh. The physical thickness of the layers is not important in frequency domain models. Here a real stretching is applied to mathematically scale the thickness relative to the wavelength. The thickness should however be such that the mesh is more or less regular (avoid too thin mesh elements). In the time domain, the thickness is important, see Time Domain Perfectly Matched Layers for details.

If the PML is located close to a radiating source or a scatterer, evanescent wave components can interact with the PML stretching and generate unphysical reflections. This can be avoided by tuning the Coordinate Stretching, Scaling, and Curvature parameters. To further prevent this, it is also recommended to place the PML more than $\lambda/8$ away from these surfaces, but it is not necessary, if the PML parameters are tuned correctly.

To assess the performance of the PML it can be advantageous to plot the total radiated power to the PML, using the predefined variable `phys.P_pml`. This quantity should always be positive.

GEOMETRY TYPE (USER DEFINED OPTION)

When setting up a PML, you select the geometry type of the layer. This is only related to how the layer looks in the geometry. Typically, the predefined options **Cartesian**, **Cylindrical**, or **Spherical** can apply in most situations. Using these, COMSOL will automatically detect the layer thickness and define the local coordinates inside the PML. In some cases the automatic detection can fail (this can, for example, happen for certain imported CAD geometries). The automatic detection also fails if the domain is not the outer most entity in the geometry.

A workaround, when the automatic detection fails, is then to use the **User defined** geometry type. This advanced option makes it possible to define the local *Distance functions* and layer *Thickness* manually. For example, for a spherical PML geometry the typical distance function is `sqrt(x^2+y^2+z^2)-r0`, where `r0` is the radius of the inner domain. The user-defined option can also be used for special layer shapes.

To verify that the geometry detection is correct, or a user defined geometry type is set up correctly, it can be useful to plot the normalized distance functions of the PMLs. The values should lie be between 0 and 1. Select **Get Initial Value** on the study (it is not necessary to solve) and then plot the variable `<tag>.sDist<i>`, where `<tag>` is the

PML tag (`pml1`, `pml2`, and so on) and a number `<i>` (1, 2, and so on) is added if several stretching functions are used in the PML, for example in a corner.

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

	In the following model a user defined geometry type is used: *Headphone on an Artificial Ear*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/headphone_artificial_ear**

MESHING THE PMLS

Optimal behavior of the PML is achieved when the mesh inside the PML domain is structured. Use a mapped mesh in 2D models and a swept mesh in 3D models. Use at least 8 layers when using the default polynomial stretching option. As a good starting point for the rational stretching use 5 or 6 mesh layers inside the PML.

COORDINATE STRETCHING, SCALING, AND CURVATURE

The choice of the **Coordinate stretching type** and the **PML scaling factor** and the **PML curvature parameter** depends on the problem at hand. A detailed description is given in the PML Implementation section of the *COMSOL Multiphysics Reference Manual*. In general, the **Rational** stretching option is used for open radiation problems for propagating waves (it is efficient for many angles of incidence). The **Polynomial** stretching option is good for systems with a mix of different wave types, for example, in multiphysics problems involving structural and acoustic waves, or problems containing a combination of propagating and evanescent waves. For the polynomial stretching type, the **PML scaling curvature parameter** can in general be increase to a value between 3 and 5 for better performance at low frequencies. Note that when solving a model using an iterative solver the **Polynomial** scaling should always be used to ensure convergence.

The polynomial stretching should also be used at the end of waveguides. However, in pressure acoustics (and thermoviscous acoustic) models you should use the Port condition as the waveguide termination as is provides a superior nonreflecting condition.

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

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

Plotting the total radiated power to the PML can be a good indicator of possible issues (especially at low frequencies). The predefined variable `phys.P_pml` is readily available for plotting.

The behavior of the PMLs at low frequencies is, for example, discussed in the following model. *Lumped Loudspeaker Driver*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/lumped_loudspeaker_driver**

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

The PMLs damp a certain wavelength existing in the system. The wavelength is deduced from the frequency and a reference wave speed c_{ref}. The wave speed is defined in the Typical Wave Speed section.

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

LIMITATIONS OF THE PERFECTLY MATCHED LAYERS

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

domain the pressure DOF is interpreted as a scattered field, while it is the total field in the other. Note that you can set up models that contain this feature configuration as long as the PMLs do not touch.

!	When a perfectly matched layer (PML) is present in the model do not apply an Incident Pressure Field on its outer boundaries. The PML is applied to absorb waves that move out of the computational domain. Defining an incident field on its boundary will lead to unphysical results.

TIME DOMAIN PERFECTLY MATCHED LAYERS

In the time domain the PML does not include a real stretching component. This means that the geometrical thickness L, of the layer in the geometry, needs to be set adequately. When meshing the PMLs for time domain simulations, it is recommended to use a structured mesh in the same way as in the frequency domain. In the time domain use at least 8 mesh layers for both the rational scaling and the polynomial scaling; and the same mesh element size as that in the adjacent physical domain (a detailed investigation is available in Ref. 42).

The frequency content of the transient signal should be considered when defining the layer thickness L. Assume most of the energy is carried in a band from $f_{\min}$ to $f_{\max}$. This represents wavelengths from $\lambda_{\max}$ to $\lambda_{\min}$ (remember $\lambda = c/f$). The geometrical thickness of the layer has two considerations to fulfill:

1. The thickness should be larger than $\lambda_{\max}/8$
2. The thickness should be such that you have a minimum of 6 (or 8 depending on the scaling) mesh elements in the thickness at $f_{\max}$. The mesh in the layer has to also resolve the minimum wavelength.

In the limit of a signal with a narrow band frequency content (or a single frequency), the two points will be fulfilled by defining a thickness $L = \lambda$.

The recommended values of the *PML scaling factor* and the *PML scaling curvature parameter* are 1, 3 and 1, 1 for the **Polynomial** and the **Rational** stretching types, respectively. For the **Polynomial** stretching, the *PML scaling factor* equal to 1 corresponds to the theoretical reflection coefficient $R_0 = 10^{-3}$ from the interface between the physical domain and the PML for a plane wave.

Note that the absence of a real stretching makes the time domain PMLs unable to efficiently absorb evanescent waves.

	In the settings for the **Perfectly Matched Layer** it is important that if you change the **Typical wavelength from** option to **User defined**, then it is not the actual wavelength that should be entered but rather the speed of sound per Hertz. For example, if **User defined** is selected in a normal air domain, then enter `343[m/s]/1[Hz]`. The reason is that in the time domain the PML is not related to wavelength but to speed of sound. Transient signals typically include many Fourier frequency components.

	See Theory for the Perfectly Matched Layers in the Time Domain for details about the implementation.

PERFECTLY MATCHED LAYERS IN AN EIGENFREQUENCY ANALYSIS

Care should be taken if a perfectly matched layer (PML) is used when performing an eigenfrequency analysis. Because the frequency is not a priori known (it is solved for), the real scaling of PML will be defined with the typical wavelength set equal to the PML layer thickness. To obtain adequate scaling and absorption, the **Typical wavelength from** should be set to **User defined** and wavelength entered, given at the expected eigenfrequency searched for (remember $\lambda = c/f$). In practice the solution will only be correct in the vicinity of that frequency. Use the **Search for eigenfrequencies around** option to tune the analysis.

Note also that the PML is a mathematical, unphysical construction. Firstly, this means that it will introduce damping into the model. This will typically reflect in a complex valued eigenfrequency (where the magnitude of the complex part cannot be associated with a physical damping mechanism). Secondly, spurious (unphysical) modes inside of the PML can exist. Therefore, it is always important to inspect the computed modes to ensure that they represent a physical solution.

Typically, a good approach is to use a radiation condition instead of a PML in open problems, solved in an eigenfrequency study.

Postprocessing Variables

Several specialized variables specific to acoustics are predefined in the Acoustics Module and can be used when analyzing the results of an acoustic simulation. The variables are available from the expression selection menus when plotting.

In this section:

- Intensity Variables
- Power Dissipation Variables
- Pressure Acoustics, Boundary Mode Variables

In the *COMSOL Multiphysics Reference Manual*:

- Results Analysis and Plots
- Operators, Functions, and Constants

INTENSITY VARIABLES

The propagation of an acoustic wave is associated with a flow of energy in the direction of the wave motion, the intensity vector **I**. The sound intensity in a specific direction (through a specific boundary) is defined as the time average of (sound) power per unit area in the direction of the normal to that area.

Knowledge of the intensity is important when characterizing the strength of a sound source — that is, the power emitted by the source. The power is given by the integral of $\mathbf{n}\cdot\mathbf{I}$ on a surface surrounding the source, where **n** is the surface normal. The intensity is also important when characterizing transmission phenomena, for example, when determining transmission loss, insertion loss curves, or absorption.

The acoustic intensity vector **I** (SI unit: $\mathrm{W/m^2}$) is defined as the time average of the instantaneous rate of energy transfer per unit area (sound power) $p\mathbf{u}$, such that

$$\mathbf{I} = \frac{1}{T}\int_0^T p\mathbf{u}\,dt \tag{2-4}$$

where p is the pressure and **u** the particle velocity.

Frequency Domain

In the frequency domain (harmonic time dependence), the time averaging integral Equation 2-4 reduces to

$$\mathbf{I} = \frac{1}{2}\mathrm{Re}(p\mathbf{u}^*) = \frac{1}{4}(p\mathbf{u}^* + p^*\mathbf{u}) \quad (2\text{-}5)$$

where **?*** denotes complex conjugation. In the frequency domain the velocity is readily expressed in terms of the pressure as $\mathbf{u} = -1/(i\omega\rho)\nabla p$. Using the specific characteristic acoustic impedance for plane waves, the intensity can also be expressed in terms of the root mean square (RMS) pressure as

$$\mathbf{I} = \frac{1}{2}\frac{pp^*}{\rho c}$$

In the Pressure Acoustics, Frequency Domain interface the general formulation, valid for all types of waves given in Equation 2-5, is used to define the intensity.

When using the scattered field formulation, by adding a Background Pressure Field feature, the intensity variables for the total, background, and scattered fields are available in postprocessing if the **Calculate background and scattered field intensity** option is selected.

Time Domain

For time-dependent problems, the equivalent quantity is the instantaneous value of the intensity, defined as

$$\mathbf{i}_{\mathrm{inst}} = p\mathbf{u}$$

This expression is difficult to recover in pressure acoustics and would require the solution of an additional PDE to calculate the velocity from the pressure dependent variable. Only the intensity $\mathbf{I}$ (time averaged) is available as postprocessing variable in the frequency domain and can be selected from the expressions menus when plotting. The instantaneous intensity does exists as a postprocessing variable in transient interfaces such as Linearized Euler, Linearized Navier–Stokes, or Thermoviscous Acoustics where the velocity is solved for explicitly, see Modeling with the Aeroacoustics Branch or Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces) for details.

Postprocessing Variables

The variables are defined in Table 2-3, Table 2-4, and Table 2-5. In the variable names, *phys_id* represents the interface name, for example, acpr for a Pressure Acoustics, Frequency Domain interface.

TABLE 2-3: INTENSITY VARIABLES IN 3D.

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ix	x-component of the intensity vector (frequency domain only)
phys_id.Iy	y-component of the intensity vector (frequency domain only)
phys_id.Iz	z-component of the intensity vector (frequency domain only)

TABLE 2-4: INTENSITY VARIABLES IN 2D AXISYMMETRIC.

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ir	r-component of the intensity vector (frequency domain only)
phys_id.Iz	z-component of the intensity vector (frequency domain only)

TABLE 2-5: INTENSITY VARIABLES IN 2D.

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ix	x-component of the intensity vector (frequency domain only)
phys_id.Iy	y-component of the intensity vector (frequency domain only)

In the *COMSOL Multiphysics Reference Manual*:

- Results Analysis and Plots
- Expressions and Predefined Quantities

POWER DISSIPATION VARIABLES

Common to the Pressure Acoustics fluid models (also porous materials) and The Thermoviscous Acoustics, Frequency Domain Interface is that all the interfaces model some energy dissipation process, which stem from viscous and thermal dissipation processes. The amount of dissipated energy can be of interest as a results analysis variable or as a source term for a multiphysics problem. An example could be to

determine the amount of heating in the human tissue when using ultrasound. In the Acoustics Module special variables exist for the dissipation.

- For the case of a plane wave propagating in the bulk of a fluid (the general thermal and viscous fluid models described in Thermally Conducting and Viscous Fluid Model) the dissipation is

$$\Delta = \frac{1}{(\rho c^2)^2}\left[\mu\left(\frac{4}{3}+\frac{\mu_b}{\mu}\right)+\frac{k(\gamma-1)}{C_p}\right]\left(\frac{\partial p}{\partial t}\right)^2$$

  and in the frequency domain after averaging over one period

$$\Delta = \frac{1}{(\rho c^2)^2}\left[\mu\left(\frac{4}{3}+\frac{\mu_b}{\mu}\right)+\frac{k(\gamma-1)}{C_p}\right]\frac{\omega^2}{2}pp^* \tag{2-6}$$

  where * in Equation 2-6 is the complex conjugate operator.

- In addition, an approximate expression for the dissipated energy density from a propagating plane wave exists for the Narrow Region Acoustics, the Poroacoustics models and attenuation in Pressure Acoustics. This total dissipated power density Q_{pw} is defined by

$$Q_{pw} = -2|\mathbf{I}|\text{imag}(k)$$

  where $|\mathbf{I}|$ is the magnitude of the intensity vector $\mathbf{I}$ and k is the wave number. This expression is an approximation and is only valid for traveling plane waves (or waves that are close to plane); however, it has many uses as a first estimate of the dissipation since it is easy to calculate in many different situations. The expression is, for example, not valid for standing waves in resonant systems. When the above expression is not valid, the dissipated energy should be calculated using an energy balance approach.

The power dissipation variables are defined in Table 2-6. In the variable names, *phys_id* represents the name (acpr, for example, for a pressure acoustics interface).

TABLE 2-6: POWER DISSIPATION VARIABLES.

VARIABLE	DESCRIPTION
phys_id.diss_therm	Thermal power dissipation density
phys_id.diss_visc	Viscous power dissipation density
phys_id.diss_tot	Total thermal and viscous power dissipation density
phys_id.Q_pw	Plane-wave total dissipated power density

PRESSURE ACOUSTICS, BOUNDARY MODE VARIABLES

A series of special variables exist for postprocessing after solving a boundary mode acoustics problem. They include in-plane and out-of-plane components of the velocity **v** and acceleration **a**.

The in-plane (ip) and out-of-plane (op) components to the acceleration and velocity are defined as

$$\begin{aligned}\mathbf{a}_{\text{ip}} &= \mathbf{a} - (\mathbf{a}\cdot\mathbf{n})\mathbf{n}\\ \mathbf{v}_{\text{ip}} &= \mathbf{v} - (\mathbf{v}\cdot\mathbf{n})\mathbf{n}\\ \mathbf{a}_{\text{op}} &= (\mathbf{a}\cdot\mathbf{n})\mathbf{n}\\ \mathbf{v}_{\text{op}} &= (\mathbf{v}\cdot\mathbf{n})\mathbf{n}\end{aligned}$$

where **n** is the normal to the surface being modeled. The velocity and acceleration are defined in terms of the gradient of the pressure p as follows

$$\nabla p = \nabla_{||} p - ik_{\text{n}} p\mathbf{n} \qquad \text{in 3D}$$

$$\nabla p = \nabla_{||} p - ik_{\text{n}} p(n_r, 0, n_z) + p\left(0, -i\frac{m}{r}, 0\right) \qquad \text{in 2D axisymmetry}$$

and

$$\mathbf{v} = \frac{i}{\omega\rho}\nabla p \qquad \text{and} \qquad \mathbf{a} = i\omega\mathbf{v}$$

where k_{n} is the out-of-plane wave number solved for, m is a possible radial wave mode number, and $\nabla_{||}$ is the tangential derivative along the boundary.

The boundary mode acoustics variables are defined in Table 2-7. In the variable names, *phys_id* represents the name (`acbm`, for example, for a Boundary Mode Acoustics interface).

TABLE 2-7: BOUNDARY MODE ACOUSTICS VARIABLES IN 3D.

VARIABLE	DESCRIPTION
phys_id.vipx	In-plane velocity, x-component
phys_id.vipy	In-plane velocity, y-component
phys_id.vipz	In-plane velocity, z-component
phys_id.vip_rms	In-plane velocity RMS value
phys_id.aipx	In-plane acceleration, x-component
phys_id.aipy	In-plane acceleration, y-component

TABLE 2-7: BOUNDARY MODE ACOUSTICS VARIABLES IN 3D.

VARIABLE	DESCRIPTION
phys_id.aipz	In-plane acceleration, z-component
phys_id.aip_rms	In-plane acceleration RMS value
phys_id.vopx	Out-of-plane velocity, x-component
phys_id.vopy	Out-of-plane velocity, y-component
phys_id.vopz	Out-of-plane velocity, z-component
phys_id.vop_rms	Out-of-plane velocity RMS value
phys_id.aopx	Out-of-plane acceleration, x-component
phys_id.aopy	Out-of-plane acceleration, y-component
phys_id.aopz	Out-of-plane acceleration, z-component
phys_id.aop_rms	Out-of-plane acceleration RMS value

Evaluating the Acoustic Field in the Exterior: Near- and Far-Field

The Acoustics Module has functionality to evaluate the acoustic pressure field in the exterior or far-field region outside of the computational domain. This is the Exterior Field Calculation feature available for pressure acoustics problems. This section first gives a definition of the far- and near-field and then some general advice for analyzing the exterior field.

THE NEAR-FIELD AND FAR-FIELD REGIONS

The solution domain for a scattering or radiation problem can be divided into two zones, reflecting the behavior of the solution at various distances from objects and sources. In the *far-field* region, scattered or emitted waves are locally planar, velocity and pressure are in phase with each other, and the ratio between pressure and velocity approaches the free-space characteristic specific impedance of a plane wave rc.

Moving closer to the sources into the *near-field* region, pressure and velocity gradually slide out of phase. This means that the acoustic field contains energy that does not travel outward or radiate. These evanescent wave components are effectively trapped close to the source. Looking at the sound pressure level, local maxima and minima are apparent in the near-field region.

Naturally, the boundary between the near-field and far-field regions is not sharp. A general guideline is that the far-field region is that beyond the last local energy maximum, that is, the region where the pressure amplitude drops monotonously at a rate inversely proportional to the distance R from any source or object.

A similar definition of the far-field region is the region where the radiation pattern — the locations of local minima and maxima in space — is independent of the distance to the wave source. This is equivalent to the criterion for Fraunhofer diffraction in optics, which occurs for Fresnel numbers, $F = a^2/\lambda R$, much smaller than 1. For engineering purposes, this definition of the far-field region can be applied:

$$R > \frac{8a^2}{\lambda} = \frac{8}{2\pi} ka^2 \tag{2-7}$$

In Equation 2-7, a is the radius of a sphere enclosing all objects and sources, λ is the wavelength, and k is the wave number. Another way to write the expression leads to the useful observation that the size of the near-field region expressed in source-radius units is proportional to the dimensionless number ka, with a prefactor slightly larger than one. This relation is known as the Rayleigh radius $R_0 = S/\lambda$, where S is the source area, for example for a piston.

Knowing the extent of the near-field region is useful when applying radiation boundary conditions because these are accurate only in the far-field region. PMLs, on the other hand, can be used to truncate a domain already inside the near-field region.

THE HELMHOLTZ–KIRCHHOFF INTEGRAL REPRESENTATION

In many cases, solving the acoustic Helmholtz equation everywhere in the domain where results are requested is neither practical nor necessary. For homogeneous media, the solution anywhere outside a closed surface containing all sources and scatterers can be written as a boundary integral in terms of quantities evaluated on the surface. To evaluate this *Helmholtz–Kirchhoff* integral, it is necessary to know both Dirichlet and Neumann values on the surface. Applied to acoustics, this means that if the pressure *and* its normal derivative is known on a closed surface, the acoustic field can be calculated at any point outside, including amplitude and phase. This functionality is included in the Exterior Field Calculation feature. The feature has two options for the evaluation, one full integral (the default) and one that only looks in the extreme far field. See the section Theory for the Exterior Field Calculation: The Helmholtz–Kirchhoff Integral for further details.

FULL INTEGRAL

To evaluate the full Helmholtz–Kirchhoff integral, use the default **Full integral** option in the **Type of integral** options. The full Helmholtz–Kirchhoff integral gives the pressure (including phase) at any point at a finite distance from the source surface. This is necessary in many acoustic applications, for example, when analyzing the sensitivity

of transducers. Evaluation inside the exterior field calculation surface gives unphysical values. See Exterior Field Calculation.

THE FAR-FIELD LIMIT

In many scattering applications, the quantity of interest is the far-field radiation pattern, which can be defined as the limit of $r|p|$ when r goes to infinity in a given direction. To evaluate the pressure in the far-field limit set the **Type of integral** to **Far-field integral approximation for $r \to \infty$** in the settings for the exterior field variables. With this option, only the direction of the evaluation point $\mathbf{r}$ is of interest not the distance $|\mathbf{r}|$. See Exterior Field Calculation.

EXTERIOR FIELD OPERATOR

The exterior field pressure is evaluated using the exterior field operator. The name is defined in the **Exterior field variable name** input field and the default is `pext`.

The exterior field pressure can be evaluated in a given point (x_0, y_0, z_0) by simply writing `pext(x0,y0,z0)`. To evaluate the sound pressure level in the same point, it is advantageous to use the `subst()` operator and, for example, write `subst(acpr.efc1.Lp_pext,x,x0,y,y0,z,z0)`.

For an example, see *Loudspeaker Driver*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/loudspeaker_driver**.

These evaluations are automatically carried out in the dedicated postprocessing features presented in the section below.

Dedicated Acoustics Plots for Postprocessing

Dedicated acoustics postprocessing functionality exists to plot responses, transfer functions, sensitivity curves, directivity and radiation patterns, and more. They require the Acoustics Module license.

In this section:

- Radiation Pattern/Bubble Plots (Spatial Response)
- Grid and Parametric Datasets
- Octave Band Plots
- Directivity Plots

RADIATION PATTERN/BUBBLE PLOTS (SPATIAL RESPONSE)

Evaluating and plotting the acoustic pressure around radiating objects and sources is essential for the development of several acoustic devices. This is known as the spatial response, the radiation pattern, or radiation bubble. Application areas range from underwater acoustic transducers and loudspeakers, to determining the spatial sensitivity of microphone systems (for example, using reciprocity). Using the Exterior Field Calculation feature can help to reduce the computational domain while still being able to determine the pressure and thus the spatial response in the exterior field (near-field to far-field). The *Radiation Pattern plots* are specially designed for easy evaluation of the exterior field variables, that is, the acoustic exterior pressure and the exterior field sound pressure level. The plot can readily also be used to evaluate the results obtained with the The Pressure Acoustics, Boundary Elements Interface.

In the radiation pattern plots, the exterior field variable (pressure or sound pressure level) is represented in a polar plot for a selected number of angles. Data is retrieved on an evaluation circle in 2D, 2D axial symmetry, or 3D. The angle interval and the number of angles can be manually specified. The evaluation circle origin, orientation, and radius can be specified as well as the reference direction. The evaluation circle can be visualized using a **Preview Evaluation Plane** functionality. There is also a built-in option to calculate the **Beamwidth** of the plotted data. The data is stored in a table that can be plotted using the **Table Graph** option.

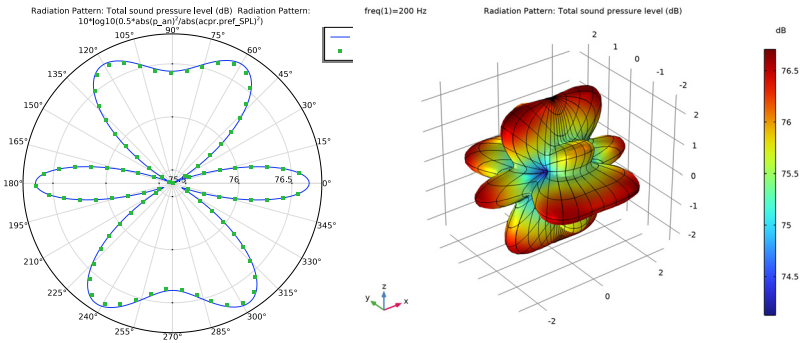


Figure 2-9: Example of a 2D and 3D radiation pattern plot from the Bessel Panel tutorial model.

The 3D radiation pattern plots create a 3D polar plot (or bubble plot) where the data is evaluated on a sphere. You can specify a separate expression for the surface color by deselecting the **Use as color expression** checkbox. A classical bubble plot is created by

setting a constant value for the **Expression** field and using the sound pressure level for the color expression. In the 3D radiation pattern plots, the directivity can also be calculated by enabling the **Compute directivity** option. The result is given in a table and is calculated as the maximum power relative to the average power. The table comprises the maximal radiation direction (polar angles), the directivity, and the directivity given in dB (also known as the directivity index). Note that this definition of the directivity can deviate from other standard formulations (see Ref. 6), if the radiation pattern is not plotted for all directions or if the main lobe is not on-axis.

The main advantage with the radiation pattern plot, as compared to making a line graph, is that the circle/sphere that you use for defining the plot directions, is not part of your geometry for the solution. Thus, the number of plotting directions is decoupled from the discretization of the solution domain.

Variables defined by the **Exterior Field Calculation** feature like, for example, the pressure `acpr.efc1.pext` or sound pressure level acpr.efc1.Lp_pext are globally defined variables and can be directly evaluated in the *Radiation Pattern* plot and *Directivity* plot. The values plotted only make physical sense if they are evaluated outside of the **Exterior Field Calculation** boundary. For results from The Pressure Acoustics, Boundary Elements Interface the variables `pabe.Lp` or `pabe.p` can readily be used in the plot.

To visualize, for example, the sound pressure level inside of the **Exterior Field Calculation** boundary plot the expression: `at2(x,y,acpr.Lp)` in 2D, `at2(r,z,acpr.Lp)` in 2D axisymmetry, or `at3(x,y,z,acpr.Lp)` in 3D. The `at2()` and `at3()` operators ensure the variables are globally defined.

The resulting radiation bubble can be exported to a `.txt` file for easy use in a subsequent *Ray Acoustics* model using the Radiation Pattern (Export) node. The data can then subsequently be used in the Source with Directivity feature.

Default exterior field plots (that use the radiation pattern plots) are automatically added to any model that uses exterior field calculation feature, when **Generate default plots** is selected (the default) on the study node.

In a 3D *Radiation Pattern* plot, the definition of the polar angle θ and the azimuthal angle φ follows the spherical coordinate system (r, θ, φ) typically used in physics.

To help visualize the angles, assuming the center is at the origin, simply plot the following expressions:

- Polar angle θ: `acos(z/sqrt(x^2+y^2+z^2))`
- Azimuthal angle φ: `atan2(y,x)`

For 3D examples, see:

- *Bessel Panel*: Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/bessel_panel**.
- *Loudspeaker Driver in a Vented Enclosure*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/vented_loudspeaker_enclosure**

- Evaluating the Acoustic Field in the Exterior: Near- and Far-Field
- Radiation Pattern and Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*
- Radiation Pattern (Export) and Results Analysis and Plots in the *COMSOL Multiphysics Reference Manual*

GRID AND PARAMETRIC DATASETS

Another way of evaluating and depicting the exterior field is by using either the **Grid 2D** , the **Grid 3D** , the **Parametric Curve 2D** and **Parametric Curve 3D**, or the **Parametric Surface** datasets. All of these datasets allow the evaluation of global quantities, like the exterior field variables, outside of the computational domain (outside of the mesh).

The exterior field is depicted in several ways in the model *Piezoelectric Tonpilz Transducer*: Application Library path **Acoustics_Module/Piezoelectric_Devices/tonpilz_transducer**.

OCTAVE BAND PLOTS

A dedicated **Octave Band** ( ) plot exists to plot frequency response, transfer functions, transmission loss, and insertion loss curves. The plot has several built-in acoustics specific features like predefined weighting (Z, A, C, and User defined) as well as the possibility to plot the response in octaves, 1/3 octaves, 1/6 octaves, or as a continuous response.

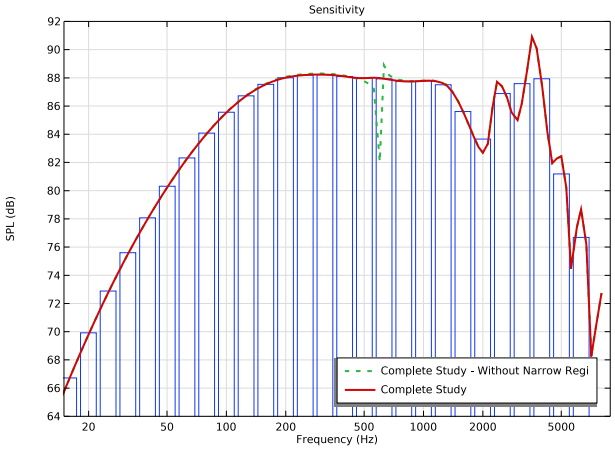


Figure 2-10: Sensitivity curve plotted as 1/3 octave bands and a continuous curve. From the Loudspeaker Driver model in the Application Library.

For details see Octave Band in the Results Analysis and Plots chapter in the *COMSOL Multiphysics Reference Manual*.

The predefined Z, A, and C weightings are defined in IEC 61672-1. See *IEC 61672-1 Electroacoustics - Sound level meters - Part 1: Specifications* for details.

For examples that use an Octave Band plot, see:

- *Absorptive Muffler*. Application Library path **Acoustics_Module/Automotive/absorptive_muffler**
- *The Brüel & Kjær 4134 Condenser Microphone*. Application Library path **Acoustics_Module/Electroacoustic_Transducers/bk_4134_microphone**.

DIRECTIVITY PLOTS

Another acoustics specific plot, especially used for loudspeakers, is the dedicated **Directivity** ( ) plot. The plot allows audio engineers to depict the spatial response of a loudspeaker as function of both frequency and spatial angle in a contour-like plot. Representing the spatial response in this manner is a very common in the loudspeaker industry. Measured data is often also represented in the same manner. The plot includes many options to format the plot to achieve maximal insight into the modeled data, for example, linear and logarithmic frequency axis scaling options, easy switch of the x- and y-axis (frequency and polar angle axis), as well as options to normalize the data with respect to a given angle or the maximal value.

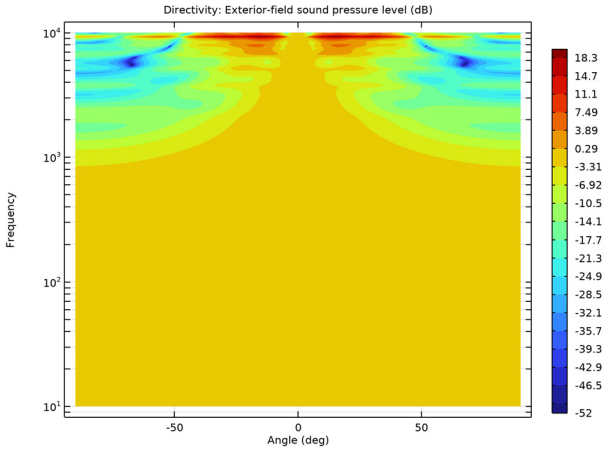


Figure 2-11: Example of the directivity plot from the Lumped Loudspeaker driver model in the Application Library.

For details see Directivity in the Results Analysis and Plots chapter in the *COMSOL Multiphysics Reference Manual*.

Lumped Loudspeaker Driver. Application Library path **Acoustics_Module/Electroacoustic_Transducer/lumped_loudspeaker_driver**

About the Material Databases for the Acoustics Module

The Acoustics Module includes two material databases: Liquids and Gases, with temperature-dependent fluid dynamic and thermal properties, and a Piezoelectric Materials database with over 20 common piezoelectric materials.

	For detailed information about Materials, the Liquids and Gases Material Library, and the Piezoelectric Materials Library see the *COMSOL Multiphysics Reference Manual*.

	For a discussion about the air and water materials as commonly used in acoustics, as well as attenuation models, and other material properties see the Acoustic Properties of Fluids chapter of this manual.

Specifying Frequencies: Logarithmic and ISO Preferred

When solving a model in the frequency domain, you need to specify the **Frequencies** to solve for in the **Frequency Domain** study step. Several built-in options exist when selecting the **Range** ( ). Specifically for acoustics modeling the **Entry method** has the **Logarithmic** and the **ISO preferred frequencies**. The latter option is only available with the Acoustics Module and is activated by clicking the **Show More Options** button ( ) and selecting **Advanced Physics Options** in the **Show More Options** dialog. For **ISO preferred frequencies**, select a start and stop frequency and the interval (Octave, 1/3 Octave, 1/6 Octave, 1/12 Octave, and 1/24 Octave). The 1/3 octave preferred frequencies are defined based on ISO 266. They are extended to frequencies based on the preferred numbers of ISO 3 (series R20, R40, and R80).

Modeling with the Pressure Acoustics Branch (BEM-Based Interface)

This section contains good practices and modeling tips and tricks for the boundary element method (BEM) based interface of the Pressure Acoustics Branch: The Pressure Acoustics, Boundary Elements Interface.

In this section:

- When to Use BEM
- Selections: Infinite Void and Finite Voids
- Solvers for BEM Models
- Meshing BEM Models
- Postprocessing BEM Results

When to Use BEM

Solving acoustics problems using the boundary element method (BEM) can be very advantageous compared to the finite element method (FEM) for some types of problems. For others problems, FEM has its advantages. The recommended approach is to use the BEM and FEM methods where they apply best and also to combine them. Since the BEM interface is fully enabled with multiphysics capabilities, it can be coupled seamlessly with the physics interfaces that are based on FEM. This functionality includes coupling to vibrating structures with the Acoustic–Structure Boundary multiphysics coupling and to other acoustic domains with the Acoustic FEM–BEM Boundary multiphysics coupling. This approach enables modeling in a FEM-BEM framework, using the strength of each formulation adequately. The BEM-based interface is especially well suited for radiation and scattering problems.

The advantage of the boundary element method is that only boundaries need to be meshed and the DOFs solved for are restricted to the boundaries. This introduces some clear ease-of-use for handling complex geometries. However, the BEM technique results in fully populated or dense matrices that need dedicated numerical methods. Assembling and solving these matrices can be very demanding. This means that when solving acoustic models of small and medium size, The Pressure Acoustics, Frequency Domain Interface will often be faster than solving the same problem with the BEM interface.

The challenge with using the FEM interface is to set up open boundaries in an efficient way, for example, by using PMLs. When PMLs cannot easily be placed snugly around radiating structures or two structures are far apart, large air domains need to be meshed. This costs a lot on the computational side, as the frequency is increased. These problems are well suited for the BEM interface, as only the boundary of the radiating structure needs to be meshed.

Interior problems can be solved with the BEM interface, but typically it is much more efficient to solve these problems using the FEM-based interfaces. Remember also that an iterative solver will struggle to solve interior problems, with no or little damping, when resonances appear. If you are close to an eigenfrequency with a high Q, the solution will also be very mesh sensitive.

Note also that at very low frequencies, the BEM method may "break down" for interior problems and give inaccurate results. This is in practice not an issue, as this kind of problem should be solved with The Pressure Acoustics, Frequency Domain Interface.

Selections: Infinite Void and Finite Voids

For modeling with BEM, a geometry model can have multiple domains and multiple finite voids. However, there can be only one infinite void.

- A *domain* corresponds to a finite region where the geometry model is of the type solid. The geometry in most models based on FEM consists of domains only.
- Any other finite region is a *finite void*. A finite void can be created as a geometric primitive of type curve (in 2D) or surface (in 3D), for example, or by a Boolean difference operation, subtracting a closed domain from the model geometry.
- The *infinite void* is the single unbounded region surrounding the geometry model. This is the typical region selected when modeling scattering or radiation problems with the Pressure Acoustics, Boundary Elements interface.
- If the model contains finite voids and an infinite void, those volumetric regions appear in the domain selection lists for the BEM interface and in the **Selection List** window as **Finite void 1**, and so on, and as **Infinite void**.
- It is important to also assign the materials under the **Materials** node to the appropriate selections of the Pressure Acoustics model. Note that material parameters need to be constant for each applied Pressure Acoustics model.
- Assigning the Pressure Acoustics model to a selection adds the necessary equations to the exterior boundaries of the selections. Only these boundaries need to be

meshed. The condition at infinity (of the *infinite void*) is selected in the Condition at Infinity section. The typical and default is an outgoing wave condition.

Solvers for BEM Models

The default for The Pressure Acoustics, Boundary Mode Interface is to use an iterative solver for 3D models and a direct dense matrix solver for 2D problems. If the Far-Field Approximation is turned off, the direct dense matrix solver will be used in all situations. The dense matrix solver is adequate for small BEM problems but quickly runs out of memory for increasing number of DOFs. It is recommended to use the default settings.

In 3D two iterative solver suggestions are generated. The default iterative suggestion uses the Space Approximate Inverse (SAI) as preconditioner. This approach is memory efficient but for increasing frequencies the convergence is compromised. In this case the **Stabilized formulation** can be used (see below). As an alternative to using the stabilized formulation the second iterative suggestion can be used (enable the solver suggestion). This solver uses a Hierarchical LU (HLU) preconditioner which produces LU factors of high quality. This will ensure better convergence for the iterative GMRES solver. The memory consumption is higher for this approach.

When the **Stabilized formulation** is used, a different solver suggestion is generated. Make sure to reset the solver to default if changes are made. The solver uses PARDISO as a direct preconditioner (with hybridization splitting the BEM and the stabilized variables) and GMRES as the iterative method.

The boundary element method (BEM) used in the Pressure Acoustics, Boundary Elements interface is based on the direct method with Costabel's symmetric coupling. The adaptive cross approximation (ACA+) fast summation method is used for solving the resulting linear system. This is the compression method (or Far-Field Approximation) used per default. This method uses partial assembly of the matrices where the effect of the matrix vector multiplication is computed. The default iterative solver used is GMRES.

The Far-Field Approximation settings are used for matrix assembly and postprocessing. They allow characterization of interactions occurring in boundary element method into near-field and far-field interactions. While the near-field interactions are represented explicitly, the far-field interactions can be represented in an approximate way. This approach results in considerable memory and performance improvements when used in combination with iterative solvers using matrix-free format or during

postprocessing. The near-field part of stiffness matrix is used as input by the Direct and Sparse Approximate Inverse (SAI) preconditioners.

For multiphysics problems that combine BEM and FEM interfaces and are set up with one of the multiphysics couplings; the default approach is to use hybridization with SAI for BEM and an appropriate preconditioner for the FEM part of the problem (direct or multigrid). Several solver suggestions are given for the multiphysics FEM-BEM problems.

Further details about the BEM formulation are found in The PDE, Boundary Elements Interface documentation in the *COMSOL Multiphysics Reference Manual.*

Meshing BEM Models

When solving an acoustics problem with the boundary element method, the same restrictions exist on the mesh as for the finite-element-based interfaces. In general, the mesh should resolve the wavelength with 5 to 6 elements. The mesh should also resolve small details and curved surfaces adequately. For exterior problems, the mesh can in practice be slightly coarser and it typically suffices to use 4 elements per wavelength. Using a structured quad surface mesh will result in fewer degrees of freedom and should be used if possible.

Postprocessing BEM Results

When solving a problem with The Pressure Acoustics, Boundary Elements Interface, the resulting solution will consist of the dependent variables on boundaries. Evaluating the solution in a domain is based on a kernel evaluation.

On boundaries, dependent variables exist for the pressure `p`, up and down pressure-dependent variables on interior boundaries (`pabe.p_up` and `pabe.p_down`), as well as a normal flux variable `pabe.pbam1.bemflux`.

The Pressure Acoustics, Boundary Elements Interface sets up predefined postprocessing variables that combine the properties of the boundary variables, when needed, with variables based on the kernel evaluation.

The BEM solution can be evaluated at a given coordinate using the `at3_spatial(...)` operator. In a 3D model you can evaluate the sound pressure level in the point $(x,y,z) = (1\text{ m},0,0)$ by typing: `at3_spatial(1[m],0,0,pabe.Lp,'minc').`

This can, for example, be used to normalize the data in a **Radiation Pattern** plot by typing:
`pabe.Lp-at3_spatial(1[m],0,0,pabe.Lp,'minc')`

Domain Evaluation

The BEM solution can in general be evaluated in domains using the **Grid 2D** and **Grid 3D** datasets. This sets up an underlying grid for the kernel evaluation of the solution. The grid dataset can, for example, be selected as the input **Dataset** for a slice plot. A grid dataset and a multislice plot are generated and used in the default plots when a BEM model is solved. Note also that the kernel-based variables can be evaluated on domains that are meshed.

All the Dedicated Acoustics Plots for Postprocessing that exist for the FEM interfaces can be used with the variables defined by the BEM solution. This is, for example, the **Radiation Pattern** plot or the **Directivity** plot used for analyzing the spatial radiation pattern of a transducer or a scattered field.

Symmetries and Indefinite Boundaries

There are important aspect on how results are visualized when the Symmetry/Infinite Boundary Condition functionality is used:

- First, when setting the **Parameter Bounds** in the grid dataset. Say the condition represents a symmetry plane then visualize the solution in the entire domain. On the other hand if the condition represents an infinite sound hard boundary, like the ground or an infinite baffle, then only visualize the solution above that plane by setting the **Parameter Bounds** in the **Grid 3D** or **Grid 2D** dataset in **Results**.
- Secondly, evaluation on the grid dataset will also show results inside radiating/scattering objects, on the symmetry side. These are unphysical results and can be removed using a **Filter** subnode with a logic expression.

Boundary Evaluation

When evaluating the solution on boundaries, the plot lists are populated with special boundary variables in a **Boundary variables** submenu located under, for example, the **Pressure and sound pressure level** or **Acceleration and velocity** menu. The boundary variables have a "`_bnd`" added to their names. For example, the total acoustic pressure on a boundary is `pabe.p_t_bnd`. On interior boundaries, there are also up and down variants of the variables, with "`_up`" and "`_down`" added, respectively. On interior boundaries (where an interior boundary condition has been added), the boundary variable is the average of the up and down variables.

The boundary variables should be used when plotting on surfaces, as they do not rely on the kernel evaluation but are directly related to the DOFs solved for in the BEM problem.

Modeling with the Pressure Acoustics Branch (DG-FEM-Based Interface)

This section contains good practices and modeling tips and tricks for the discontinuous Galerkin finite element method (dG-FEM) based interface of the Pressure Acoustics Branch: The Pressure Acoustics, Time Explicit Interface. In this section:

- Meshing, Discretization, and Solvers
- Using the Partial Fraction Fit for Acoustic Data
- Results: Variables and Quality
- Absorbing Layers
- Storing Solution on Selections for Large Models
- Assemblies and Pair Conditions

Meshing, Discretization, and Solvers

The same considerations for the mesh, discretization, and solvers apply for the Pressure Acoustics, Time Explicit interface as for The Convected Wave Equation, Time Explicit Interface. Details are found in Meshing, Discretization, and Solvers under the modeling section for that physics. The interface is supported on all mesh types and for all combinations of mesh.

PHYSICS-CONTROLLED MESH

The Pressure Acoustics, Time Explicit interface supports physics-controlled mesh. To get an appropriate mesh, remember to set the value of the **Maximum frequency to resolve** in the Transient Mesh Settings section. The automatically generated mesh will use this value and the speed of sound defined in the model to set up an appropriate mesh that follows the best practices.

Using the Partial Fraction Fit for Acoustic Data

The Partial Fraction Fit function is used to fit frequency-dependent data for use in time domain simulations. Specific for acoustics applications, it can be frequency-dependent admittance data used with the **Impedance** condition in The Pressure Acoustics, Transient Interface or The Pressure Acoustics, Time Explicit Interface and

frequency-dependent data for the equivalent density and compressibility used for the **Poroacoustics** model in the The Pressure Acoustics, Time Explicit Interface.

GENERAL

Let $F(\omega)$ be a complex-valued function of angular frequency $\omega = 2\pi f$. An approximate analytical representation of its time-domain original, $F(t)$, can be retrieved from a rational approximation of $F(\omega)$ that is defined on the whole complex plane:

$$F(\omega) \approx \frac{a_0 + a_1(i\omega) + \ldots + a_N(i\omega)^N}{1 + b_1(i\omega) + \ldots + b_N(i\omega)^N} = F_\infty + \sum_{k=1}^{N} \frac{A_k}{i\omega - \alpha_k}. \tag{2-8}$$

Indeed, each fraction term on the right-hand side of Equation 2-8 corresponds to an exponential decay in the time domain

$$L^{-1}\left(\frac{A_k}{s - \alpha_k}\right) = A_k e^{\alpha_k t} H(t), \tag{2-9}$$

where L^{-1} is the inverse Laplace transform and $H(t)$ is the Heaviside step function.

The frequency dependent admittance and thus its partial fraction approximation Equation 2-8 has to fulfill the following conditions for the results to be physically possible:

- Causality, $F(\omega)$ is analytic and nonzero in $\Im(\omega) > 0$;
- Reality, $\overline{F}(\omega) = F(-\omega)$, that is, $\Re(F(\omega))$ is even and $\Im(F(\omega))$ is odd;
- Passivity, $\Re(F(\omega)) > 0$ for all real ω.

The causality and reality conditions are fulfilled if F_∞ is real; the residues, A_k, and poles, α_k, are either real or come in complex-conjugate pairs; and $\Re(\alpha_k) < 0$ (for the exponentials in Equation 2-9 to decay as the time increases). That is

$$F(\omega) \approx F_\infty + \sum_{k=1}^{N_R} \frac{R_k}{i\omega - \xi_k} + \frac{1}{2}\sum_{k=1}^{N_C}\left[\frac{Q_k}{i\omega - \zeta_k} + \frac{\overline{Q_k}}{i\omega - \overline{\zeta_k}}\right], \tag{2-10}$$

where N_R and N_C are the numbers of pure real poles and complex-conjugate pole pairs, respectively.

The partial fraction approximation (expansion) obtained through the built-in **Partial Fraction Fit** function approximates a given function of frequency within a given frequency range, for example, from $f_{\min}$ to $f_{\max}$. The form given by Equation 2-10 ensures that the reality condition is fulfilled. This is not always the case for the causality

and passivity conditions. However, the **Partial Fraction Fit** function has the necessary tools that can be used to make the result fulfill the causality and reality conditions, thus making the time-domain impedance boundary condition physical.

The **Partial Fraction Fit** function fits the input data based on a given tolerance (default 10^{-3}), number of iterations (default 3), or either of those. The options are found in the **Advanced** section. A higher tolerance error results in a higher degree, N, of the polynomials in Equation 2-8 and therefore a larger number of terms in the expansion Equation 2-10. The default tolerance provides a balance between the approximation accuracy and the number of terms in the expansion, thus the computation costs when the expansion is used in an impedance boundary condition. On the other hand, increasing the tolerance may yield better results. The iteration-based approach can come in handy if you would like to see how the fit evolves as the number of iterations, and thus the degree of the polynomials, increases. This option is also useful when the fitted data is polluted by noise. Figure 2-12 and Figure 2-13 demonstrate the situation, where the functionality is used to fit the same "clean" and "noisy" admittance data, respectively.

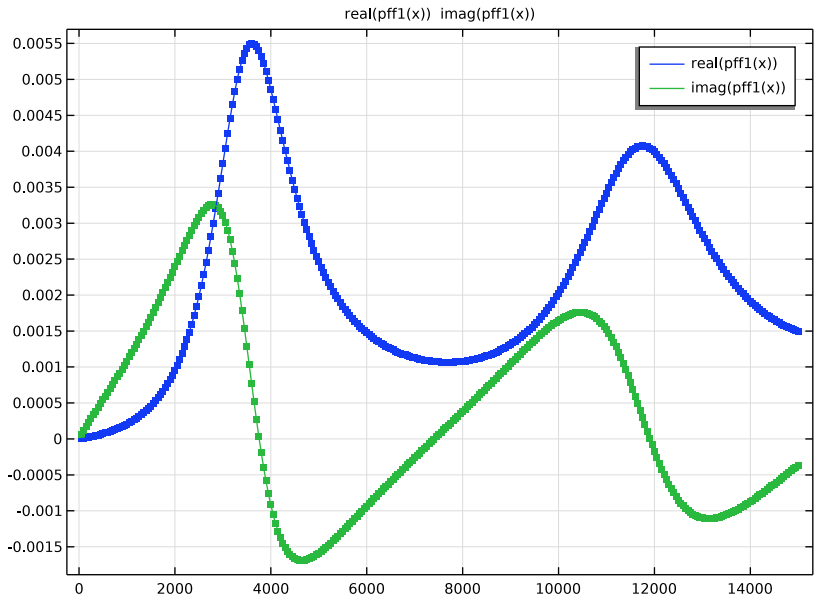


Figure 2-12: Admittance curve with fitted based on the default tolerance.

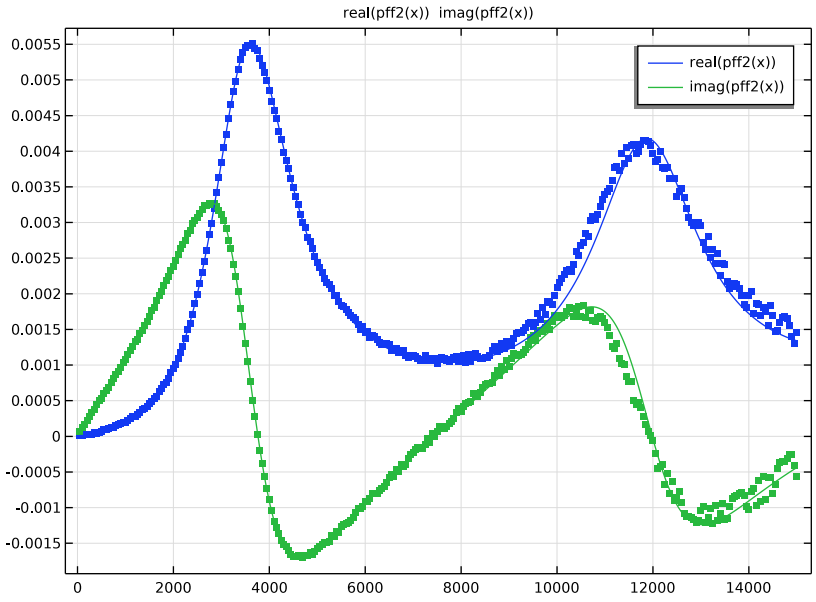


Figure 2-13: Admittance curve with noise fitted based on the default number of iterations.

The "clean" data is fitted with the tolerance of 10^{-3} and results in 3 real poles and 3 pairs of complex-conjugate poles. The "noisy" data is fitted with 3 iterations, which results in a fairly good approximation with 1 real pole and 2 pairs of complex-conjugate poles. If the tolerance-based criterion is used instead, the approximant will interpolate the polluted data to reach the given tolerance. This will result in an undesired fit with an enormous number of poles. For example, fitting the "noisy" data with the tolerance of 10^{-2} yields 70 pairs of complex-conjugate poles, some of them located very close to the imaginary axis.

The violation of the causality condition takes place when one or more poles in the partial fraction expansion have positive real part (that is, they are unstable), which can usually happen to the pure real poles. Click the **Flip Poles** button to flip the unstable poles to the left half plane and circumvent the issue. The residues will be recomputed and updated automatically.

Sometimes the poles (stable or not) lie close to the zeros of the approximant. They can be judged by the residues that have absolute values much smaller than the others, which indicates that the poles and zero nearly cancel, and their influence is often

localized. Such poles often referred to as spurious poles or Froissart doublets. You can either discard them (delete the row on the list) with the following update of the residues by clicking the **Update Residues** button; or run the algorithm with the option **Automatically detect and remove Froissart doublets** enabled. Note that results can be different. In contrast to simply discarding spurious poles, the detection procedure finds residues with absolute values lower than the given threshold compared to the residue with the maximum absolute value and performs a cleanup recomputing the approximant. If desired, change the threshold value in the **Threshold** field (default: 10^{-3}).

- *Wave-Based Time-Domain Room Acoustics with Frequency-Dependent Impedance*. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/wave_based_room**
- *Porous Absorber with Local and Extended Reacting Approximations for Time-Domain Modeling*. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/porous_absorber_time_domain**

USE OF FITTING FOR IMPEDANCE BOUNDARY CONDITIONS AND POROACOUSTICS

When the fitted frequency-dependent data is used for modeling in the time domain, the number of poles/residues in the expansion will correspond to the number of ODEs solved for the memory variables (the so-called auxiliary differential equation method).

Real-valued poles correspond to a pure exponential decay. It is common to get expansions with only real-valued poles when fitting the equivalent density and compressibility used in **Poroacoustic** as they describe the dissipative nature of porous media. For the admittance data used for the **Impedance** boundary condition, the typical behavior is locally-resonant and therefore complex-conjugate pole pairs are common.

It is important that the approximants fulfill all three conditions. However, sometimes there are poles (usually pure real) with the absolute value much larger than that of the others. This results in a stiff ODE, which affects the stability of the time-integration scheme. Specifically, the The Pressure Acoustics, Time Explicit Interface physics interface relies on explicit time-integration schemes. The time step is deduced solely from the minimum mesh element size in a model and the speed of sound. This time step can be too large to achieve a stable solution of the ODE. For the **Impedance**

condition, used in the The Pressure Acoustics, Transient Interface, this is not an issue as it relies on an implicit time integration method that does not exhibit the same stability challenges.

Results: Variables and Quality

The same considerations for results evaluation apply for the Pressure Acoustics, Time Explicit interface as for The Convected Wave Equation, Time Explicit Interface. Details are found in Results: Variables and Quality under the modeling section for that physics.

Absorbing Layers

The same considerations for the absorbing layers and their setup apply for the Pressure Acoustics, Time Explicit interface as for The Convected Wave Equation, Time Explicit Interface. Details are found in Absorbing Layers under the modeling section for that physics.

Storing Solution on Selections for Large Models

The same considerations for storing data apply for the Pressure Acoustics, Time Explicit interface as for The Convected Wave Equation, Time Explicit Interface. Details are found in Storing Solution on Selections for Large Models under the modeling section for that physics.

Assemblies and Pair Conditions

The same considerations for using assemblies and pair condition apply for the Pressure Acoustics, Time Explicit interface as for The Convected Wave Equation, Time Explicit Interface. Details are found in Assemblies and Pair Conditions under the modeling section for that physics.

Modeling with the Aeroacoustic Flow Source: Flow-Induced Noise

This section shortly describes the theory behind the Lighthill acoustic analogy used in the Aeroacoustic Flow Source. Then the hybrid computational aeroacoustic (CAA) method used is introduced. Finally, best practices for setting up an aeroacoustic model in COMSOL using the Aeroacoustic Flow Source, the Aeroacoustic Flow Source Coupling multiphysics, and the necessary study steps such as **Transient Mapping** and **Time to Frequency FFT** are given.

In this section:

- The Lighthill Analogy
- Hybrid Computational Aeroacoustics (CAA)
- Setting up a Flow-Induced Noise Problem in COMSOL

The Lighthill Analogy

In Lighthill's seminal work (Ref. 64 and Ref. 65) he transformed (rearranged) the full set of Navier–Stokes equations (momentum and mass conservation) into an inhomogeneous wave equation. The source term of the wave equation only being important within turbulent flow regions and thus representing the flow-induced noise source. The equation is given by

$$\frac{1}{c_0^2}\frac{\partial^2 p'}{\partial t^2} - \nabla \cdot \nabla(p') = \frac{\partial^2 L_{ij}}{\partial x_i \partial x_j} \qquad p' = c_0^2(\rho - \rho_0) \tag{2-11}$$

$$L_{ij} = \rho v_i v_j + ((p - p_0) - c_0^2(\rho - \rho_0))\delta_{ij} - \tau_{ij}$$

where L_{ij} is called the Lighthill stress tensor, c is the speed of sound, p is the pressure (total), ρ is the density (total), τ_{ij} is the viscous stress tensor, and the subscript 0 refers to the reference properties of the fluid at rest. It is assumed that p' represents the linear acoustic pressure variations outside of the source domain. The Lighthill stress tensor consists of three contributing parts (see Ref. 66): the Reynolds stress (first term), a term containing the excess pressure generated by nonlinear processes and compressibility effects in the moving fluid (second term), and the viscous stress tensor that accounts for attenuation of the sound (third term).

Lighthill's wave equation is often simplified, specifically by simplifying the source term. It can be shown that (see Ref. 66):

$$\rho v_i v_j = \rho_0(1 + O(\mathrm{Ma}^2))(v_i v_j) \approx \rho_0 v_i v_j$$
$$(p - p_0) - c_0^2(\rho - \rho_0) \sim O(\rho_0 v^2 \mathrm{Ma}^2)$$

where Ma is the Mach number. This implies that for low Mach numbers the Lighthill stress tensor is well approximated by

$$L_{ij} = \rho_0 v_i v_j - \tau_{ij} \tag{2-12}$$

This is the expression for the source term stress tensor used when the **Acoustic analogy** option is set to **Lighthill**. The viscous dissipation can optionally be included by selecting **Include viscous stress**. This is a symmetric tensor that contains 9 elements in 3D of which 6 are unique.

The Lighthill wave equation can be further approximated at low Mach numbers for an incompressible flow (see Ref. 66, 68, and 69). This leads to a simplified inhomogeneous wave equation known as the *aeroacoustic wave equation* or AWE, given by

$$\frac{1}{c_0^2}\frac{\partial^2 p}{\partial t^2} - \nabla \cdot \nabla(p) = \frac{1}{c_0^2}\frac{\partial^2 p^{\mathrm{ic}}}{\partial t^2} \tag{2-13}$$

where p is the linear acoustic pressure fluctuation and p^{ic} is the pressure variation associated with the incompressible flow simulation. This equation is solved (in the frequency domain) when the he **Acoustic analogy** option is set to **Aeroacoustic wave equation**. In the Aeroacoustic Flow Source the right hand side is included as the Fourier components of the second time derivative of the incompressible flow variation, $p_{\mathrm{tt}}^{\mathrm{var}}$ in the **Equation** section.

Hybrid Computational Aeroacoustics (CAA)

The computational aeroacoustic (CAA) method used here is of the so-called hybrid type. It is based on a one way coupling between the turbulent flow sources and the acoustic problem. It is thus assumed that no important back-coupling exists from the acoustic field to the flow. The computational method is based on the finite element method (FEM) discretization of Lighthill's acoustic analogy (wave equation). This formulation of the equations ensures that any solid (fixed or vibrating) boundaries are

implicitly taken into account. The method is also applicable for interior problems (see Ref. 66).

The hybrid method is based on a mapping and time to frequency FFT approach, in order to solve the acoustic problem in the frequency domain and on a mesh appropriate for the acoustic problem. The mesh used to solve the LES/DES flow problem has characteristics necessary to resolve the turbulence as well as the flow boundary layers. The mesh also corresponds to linear discretization for most LES/DES methods. The acoustic mesh, on the other hand, has to resolve the wavelength and geometry, and it is typically based on the default second order spatial discretization (see Ref. 66 and Ref. 67).

Setting up a Flow-Induced Noise Problem in COMSOL

When setting up and solving an aeroacoustic flow-induced noise problem in COMSOL Multiphysics, using the built-in functionality of the Acoustics Module, four main steps are necessary:

1. Solve the flow problem using an LES/DES model to resolve the structure of the turbulence. This problem is solved on an appropriate mesh for the flow with appropriate time stepping settings. This typically consists of several steps, for example, first solving a stationary RANS model to get a good initial condition for the transient LES/DES model. Then an initial LES/DES model has to be solved to get a fully developed turbulent flow. In this step the solution only needs to be stored at the end time. Finally, the LES/DES model is run with the fully developed turbulent flow as initial condition. Here, the solution is stored with an appropriate time resolution defined in the **Output times**.

   A time resolution Δt (for the output times) will lead to a maximum frequency resolution of $f_{\max} = 1/\Delta t/2$ in the frequency domain acoustics problem. Moreover, the length of the simulation (T_{end}) will lead to a frequency resolution of $\Delta f = 1/T_{\text{end}}$. Create parameters for these values.
2. Map the solution from the CFD mesh to the acoustics mesh. This is done by using the **Transient Mapping** study step. The input to this study step is only the **Time step** Δt used in the LES/DES study. In the **Transient Mapping** study step it is only the **Aeroacoustic Flow Source Coupling** that is solved for.
3. Transform the mapped flow sources to the frequency domain using the **Time to Frequency FFT** study step. In the **Study Settings** for the study step, the **End time** is set to T_{end} and **Maximum output frequency** is set to $f_{\max}$. In the **Time to Frequency FFT** study step it is also only the **Aeroacoustic Flow Source Coupling** that is solved for.

4 Finally, solve the acoustic problem in the frequency domain. In the **Frequency Domain** study step it is only the **Pressure Acoustics, Frequency Domain** physics that is solved for. The **Values of variables not solved for** settings should point to the **Time to Frequency FFT** study.

Cavity Flow Noise tutorial available in the COMSOL Application Gallery at: https://www.comsol.com/model/cavity-flow-noise-105021

For further details about LES and DES see the *CFD Module User's Guide*:

- The Creeping Flow, Laminar Flow, Turbulent Flow, Large Eddy Simulation, and Detached Eddy Simulation Interfaces
- Theory for the Large Eddy Simulation Interfaces
- Theory for the Detached Eddy Simulation Interfaces

Theory Background for the Pressure Acoustics Branch

This section describes the governing equations and the mathematical formulation of the governing equations as used in the Pressure Acoustics branch of the Acoustics Module. Details are also given regarding some of the boundary conditions, including the radiation boundary conditions as well as the exterior field calculation feature. The section starts with a general introduction to the governing equations used in pressure acoustics.

In this sections:

- The Governing Equations
- Pressure Acoustics, Frequency Domain Equations
- Pressure Acoustics, Transient Equations
- The Nonlinear Westervelt Equation
- Pressure Acoustics, Boundary Mode Equations
- Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions
- Theory for the Exterior Field Calculation: The Helmholtz–Kirchhoff Integral

The Governing Equations

Pressure acoustic problems involve solving for the small acoustic pressure variations p (also named p' or p_1) on top of the stationary background pressure p_0. Mathematically, this represents a linearization (small parameter expansion) of the dependent variables around the stationary quiescent values.

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

$$\begin{aligned} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= M \\ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} &= -\frac{1}{\rho}\nabla p + \mathbf{F} \\ \frac{\partial s}{\partial t} + \nabla \cdot (s\mathbf{u}) &= 0 \end{aligned} \tag{2-14}$$

where ρ is the total density, p is the total pressure, $\mathbf{u}$ is the velocity field, s is the entropy, M and $\mathbf{F}$ represent possible source terms. In classical pressure acoustics, all thermodynamic processes are assumed to be reversible and adiabatic, that is, isentropic processes. The small parameter expansion is performed on a stationary fluid ($\mathbf{u}_0 = \mathbf{0}$) of density ρ_0 (SI unit: kg/m^3) and at pressure p_0 (SI unit: Pa) such that:

$$\begin{aligned} p &= p_0 + p_1 \\ \rho &= \rho_0 + \rho_1 \\ \mathbf{u} &= \mathbf{0} + \mathbf{u}_1 \\ s &= s_0 + s_1 \end{aligned} \qquad \text{with} \qquad \begin{aligned} p_1 &\ll p_0 \\ \rho_1 &\ll \rho_0 \\ |\mathbf{u}_1| &\ll c \end{aligned}$$

where the variables with subscript 1 represent the small acoustic variations (the 1st order expansion). Assuming the initial entropy s_0 to be zero, then it will remain and $s_0 = 0$. Inserting these values into the governing equations and only retaining terms linear in the acoustic variables yields

$$\begin{aligned} \frac{\partial \rho_1}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u}_1) &= M \\ \frac{\partial \mathbf{u}_1}{\partial t} &= -\frac{1}{\rho_0}\nabla p_1 + \mathbf{F} \\ \frac{\partial p_1}{\partial t} &= c_s^2\left(\frac{\partial \rho_1}{\partial t} + \mathbf{u}_1 \cdot \nabla \rho_0\right) \end{aligned} \tag{2-15}$$

where c_s is recognized as the (isentropic) speed of sound (SI unit: m/s) at constant entropy s. The last equation for the pressure time differential is derived from the entropy equation, for more details see Theory Background for the Thermoviscous Acoustics Branch. For constant material parameters the last equation reduces to the usual relation

$$p_1 = c_s^2 \rho_1$$

This expression gives a useful condition that needs to be fulfilled for the linear acoustic equations to hold:

$$|p_1| \ll \rho_0 c_s^2$$

The subscript "s" is dropped in the following along with the subscript "0" on the background density ρ_0. Finally, rearranging Equation 2-15, renaming the source terms, and dropping the subscript "1" yields the wave equation for pressure waves in a lossless medium

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p - \mathbf{q}_\mathrm{d})\right) = Q_\mathrm{m}. \tag{2-16}$$

Here ρ (SI unit: $\mathrm{kg/m^3}$) refers to the density, and c (SI unit: m/s) denotes the speed of sound. In this general formulation of the scalar wave equation, the speed of sound and density may in general be space dependent, for example, through their dependency on the background temperature. The equation includes two optional source terms:

- The *monopole domain source* Q_m (SI unit: $1/\mathrm{s^2}$). This source corresponds to a mass source on the right-hand side of the continuity equation.
- The *dipole domain source* $\mathbf{q}_\mathrm{d}$ (SI unit: $\mathrm{N/m^3}$). This source corresponds to a domain force source on the right-hand side of the momentum equation.

The combination ρc^2 is called the *adiabatic bulk modulus*, commonly denoted K_s (SI unit: Pa). The bulk modulus is equal to the one over the adiabatic compressibility coefficient $\beta_\mathrm{s} = 1/K_\mathrm{s} = 1/\rho c^2$ (SI unit: 1/Pa).

In Equation 2-16 both the speed of sound $c = c(\mathbf{x})$ and the density $\rho = \rho(\mathbf{x})$ can depend on the spatial coordinates $\mathbf{x}$ while they are independent of time, or only slowly varying in time (that is, at a time scale much slower than the variations in the acoustic signal). If both domain sources are set to zero and the density is constant in space, you can recover the standard wave equation

$$\frac{\partial^2 p}{\partial t^2} = c^2 \nabla^2 p$$

Some classical references on acoustics for further reading are found in Ref. 4, Ref. 5, Ref. 6, Ref. 7, and Ref. 8.

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

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

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

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p - \mathbf{q}_d)\right) - \frac{\omega^2 p}{\rho_c c_c^2} = Q_m . \tag{2-17}$$

In this equation, the subscript "c" on the density and the speed of sound refers to that they may be complex valued. Lossy media, like porous materials or highly viscous fluids, can be modeled by using the complex-valued speed of sound and density. A selection of such fluid models is available in The Pressure Acoustics, Frequency Domain Interface. The attenuation in these fluid models is frequency dependent in different ways, depending on the physical origin of the damping. A description of the different fluid models is given in Theory for the Equivalent Fluid Models.

In the time domain, only certain frequency dependencies can be modeled, which limits the number of fluid models that can be used in The Pressure Acoustics, Transient Interface. One way to model damping in the time domain is to introduce an additional term of first order in the time derivative to account for attenuation of the sound waves

$$\frac{1}{\rho c^2}\frac{\partial^2 p}{\partial t^2} - d_a\frac{\partial p}{\partial t} + \nabla \cdot \left(-\frac{1}{\rho}(\nabla p - \mathbf{q}_d)\right) = Q_m \tag{2-18}$$

The damping term in Equation 2-18 is absent from the standard PDE formulations in the Pressure Acoustics, Transient interface, but it corresponds to a monopole domain source proportional to the time derivative of the pressure. This approach is, however, not used in the viscous and thermally conducting fluid models that contain damping for transient acoustics (see Viscous Fluid Model, Thermally Conducting Fluid Model, and Thermally Conducting and Viscous Fluid Model). The physical origin of the damping corresponds here to a dipole-like source.

Even when sound waves propagate in a lossless medium, attenuation can occur by interaction with the surroundings at the system boundaries. In particular, this applies to the impedance boundary conditions.

Alternatively, treat the Helmholtz Equation 2-17 as an eigenvalue PDE to solve for eigenmodes and eigenfrequencies, see the Eigenfrequency and Mode Analysis in 2D and 1D axisymmetric sections below.

In order to solve the governing equations, boundary conditions are necessary. Typical boundary conditions used in acoustics are:

- Sound Hard Boundary (Wall)
- Sound Soft Boundary (zero acoustic pressure)
- Specified acoustic Pressure
- Specified Normal Acceleration
- Impedance boundary conditions (see also Theory for the Boundary Impedance Models)
- Plane Wave Radiation
- Spherical Wave Radiation
- Cylindrical Wave Radiation

Pressure Acoustics, Frequency Domain Equations

The Pressure Acoustics, Frequency Domain Interface exists for several types of studies. Here the equations are presented for the frequency domain, eigenfrequency, and mode analysis studies. All the interfaces solve for the acoustic pressure p. It is available in all space dimensions — for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

FREQUENCY DOMAIN

The frequency domain, or time-harmonic, formulation uses the inhomogeneous Helmholtz equation:

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2 p_t}{\rho_c} = Q_m \tag{2-19}$$

This is Equation 2-17 repeated with the introduction of the wave number k_{eq} used in the equations. It contains both the ordinary wave number k as well as out-of-plane and azimuthal (or circumferential) contributions, when applicable. Note also that the pressure is here the total pressure p_t which is the sum of a possible Background Pressure Field p_b and the scattered field p_s. This enables for a so-called scattered field formulation of the equations. If no background field is present $p_t = p_s = p$.

In this equation, $p = p(\mathbf{x},\omega) = p(\mathbf{x})e^{i\omega t}$ (the dependence on ω is henceforth not explicitly indicated). Compute the frequency response by doing a parametric sweep over a frequency range using harmonic loads and sources.

When there is damping, ρ_c and c_c are complex-valued quantities. The available damping models and how to apply them is described in the sections Pressure Acoustics and Theory for the Equivalent Fluid Models.

Equation 2-19 is the equation that the software solves for 3D geometries. In lower-dimensional and axisymmetric cases, restrictions on the coordinate dependence mean that the equations differ from case to case. Here is a brief summary of the situation.

2D

In 2D, the pressure is of the form

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

which inserted in Equation 2-19 gives

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) - \frac{k_{eq}^2}{\rho_c} p_t = Q_m \qquad (2\text{-}20)$$

$$k_{eq}^2 = \frac{\omega^2}{c_c^2} - k_z^2$$

The *out-of-plane wave number* k_z can be set on the Pressure Acoustics page. By default its value is 0. In the mode analysis study $-ik_z$ is used as the eigenvalue λ.

2D Axisymmetry

For 2D axisymmetric geometries the independent variables are the radial coordinate r and the axial coordinate z. The only dependence allowed on the azimuthal coordinate φ is through a phase factor,

$$p(r,\varphi,z) = p(r,z)e^{-im\varphi} \qquad (2\text{-}21)$$

where m denotes the *azimuthal mode number*. The mode number defines an *azimuthal wave number* $k_m = m/r$. Because the azimuthal coordinate is periodic, m must be an integer. Just like k_z in the 2D case, m can be set in the Settings window for Pressure Acoustics.

As a result of Equation 2-21, the equation to solve for the acoustic pressure in 2D axisymmetric geometries becomes

$$\frac{\partial}{\partial r}\left[-\frac{r}{\rho_c}\left(\frac{\partial p}{\partial r}-q_r\right)\right]+r\frac{\partial}{\partial z}\left[-\frac{1}{\rho_c}\left(\frac{\partial p}{\partial z}-q_z\right)\right]-\frac{k_{eq}^2}{\rho_c}rp = rQ_m$$

$$k_{eq}^2 = \left(\frac{\omega}{c_c}\right)^2 - k_m^2 \qquad k_m = \frac{m}{r}$$

1D Axisymmetry

In 1D axisymmetric geometries,

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

leading to the radial equation

$$\frac{\partial}{\partial r}\left[-\frac{r}{\rho_c}\left(\frac{\partial p}{\partial r}-q_r\right)\right]-\frac{k_{eq}^2}{\rho_c}rp = rQ_m$$

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

where both the *azimuthal wave number* k_m and the *axial wave number* k_z, appear as parameters.

1D

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

$$\frac{d}{dx}\left(-\frac{1}{\rho_c}\left(\frac{dp}{dx}-q_d\right)\right)-\frac{k_{eq}^2}{\rho_c}p = Q_m$$

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

EIGENFREQUENCY

In the eigenfrequency formulation, the source terms are absent; the eigenmodes and eigenfrequencies are solved for:

$$\nabla\cdot\left(-\frac{1}{\rho_c}\nabla p\right)+\frac{\lambda^2 p}{\rho_c c_c^2} = 0 \tag{2-22}$$

The eigenvalue λ introduced in this equation is related to the eigenfrequency f, and the angular frequency ω, through $\lambda = -i2\pi f = -i\omega$. In the user interface the variable `lambda` is a reserved variable. Because they are independent of the pressure, the solver

ignores any dipole and monopole sources unless a coupled eigenvalue problem is being solved.

Equation 2-22 applies to the 3D case. The equations solved in eigenfrequency studies in lower dimensions and for axisymmetric geometries are obtained from their time-harmonic counterparts, given in the previous subsection, by the substitution $\omega^2 \rightarrow -\lambda^2$.

Vibrations of a Disk Backed by an Air-Filled Cylinder: Application Library path **Acoustics_Module/Verification_Examples/coupled_vibrations**

MODE ANALYSIS IN 2D AND 1D AXISYMMETRIC

See Mode Analysis Study in the Pressure Acoustics, Boundary Mode Equations section. The mode analysis study type is only available for the Pressure Acoustics, Frequency Domain interface in 2D and 1D axisymmetric components, where the solver solves for the eigenvalues $\lambda = -ik_z$ for a given frequency. Here k_z is the out-of-plane wave number of a given mode and the resulting pressure field p represents the mode on the cross section of an infinite waveguide or duct.

Pressure Acoustics, Transient Equations

Use the Time Dependent study type to model transient acoustic phenomena in a stationary fluid and to solve the wave equation

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

for the acoustic pressure, $p = p(\mathbf{x}, t)$. Here c is the speed of sound and ρ denotes the equilibrium density, while $\mathbf{q}_\mathrm{d}$ and Q_m are dipole and monopole sources, respectively. The density and speed of sound can both be nonconstant in space. In contrast, they are assumed to vary with time on scales much larger than the period for the acoustic waves and are therefore considered time independent in the previous equation. This physics interface is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

The Nonlinear Westervelt Equation

To extend the validity of the small perturbation wave equation to finite amplitude waves, nonlinear effects need to be taken into account. The first idea is to expand the

density pressure relation to second order to include the effect of finite amplitude waves. This gives

$$p_1 = c_0^2\rho_1 + \frac{c_0^2}{\rho_0}\frac{B}{2A}\rho_1^2 + \ldots \tag{2-23}$$

where we have kept the subscript "1" to highlight the acoustic perturbation quantities and a subscript "0" for the quiescent quantities. See Ref. 6 and 43 for details. The quantity B/A is the parameter of nonlinearity (see Ref. 52). This expression basically means that the different parts of the acoustic pressure wave travel at different speeds.

Inserting the above into the governing Equation 2-14 (retaining the dissipative terms) will lead to the full nonlinear second-order wave equation. This is the nonlinear equivalent to Equation 2-15 (or actually the nonlinear equivalent to the full equations of thermoviscous acoustics). Next assume that boundary layer effects can be disregarded and cumulative nonlinear effects dominate local nonlinear effects, for example, when the propagation distance is greater than the wavelength. This leads to the well-known Westervelt equation (see Ref. 43)

$$\frac{1}{\rho c^2}\frac{\partial^2 p_t}{\partial t^2} + \nabla\cdot\left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d) - \frac{\delta}{\rho c^2}\frac{\partial \nabla p_t}{\partial t}\right) = \frac{\beta}{\rho^2 c^4}\frac{\partial^2 p_t^2}{\partial t^2} + Q_m \tag{2-24}$$

Here we have eliminated the density fluctuations and then removed the subscripts. Hence, p_t is the (total) acoustic pressure, ρ is the quiescent density, δ is the diffusivity of sound, and β is the coefficient of nonlinearity (related to the parameter of nonlinearity B/A).

The Westervelt equation supports the propagation of shocks. Shocks can be hard to capture numerically and require the addition of stabilization. This can, for example, be done by adding diffusivity near the shock location. One method (used in COMSOL) is to add an effective diffusivity term that reads

$$-\delta\kappa\left(1 - \left|\frac{\partial}{\partial t}\nabla p\right|^{q-1}\right)$$

The highest artificial damping is achieved where the acoustic pressure increases or decreases the most rapidly. It reaches its maximal values where the pressure endures discontinuities, that is, where shocks arise. Thus this technique provides a shock-capturing stabilization. See Ref. 53 for more details. Specify the so-called q-Laplacian exponent q and the so-called q-Laplacian factor κ to get the desirable amount of artificial damping. Note that the damping must not be too high nor too

low. The particular values of q and κ depend on the material and the input signal frequency. The two parameters that control the stabilization require manual tuning. A suggested approach is to use a simple 1D model to tune the parameters based on fluid material properties and frequency content.

Nonlinear Acoustics — Modeling of the 1D Westervelt Equation: Application Library path **Acoustics_Module/Nonlinear_Acoustics/nonlinear_acoustics_westervelt_1d**

Pressure Acoustics, Boundary Mode Equations

When an acoustic wave of a given angular frequency ω is fed into a waveguide or a duct, only a finite set of shapes, or modes, for the transverse pressure field can propagate over long distances inside the structure. The higher the frequency, the higher the number of sustainable modes.

Take, as an example, a uniform straight duct whose axis is in the z direction. The acoustic field in such a duct can be written as a sum of the form

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

The constant k_{zj} is the axial wave number of the jth propagating transverse mode, $p_j(x,y)$. These transverse modes and their associated axial wave numbers are solutions to an eigenvalue problem defined on the duct's cross section. The mode analysis capabilities in The Pressure Acoustics, Boundary Mode Interface makes it possible to solve such eigenvalue problems. The physics interface is available for 3D Cartesian and 2D axisymmetric geometries and solves for the transverse eigenmodes for the acoustic pressure p and the associated propagation constants k_z. The Mode Analysis Study is briefly discussed.

MODE ANALYSIS STUDY

The eigenvalue solver computes a specified number of solutions $\{p_j, \lambda_j\}$ to the equation

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\nabla p - \mathbf{q}_d)\right) - \left(\frac{\omega^2}{\rho_c c_c^2} - \frac{k_n^2}{\rho_c}\right)p = Q_m \qquad (2\text{-}25)$$

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

The out-of-plane wave number is denoted k_n, and is in the normal direction to the two-dimensional surface on which Equation 2-25. As for a mode analysis study in the frequency domain the propagation direction does not necessarily have to be normal to the z-axis for 3D geometries.

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

Notice that the above equation is identical to the time-harmonic equation for pressure acoustics, except that k_n is interpreted as an eigenvalue and not as a parameter.

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

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

Here m, the *azimuthal mode number*, is an integer-valued parameter. The equation is defined on the interval $r_1 < r < r_2$. The eigenvalue λ is defined in terms of the axial wave number k_z through the equation $\lambda = -ik_z$.

Theory for the Plane, Spherical, and Cylindrical Radiation Boundary Conditions

Specify a Plane Wave Radiation, Spherical Wave Radiation, or Cylindrical Wave Radiation boundary condition to allow an outgoing wave to leave the modeling domain with minimal reflections. The condition can be adapted to the geometry of the modeling domain. The *plane wave* type is suitable for both far-field boundaries and ports (for plane waves only). For general radiation boundary conditions for waveguides (supporting multimodes) it is recommended to use the Port boundary condition.

Radiation boundary conditions are available for all types of studies. For the frequency-domain study, Givoli and Neta's reformulation of the Higdon conditions (Ref. 1) for plane waves has been implemented to the second order. For cylindrical and spherical waves, COMSOL Multiphysics uses the corresponding 2nd-order expressions from Bayliss, Gunzburger, and Turkel (Ref. 2). The Transient, Mode analysis, and Eigenfrequency studies implement the same expansions to the first order.

The first-order radiation boundary conditions in the frequency domain read

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)+(ik+\kappa(r))\frac{p}{\rho_c}=(ik+\kappa(r))\frac{p_i}{\rho_c}+\mathbf{n}\cdot\left(\frac{\nabla p_i}{\rho_c}\right)$$

where k is the wave number and $\kappa(r)$ is a function whose form depends on the wave type:

- Plane wave: $\kappa(r)=0$
- Cylindrical wave: $\kappa(r)=1/(2r)$
- Spherical wave: $\kappa(r)=1/r$

In the cylindrical and spherical wave cases, r is the shortest distance from the point $\mathbf{r}=(x,y,z)$ on the boundary to the source. The right-hand side of the equation represents an optional incoming pressure field p_i (see Incident Pressure Field).

The second-order radiation boundary conditions in the frequency domain are defined below. In these equations, $\Delta_{||}$ at a given point on the boundary denotes the Laplace operator in the tangent plane at that particular point.

PLANE WAVE

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)+i\frac{k}{\rho_c}p+\frac{i}{2k\rho_c}\Delta_{||}p=\frac{i}{2k\rho_c}\Delta_{||}p_i+i\frac{k}{\rho_c}p_i+\mathbf{n}\cdot\frac{1}{\rho_c}\nabla p_i$$

In the notation of Givoli and Neta (Ref. 1), the above expressions correspond to the parameter choices $C_0=C_1=C_2=\omega/k$. For normally incident waves, this gives a vanishing reflection coefficient.

CYLINDRICAL WAVE

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

$$\left(ik_{eq}+\frac{1}{2r}-\frac{1}{8r(1+ik_{eq}r)}\right)\frac{(p_i-p)}{\rho_c}+\mathbf{n}\cdot\frac{1}{\rho_c}\nabla p_i+\frac{(r\Delta_{||}p_i-r\Delta_{||}p)}{2(1+ik_{eq}r)\rho_c}$$

The *cylindrical wave* boundary condition is based on a series expansion of the outgoing wave in cylindrical coordinates (Ref. 2), and it assumes that the field is independent of the axial coordinate. Specify the axis of this coordinate system by giving an orientation (n_x, n_y, n_z) and a point (x_0, y_0, z_0) on the axis. In axisymmetric geometries, the symmetry axis is the natural and only choice.

SPHERICAL WAVE

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) + \left(ik_{\text{eq}} + \frac{1}{r}\right)\frac{p}{\rho_c} - \frac{r\Delta_{\|}p}{2\rho_c(ik_{\text{eq}}r+1)}$$
$$= -\frac{r\Delta_{\|}p_{\text{i}}}{2\rho_{0c}(ik_{\text{eq}}r+1)} + \left(ik_{\text{eq}} + \frac{1}{r}\right)\frac{p_{\text{i}}}{\rho_c} + \mathbf{n}\cdot\frac{1}{\rho_c}\nabla p_{\text{i}}$$

Use a *spherical wave* to allow a radiated or scattered wave — emanating from an object centered at the point (x_0, y_0, z_0) that is specified — to leave the modeling domain without reflections. The boundary condition is based on an expansion in spherical coordinates from Bayliss, Gunzburger, and Turkel (Ref. 2), implemented to the second order.

TRANSIENT ANALYSIS

The transient radiation boundary condition is the first-order expression

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho}(\nabla p_t - \mathbf{q}_d)\right) + \frac{1}{\rho}\left(\frac{1}{c}\frac{\partial p}{\partial t} + \kappa(r)p\right) = \frac{1}{\rho}\left(\frac{1}{c}\frac{\partial p_{\text{i}}}{\partial t} + \kappa(r)p_{\text{i}} + \mathbf{n}\cdot\nabla p_{\text{i}}\right)$$

where $\kappa(r)$ is the same wave-type dependent function as for the eigenfrequency case and p_{i} the optional Incident Pressure Field.

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

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

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

Theory for the Exterior Field Calculation: The Helmholtz–Kirchhoff Integral

The Acoustics Module has functionality to evaluate the acoustic pressure field in the exterior field region of the model (outside the computational domain). This section gives the relevant definitions and mathematical background as well as some general advice for analyzing the exterior field. Details about how to use the exterior field functionality is described in Exterior Field Calculation and in the modeling section Evaluating the Acoustic Field in the Exterior: Near- and Far-Field.

THE HELMHOLTZ–KIRCHHOFF INTEGRAL REPRESENTATION

In many cases, solving the acoustic Helmholtz equation everywhere in the domain where results are requested is neither practical nor necessary. For homogeneous media, the solution anywhere outside a closed surface containing all sources and scatterers can be written as a boundary integral in terms of quantities evaluated on the surface. To evaluate this *Helmholtz–Kirchhoff* integral, it is necessary to know both Dirichlet and Neumann values on the surface. Applied to acoustics, this means that if the pressure *and* its normal derivative (which is related to the normal velocity) is known on a closed surface, the acoustic field can be calculated at any point outside.

In general, the solution p to Helmholtz' equation

$$-\nabla \cdot \nabla p - k^2 p = 0$$

in the homogeneous domain exterior to a closed surface, S, can be explicitly expressed in terms of the values of p and its normal derivative on S:

$$p(\mathbf{R}) = \int_S (G(\mathbf{R}, \mathbf{r})\nabla p(\mathbf{r}) - \nabla G(\mathbf{R}, \mathbf{r}) p(\mathbf{r})) \cdot \mathbf{n}\, dS$$

Here the coordinate vector $\mathbf{r}$ parameterizes S. The unit vector $\mathbf{n}$ is the outward normal to the exterior infinite domain; thus, $\mathbf{n}$ points *into* the domain that S encloses. The function $G(\mathbf{R}, \mathbf{r})$ is a Green's function satisfying

$$-\nabla \cdot \nabla G(\mathbf{R}, \mathbf{r}) - k^2 G(\mathbf{R}, \mathbf{r}) = \delta^{(3)}(\mathbf{R} - \mathbf{r})$$

This essentially means that the Green's function, seen as a function of $\mathbf{r}$, is an outgoing traveling wave excited by a simple source at $\mathbf{R}$. In 3D, the Green's function is therefore:

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

In 2D, the Green's function contains a Hankel function instead of the exponential:

$$G(\mathbf{R}, \mathbf{r}) = \frac{i}{4}H_0^{(2)}(k|\mathbf{r}-\mathbf{R}|)$$

Inserting the 3D Green's function in the general representation formula gives:

$$p(\mathbf{R}) = \frac{1}{4\pi}\int_S \frac{e^{-ik|\mathbf{r}-\mathbf{R}|}}{|\mathbf{r}-\mathbf{R}|}\left(\nabla p(\mathbf{r}) + p(\mathbf{r})\frac{(1+ik|\mathbf{r}-\mathbf{R}|)}{|\mathbf{r}-\mathbf{R}|^2}(\mathbf{r}-\mathbf{R})\right)\cdot \mathbf{n}\, dS \qquad (2\text{-}26)$$

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

$$p(\mathbf{R}) = -\frac{i}{4}\int_S \left(H_0^{(2)}(k|\mathbf{r}-\mathbf{R}|)\nabla p(\mathbf{r}) + kp(\mathbf{r})\frac{H_1^{(2)}(k|\mathbf{r}-\mathbf{R}|)}{|\mathbf{r}-\mathbf{R}|}(\mathbf{r}-\mathbf{R})\right)\cdot \mathbf{n}\, dS \qquad (2\text{-}27)$$

For axially symmetric geometries, the full 3D integral must be evaluated. The Acoustics Module uses an adaptive numerical quadrature in the azimuthal direction on a fictitious revolved geometry in addition to the standard mesh-based quadrature in the rz-plane.

The default in the Exterior Field Calculation feature is to evaluate the full Helmholtz–Kirchhoff integral given in Equation 2-26 and Equation 2-27.

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 very large distances. At the same time, in some applications, the quantity of interest is the far-field radiation pattern, which can be defined as the limit of $r|p|$ when r goes to infinity in a given direction.

Taking the limit of Equation 2-26 when $|\mathbf{R}|$ goes to infinity and ignoring the rapidly oscillating phase factor, the far field, p_{far} is defined as

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

The relevant quantity is $|p_{\text{far}}|$ rather than p_{far} because the phase of the latter is undefined. For the same reason, only the direction of $\mathbf{R}$ is important, not its magnitude.

Because Hankel functions asymptotically approach an exponential, the limiting 2D integral is remarkably similar to that in the 3D case:

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

For axially symmetric geometries, the azimuthal integral of the limiting 3D case can be handled analytically, which leads to a rather complicated expression but avoids the numerical quadrature required in the general case. For zero azimuthal mode number $m = 0$, the expression is:

$$p_{\text{far}}(\mathbf{R}) \cong -\frac{1}{2}\int_S re^{ik\frac{zZ}{|\mathbf{R}|}}\left[J_0\left(\frac{krR}{|\mathbf{R}|}\right)\nabla p(\mathbf{r})\cdot\mathbf{n} - \frac{ikp(\mathbf{r})}{|\mathbf{R}|}\left(in_r RJ_1\left(\frac{krR}{|\mathbf{R}|}\right) + n_z ZJ_0\left(\frac{krR}{|\mathbf{R}|}\right)\right)\right]dS \tag{2-28}$$

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, set the **Type of integral** option to the **Far-field integral approximation for $r \to \infty$** option in the **Exterior Field Calculation** section in the **Settings** window for the feature. See Exterior Field Calculation.

THE ELKERNEL ELEMENT

These integrals can be implemented as integration coupling variables in COMSOL Multiphysics. However, such an approach is very inefficient because then the simple structure of the integration kernels cannot be exploited. In the Acoustics Module, convolution integrals of this type are therefore evaluated in optimized codes that hides all details from the user.

Theory for the Boundary Impedance Models

In this section:

- Impedance Conditions
- RCL Models
- Physiological Models
- Waveguide End Impedance Models
- Porous Layer Models
- Characteristic Specific Impedance Models

Impedance Conditions

An impedance boundary condition relates the acoustic pressure and the acoustic velocity to each other at a given point. This relationship is provided by knowledge about the conditions on the boundary, for example, a specific material or an acoustically active boundary. Therefore, an impedance boundary condition can be used to impose the properties of the boundary without modeling it explicitly. Impedance boundary conditions thus generalize the sound-hard and sound-soft boundary conditions to address a large number of cases between these two extremes.

Mathematically, a specific acoustic impedance Z_n is defined on some cross section as the ratio between the acoustic pressure p and the acoustic velocity perpendicular to the area $v_\perp$ (the normal velocity)

$$Z_n = \frac{p}{v_\perp}.$$

In Pressure Acoustics, Frequency Domain, this boundary condition is imposed as

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t - \mathbf{q}_d)\right) = -\frac{i\omega p_t}{Z_n} \tag{2-29}$$

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

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p-\mathbf{q}_d)\right)=\frac{1}{Z_{\mathrm{n}}}\frac{\partial p_{\mathrm{t}}}{\partial t} \tag{2-30}$$

Here Z_{n} is the specific acoustic (input) impedance of the external domain and it has the unit of a specific acoustic impedance. The specific acoustic impedance Z_{n} (SI unit: Pa·s/m) is related to the acoustic impedance Z_{ac} (ratio of pressure and flow rate, SI unit: Pa·s/m^3) and the mechanical impedance Z_{mech} (ratio of force and velocity, SI unit: N·s/m) via the area A of the boundary, according to

$$Z_{\mathrm{mech}}=AZ_{\mathrm{i}}=A^2Z_{\mathrm{ac}} \tag{2-31}$$

Impedance boundary conditions only relate the normal velocity (the velocity perpendicular to the boundary) to the pressure, but do not consider the tangential velocity (component parallel to the boundary). This is due to the mathematical construction of the governing equation and the fact that pressure acoustics solves only for the scalar pressure. Put differently, the impedance boundary condition only applies to the normal component of the incident field. Thus, by applying an impedance boundary condition this tangential velocity component is ignored altogether. For this reason, impedance boundary conditions are in most cases low-order approximations to the actual boundary properties. In cases where this is unacceptable, it is consequently better to either model the boundary explicitly or use a higher-order model, such as for instance the Plane Wave Radiation at an open boundary.

RCL Models

Circuit models are used as low-order models of acoustically active boundaries specifying the input impedance of the external domain. For example, modeling mechanical properties of a loudspeaker cone, a microphone diaphragm, or biological tissue, as experienced by the incident wave. For such boundaries, the pressure and velocity are modulated in response to the behavior of the boundary.

An RCL model is intended to provide a simple model to represent the frequency-dependent mechanical properties of a system that typically has some loss, a mass, and a compliance. The option includes all combinations of a three-element circuit consisting of an acoustic damper (acoustic resistance, a resistor R_{ac}), an acoustic mass (acoustic inertance, an inductor L_{ac}) and an acoustic compliance (a capacitor C_{ac}), which are the simplest resonating acoustic circuits. While full circuit models tend to be more sophisticated (see for example Ref. 26), one can typically identify a simple core circuit responsible for the basic response which consists of such three elements.

The acoustic impedance Z_{ac} is related to the specific impedance Z_i implemented in the weak equations Equation 2-29 and Equation 2-30 by the area of the boundary A as

$$Z_i = AZ_{ac}.$$

CIRCUIT MODEL OPTIONS

A schematic illustration is provided for each circuit in the Equation Display window.

Serial Coupling of R_{ac}, C_{ac}, and L_{ac}

$$Z_{ac} = R_{ac} + i\omega L_{ac} + \frac{1}{i\omega C_{ac}}$$

Parallel Coupling of R_{ac}, C_{ac}, and L_{ac}

$$Z_{ac} = \left[i\omega C_{ac} + \frac{1}{R_{ac}} + \frac{1}{i\omega L_{ac}}\right]^{-1}$$

Parallel Coupled L_{ac} and C_{ac} in Series with R_{ac}

$$Z_{ac} = R_{ac} + \left[i\omega C_{ac} + \frac{1}{i\omega L_{ac}}\right]^{-1}$$

Parallel Coupled R_{ac} and C_{ac} in Series with L_{ac}

$$Z_{ac} = i\omega L_{ac} + \left[\frac{1}{R_{ac}} + i\omega C_{ac}\right]^{-1}$$

Parallel Coupled R_{ac} and L_{ac} in Series with C_{ac}

$$Z_{ac} = \frac{1}{i\omega C_{ac}} + \left[\frac{1}{R_{ac}} + \frac{1}{i\omega L_{ac}}\right]^{-1}$$

Serial Coupled R_{ac} and C_{ac} in Parallel with L_{ac}

$$Z_{ac} = \left[\frac{1}{i\omega L_{ac}} + \left(\frac{1}{i\omega C_{ac}} + R_{ac}\right)^{-1}\right]^{-1}$$

Serial Coupled L_{ac} and C_{ac} in Parallel with R_{ac}

$$Z_{ac} = \left[\frac{1}{R_{ac}} + \left(i\omega L_{ac} + \frac{1}{i\omega C_{ac}}\right)^{-1}\right]^{-1}$$

Serial Coupled R_{ac} and L_{ac} in Parallel with C_{ac}

$$Z_{ac} = [i\omega C_{ac} + (i\omega L_{ac} + R_{ac})^{-1}]^{-1}$$

LIMITATIONS

A circuit model is one dimensional and considers only changes in the direction normal to the boundary; any variations across the boundary are ignored. Therefore, circuit models provide good results at low frequencies with wavelengths substantially larger than the dimensions of the boundary.

ACOUSTICAL OR MECHANICAL CIRCUITS

It is sometimes advantageous to use circuit models in mechanical units rather than acoustical units. In these cases, the overall structure of the impedance boundary condition equation does not change, but the numerical value of the parameters do. The mechanical impedance Z_{mech} (the ratio of force F and velocity v) and the acoustic impedance Z_{ac} are related via the boundary surface area A by the expression

$$Z_{\mathrm{mech}} = A^2 Z_{\mathrm{ac}} .$$

Physiological Models

The physiological models are simple equivalent circuit models for parts of the human body which typically are of importance in acoustical applications, namely the skin, the full human ear, the eardrum and inner ear, and the acoustic impedance from the ear's pinna. These models are good approximations to the active, acoustical properties of these body parts with particular relevance for development of hearing aids, mobile devices, and head phones.

HUMAN SKIN MODEL

This model has been developed as a lowest-order, reasonable approximation to experimental data in Ref. 27 (see their Figure 9). It consists of a simple serial RCL circuit applied on a transducer area of A_{t}, and has the impedance

$$Z_{\mathrm{n}} = \frac{R_{\mathrm{s}}}{A_{\mathrm{t}}} + \frac{i\omega L_{\mathrm{s}}}{A_{\mathrm{t}}} + \frac{1}{i\omega C_{\mathrm{s}} A_{\mathrm{t}}}$$

with $A_{\mathrm{t}} = 1.5\ \mathrm{cm}^2$, $R_{\mathrm{s}} = 9.0\ \mathrm{N{\cdot}s{\cdot}m^{-1}}$, $L_{\mathrm{s}} = 0.53{\cdot}10^{-3}\ \mathrm{N{\cdot}s^2{\cdot}m^{-1}}$ and $C_{\mathrm{s}} = 5.3{\cdot}10^{-6}\ \mathrm{m\ N^{-1}}$.

MODELS RELATED TO THE HUMAN EAR

Four models related to the human ear have been included. These detailed and experimentally-verified circuit models describe parts of the human ear, as well as the entire human ear, see Ref. 28 to 30 for further details.

Outward Human Ear Radiation

This impedance describes the acoustic radiation losses from the pinna (also known as the auricle, this is the visible part of the ear which is exterior to the head), see Figure 2-14. For cases where you model the ear canal explicitly using pressure acoustics, this boundary condition describes the acoustic losses from the outward acoustic radiation from the ear canal and into the surrounding air.

$$Z_{\text{rad}} = \left[Y_{\text{par}} + \sum_{k=1}^{3} Y_k \right]^{-1}, \qquad (2\text{-}32)$$

$$Y_{\text{par}} = \frac{1}{R_{\text{par}}} + \frac{1}{i\omega L_{\text{par}}} + i\omega C_{\text{par}} \qquad Y_k = R_k\left(1 + iQ\left[\frac{\omega}{\omega_k} - \frac{\omega_k}{\omega}\right]\right)$$

where $R_{\text{par}} = 7.0 \cdot 10^6$ N·s·m^{-5}, $L_{\text{par}} = 100$ N·s^2·m^{-5}, $C_{\text{par}} = 1.7 \cdot 10^{-12}$ m^5·N^{-1}, $Q = 6$, $R_1 = R_2 = R_{\text{par}}$, $R_3 = 2R_{\text{par}}$, $\omega_1 = 6000 \cdot 2\pi$ Hz, $\omega_1 = 9000 \cdot 2\pi$ Hz, and $\omega_1 = 13{,}000 \cdot 2\pi$ Hz.

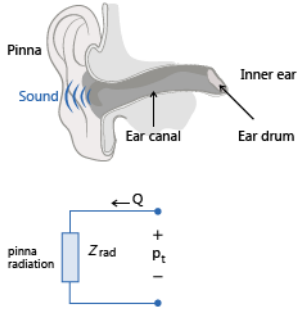


Figure 2-14: Illustration of outward human ear radiation.

Human Ear Drum Impedance

This model describes the impedance of the human ear drum and the entire inner ear, that is, the acoustic impedance experienced in the ear canal when looking into the ear

drum, see Figure 2-15. The model equations are given in Equation 2-33 and Equation 2-34, and the parameter values in Table 2-8.

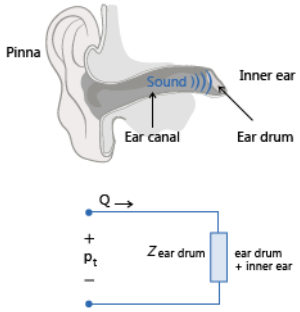


Figure 2-15: Illustration of human eardrum impedance.

$$Z_{\text{eardrum}} = Z_{\text{cav}} + \frac{k_{11}Z_{\text{sc}} + k_{12}}{k_{21}Z_{\text{sc}} + k_{22}} \qquad K = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix}$$

$$k_{11} = \frac{1}{A_{\text{D}}}(1 + Y_{\text{mi}}Z_{\text{dm}}) \qquad k_{12} = \frac{Z_{\text{dmi}}}{A_{\text{D}}}\left(1 + \frac{Y_{\text{mi}}Z_{\text{incus}}Z_{\text{dm}}}{Z_{\text{dmi}}}\right)$$

$$k_{21} = \frac{Y_{\text{ac}}}{A_{\text{D}}}(1 + Y_{\text{mi}}Z_{\text{dm}}) + A_{\text{D}}Y_{\text{mi}}$$

$$k_{22} = \frac{Y_{\text{ac}}Z_{\text{dmi}}}{A_{\text{D}}}\left(1 + \frac{Y_{\text{mi}}Z_{\text{incus}}Z_{\text{dm}}}{Z_{\text{dmi}}}\right) + A_D(1 + Y_{\text{mi}}Z_{\text{incus}})$$

$$A_{\text{D}}(\omega) = \begin{cases} A(\omega) & \text{if } \omega < \omega_{\text{Aph}} \\ |A(\omega)|e^{i\Phi_{\text{A}}(\omega)} & \text{if } \omega \geq \omega_{\text{Aph}} \end{cases} \qquad \Phi_{\text{A}}(\omega) = s_{\text{Aph}}\log\left(\frac{\omega}{\omega_{\text{Aph}}}\right) + \varphi_{\text{A}} \tag{2-33}$$

$$A(\omega) = \frac{A_0 + A_\infty}{1 - \left(\frac{\omega}{\omega_{\text{A}}}\right)^2 + \frac{i\omega}{Q_{\text{A}}\omega_{\text{A}}}} - A_\infty$$

$$Z_{\text{cav}} = \left[Y_{\text{tcav}} + \frac{1}{Z_{\text{ada}} + \frac{1}{Y_{\text{ant}}}}\right]^{-1} \qquad Y_{\text{tcav}} = \left[R_{\text{tcav}} + \frac{1}{i\omega C_{\text{tcav}}}\right]^{-1}$$

$$Z_{\mathrm{ada}} = R_{\mathrm{ada}} + i\omega L_{\mathrm{ada}} \qquad Y_{\mathrm{ant}} = i\omega C_{\mathrm{ant}} + \frac{i\omega C_{\mathrm{mac}}}{1 - \left(\frac{\omega}{\omega_{\mathrm{mac}}}\right)^2 + \frac{i\omega}{Q_{\mathrm{mac}}\omega_{\mathrm{mac}}}}$$

$$Y_{\mathrm{ac}} = \frac{1}{R_{\mathrm{ac}} + i\omega L_{\mathrm{ac}}(\omega) + \frac{1}{i\omega C_{\mathrm{ac}}}} e^{i\Phi_{\mathrm{Y}}(\omega)} \qquad L_{\mathrm{ac}}(\omega) = L_{\mathrm{ac}_0}\left(1 + \sqrt{\frac{\omega}{\omega_{Y_{L_{\mathrm{ac}}}}}}\right)$$

$$\Phi_{\mathrm{Y}}(\omega) = s_{\mathrm{Yph}} \log\left(1 + \frac{\omega}{\omega_{\mathrm{Yph}}}\right) \qquad Y_{\mathrm{mi}} = \left[R_{\mathrm{mi}} + \frac{1}{i\omega C_{\mathrm{mi}}}\right]^{-1} \tag{2-34}$$

$$Z_{\mathrm{dm}} = \frac{2}{3} Z_{\mathrm{dmi}} \qquad Z_{\mathrm{incus}} = \frac{1}{3} Z_{\mathrm{dmi}}$$

$$Z_{\mathrm{dmi}} = \frac{1}{i\omega C_{\mathrm{oss}}} + i\omega L_{\mathrm{oss}} + (Z_{\mathrm{cpl}} + Z_{\mathrm{free}})^{-1} \qquad Z_{\mathrm{cpl}} = \left[R_{\mathrm{cpl}} + \frac{1}{i\omega C_{\mathrm{cpl}}}\right]$$

$$Z_{\mathrm{free}} = [R_{\mathrm{free}} + i\omega L_{\mathrm{free}}]^{-1} \qquad Z_{\mathrm{st}} = R_{\mathrm{st}} + i\omega L_{\mathrm{st}} + \frac{1}{i\omega C_{\mathrm{st}}}$$

$$Z_{\mathrm{c}} = R_{\mathrm{c}} + i\omega L_{\mathrm{c}} + \frac{1}{i\omega C_{\mathrm{c}}} \qquad Z_{\mathrm{sc}} = Z_{\mathrm{st}} + A_F^2 Z_{\mathrm{c}}$$

The parameter values are given in Table 2-8. Note that the value of φ_A is not reported in the papers Ref. 28 to 30, but has instead been determined during model implementation. The value ensures a continuity of the phase response.

TABLE 2-8: LIST OF PARAMETERS FOR THE EAR DRUM IMPEDANCE.

SYMBOL	VALUE	DESCRIPTION
R_{tcav}	$2 \cdot 10^6$ N·s·m^{-5}	Tympanic cavity resistance
C_{tcav}	(0.5 cm^3)/(γp_A)	Tympanic cavity compliance
R_{ada}	$1.7 \cdot 10^6$ N·s·m^{-5}	Aditus ad antrum resistance
L_{ada}	880 N·s^2·m^{-5}	Aditus ad antrum inertance
C_{ant}	(0.8 cm^3)/(γp_A)	Antrum compliance
Q_{mac}	0.4	Mastoid air cell resonance quality factor
C_{mac}	(8 cm^3)/(γp_A)	Mastoid air cell compliance
ω_{mac}	$2\pi 3500$ rad s^{-1}	Mastoid air cell resonance angular frequency
R_{ac}	$4 \cdot 10^7$ N·s·m^{-5}	Ear drum shunt resistance
C_{ac}	$5 \cdot 10^{-12}$ N^{-1}·m^5	Ear drum shunt compliance
L_{ac_0}	$2.4 \cdot 10^3$ kg·m^{-4}	Zero-frequency ear drum shunt inertance
$\omega_{Y_{L_{\mathrm{ac}}}}$	$2\pi \cdot 1900$ rad·s^{-1}	Ear drum shunt inertance cross-over frequency
ω_{Yph}	$2\pi \cdot 8000$ rad·s^{-1}	Ear drum shunt admittance cross-over frequency

TABLE 2-8: LIST OF PARAMETERS FOR THE EAR DRUM IMPEDANCE.

SYMBOL	VALUE	DESCRIPTION
s_{Yph}	1.4	Ear drum shunt admittance decay slope
A_0	38 mm^2	Effective ear drum area, low frequency limit
A_∞	2 mm^2	Effective ear drum area, infinite frequency limit
ω_{A}	$2\pi \cdot 2200$ $rad \cdot s^{-1}$	Effective ear drum area resonance angular frequency
Q_{A}	1.3	Effective ear drum area resonance quality factor
s_{Aph}	-1.2	Effective ear drum area high-frequency decay slope
ω_{Aph}	$2\pi \cdot 1500$ $rad \cdot s^{-1}$	Effective ear drum area cross-over frequency
φ_{A}	-0.8038	Effective ear drum area lag phase constant
R_{mi}	1 $N \cdot s \cdot m^{-1}$	Mechanical resistance of incudomalleal joint
C_{mi}	0.04 $mm \cdot N^{-1}$	Mechanical compliance of incudomalleal joint
C_{oss}	$3 \cdot 10^{-3}$ $m \cdot N^{-1}$	Mechanical compliance of ossicles
L_{oss}	$7 \cdot 10^{-3}$ g	Mechanical inertance of ossicles
C_{cpl}	$0.5 \cdot 10^{-3}$ $m \cdot N^{-1}$	Mechanical compliance of ossicles-drum coupling
R_{cpl}	0.08 $N \cdot s \cdot m^{-1}$	Mechanical resistance of ossicles-drum coupling
R_{free}	0.02 $N \cdot s \cdot m^{-1}$	Mechanical resistance of peripheral parts of the drum
L_{free}	$12 \cdot 10^{-3}$ g	Mechanical inertance of peripheral parts of the drum
R_{st}	$18 \cdot 10^{-3}$ $N \cdot s \cdot m^{-1}$	Mechanical resistance of stapes
L_{st}	$3 \cdot 10^{-3}$ g	Mechanical inertance of stapes
C_{st}	$1.2 \cdot 10^{-3}$ $m \cdot N^{-1}$	Mechanical compliance of stapes
R_{c}	$70 \cdot 10^{-3}$ $N \cdot s \cdot m^{-1} / A_{\mathrm{F}}^2$	Resistance of cochlea
L_{c}	$10 \cdot 10^{-3}$ g $/ A_{\mathrm{F}}^2$	Inertance of cochlea
C_{c}	$11 \cdot 10^{-3}$ $m \cdot N^{-1} A_{\mathrm{F}}^2$	Compliance of cochlea
A_{F}	3 mm^2	Stapes footplate area

Human Ear Without Pinna

This model accounts for the acoustic losses associated with the ear canal and the entire human ear, see Figure 2-16. It does not include the radiation losses associated with the pinna, the visible part of the ear which is external to the head.

$$Z_{\text{ear w/o pinna}} = \frac{T_{11} + \dfrac{T_{12}}{Z_{\text{eardrum}}}}{T_{21} + \dfrac{T_{22}}{Z_{\text{eardrum}}}} \tag{2-35}$$

Here, Z_{eardrum} is the eardrum impedance defined in Equation 2-34 and T_{ij} are the components of the transfer matrix T that describes the ear canal as a two-port. The ear canal is treated as N_{tot} small segments each with length Δ_k and radius r_k so its full two port T is given by

$$T = \begin{bmatrix} T_{11} & T_{12} \\ T_{21} & T_{22} \end{bmatrix} = \prod_{k=1}^{N_{\text{tot}}} \begin{bmatrix} \cosh(\Gamma_k \Delta_k) & \sinh(\Gamma_k \Delta_k) Z_{\text{tw}_k} \\ \dfrac{\sinh(\Gamma_k \Delta_k)}{Z_{\text{tw}_k}} & \cosh(\Gamma_k \Delta_k) \end{bmatrix}. \tag{2-36}$$

where

$$\begin{gathered} Z_{\text{tw}_k} = \frac{\rho c}{\pi r_k^2} \qquad \Gamma_k = 3\alpha_k + i\frac{\omega}{c} \\ \alpha_k = \frac{\omega}{c}\left[\frac{1}{\sqrt{2}\text{Wo}_k}\zeta_1 + \frac{1}{\text{Wo}_k^2}\zeta_2 + \frac{1}{\text{Wo}_k^3\sqrt{2}}\zeta_3\right] \\ \zeta_1 = 1 + \frac{\gamma - 1}{\sqrt{\text{Pr}}} \qquad \zeta_2 = 1 + \frac{\gamma - 1}{\sqrt{\text{Pr}}} - \frac{\gamma - 1 + (\gamma - 1)^2}{2\text{Pr}} \\ \zeta_2 = \frac{7}{8} + \frac{\gamma - 1}{\sqrt{\text{Pr}}} - \frac{\gamma - 1 + (\gamma - 1)^2}{2\text{Pr}} + \frac{4(\gamma - 1)^2 + 4(\gamma - 1)^3 - (\gamma - 1)}{8\text{Pr}^{\frac{3}{2}}} \end{gathered} \tag{2-37}$$

In these expressions, γ is the ratio of specific heats, Γ_k is the propagation constant ("wave number") of the k^{th} ear canal segment which has the segment-specific attenuation α_k,

$$\text{Wo}_k = \sqrt{\frac{\rho \omega r_k^2}{\mu}}$$

is the segment-specific Womersley number, and

$$\text{Pr} = \frac{C_p \mu}{k}$$

is the Prandtl number expressed in terms of the specific heat C_p, dynamic viscosity μ, and thermal conductivity k. Notice that the papers presenting the model (Ref. 28 to 30) do not exactly specify which expression for the attenuation constants α_k is being used, but only refer to Ref. 32. The expression above for α_k is the most general expression taken from this paper. The values for Δ_k and r_k are listed in Table 2-9.

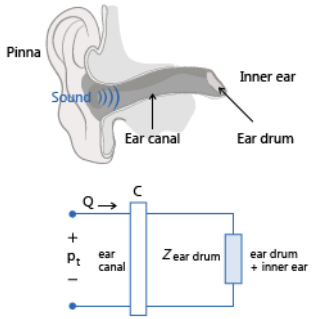


Figure 2-16: Illustration of the impedance of the human ear without pinna.

TABLE 2-9: RADII r_k AND LENGTHS Δ_k OF EAR CANAL SEGMENTS TAKEN FROM Ref. 30.

k	r_k (mm)	Δ_k (mm)
1	1.0	1
2	1.9	1
3	2.4	1
4	2.6	1
5	2.75	1
6	2.85	1
7	2.95	1
8	3.05	1
9	3.2	2
10	3.35	2
11	3.5	2
12	3.6	2
13	3.7	2
14	3.8	2
15	3.9	2
16	4.0	2

TABLE 2-9: RADII r_k AND LENGTHS Δ_k OF EAR CANAL SEGMENTS TAKEN FROM Ref. 30.

k	r_k (mm)	Δ_k (mm)
17	4.1	2
18	4.25	2

The pressure at the eardrum p_{eardrum} is calculated whenever this impedance boundary condition is applied. This pressure is available in postprocessing, and is calculated from the expression

$$p_{\text{eardrum}} = p_{\text{t}} \frac{T_{22} - \dfrac{T_{12}}{Z_{\text{ear w/o pinna}}}}{T_{11}T_{22} - T_{12}T_{21}},$$

where p_{t} is the pressure on the boundary, T_{ij} are the coefficients of the ear canal transfer matrix T and $Z_{\text{ear w/o pinna}}$ is the ear impedance; T and $Z_{\text{ear w/o pinna}}$ are defined in Equation 2-35 and Equation 2-36 above.

Human Ear, Full

This model accounts for all acoustic losses associated with the entire human ear, both the internal parts as well as the pinna, the visible, external part of the ear on the head, see Figure 2-17. The model does not include any information about the directivity of the ear (the head related transfer functions, HRTFs) which depends on the ear geometry at higher frequencies. It is valid in the low frequency limit and for normal incidence on the ear. It is given by

$$Z_{\text{ear full}} = Z_{\text{rad}} + Z_{\text{ear w/o pinna}}$$

where Z_{rad} and $Z_{\text{ear w/o pinna}}$ are given by Equation 2-32 and Equation 2-35 above.

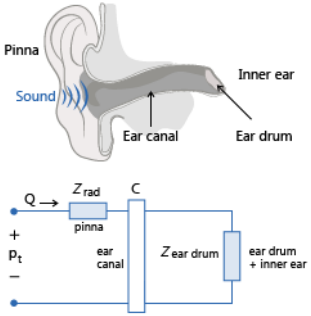


Figure 2-17: Illustration of the impedance of the full human ear including radiation losses due to the pinna.

The pressure at the eardrum p_{eardrum} is calculated whenever this impedance boundary condition model is applied. This pressure is calculated from $Z_{\text{ear full}}$ using the expression

$$p_{\text{eardrum}} = p_{\text{t}}\left(1 - \frac{Z_{\text{rad}}}{Z_{\text{ear full}}}\right)\frac{T_{22} - \dfrac{T_{12}}{Z_{\text{ear full}}}}{T_{11}T_{22} - T_{12}T_{21}}.$$

Waveguide End Impedance Models

Tubes and ducts are acoustic waveguides, and there are acoustic radiation losses when such a waveguide opens into a large domain. Idealized models for these losses have been implemented as boundary impedance models. Thus, instead of explicitly modeling the large domain, an appropriate impedance model Z_{end} can be applied with $Z_{\text{n}} = Z_{\text{end}}$. These models all assume that the domain is infinitely big, that the propagation is in the direction of the waveguide axis, and that the propagating mode is a plane wave. As with all other impedance boundary models, only the boundary-normal velocity component is taken into account.

In the following the term ρc takes different values depending on when the impedance condition is applied. Specifically, for models with damping they are equal to the complex valued quantities $\rho_c c_c$, while when applied on an Anisotropic Acoustic domain the normal direction variables are used $\rho_{\text{n}} c_{\text{n}}$.

Flanged Pipe, Circular

For a pipe of a user-specified radius a, the acoustic losses are given by (see Ref. 6)

$$Z_{\text{end}} = \rho c\left(1 - \frac{2J_1(2ka)}{2ka} + i\frac{2H_1(2ka)}{2ka}\right) \qquad k = \frac{\omega}{c}$$

where $J_1(x)$ is the Bessel function of the first kind of order 1, $H_1(x)$ is the Struve function, and k is the wave number of the wave. This expression is also known as the impedance from a baffled piston.

Flanged Pipe, Rectangular

For a rectangular duct of user-specified inner width w_{i} and inner height h_{i}, the acoustic losses are given by (see Ref. 5)

$$Z_{\text{end}} = \frac{\rho c}{2\pi w_i h_i}\left[(w_i h_i)^2 k^2 + ik(w_i h_i)^{\frac{3}{2}} f\left(\frac{w_i}{h_i}\right)\right] \qquad k = \frac{\omega}{c}$$

$$f(x) = 2x^{\frac{1}{2}}\sinh^{-1}\left(\frac{1}{x}\right) + 2x^{-\frac{1}{2}}\sinh^{-1}(x) + \frac{2}{3}\left[x^{\frac{3}{2}} + x^{-\frac{3}{2}} - (x + x^{-1})^{\frac{3}{2}}\right].$$

This relationship applies provided the following requirements are satisfied

$$w_i \geq h_i \qquad kw_i \ll 1 \qquad kh_i \ll 1 .$$

Unflanged Pipe, Circular (Low ka Limit)

For an unflanged circular pipe of a user-specified radius a in the limit of small radius (low ka), the pipe end impedance is given by the classical expression (see Ref. 6)

$$Z_{\text{end}} = \rho c\left[\frac{(ka)^2}{4} + i0.6133ka\right] \qquad k = \frac{\omega}{c} \qquad ka \ll 1 .$$

Unflanged Pipe, Circular

For an unflanged pipe of any user-specified radius a relative to the wave number k, an approximate end impedance is given in Ref. 33. It is

$$Z_{\text{end}} = \rho c\frac{1+R}{1-R} \qquad R = -|R|e^{2ika\delta(ka)} \qquad k = \frac{\omega}{c}$$

$$|R| = \begin{cases} e^{-\frac{(ka)^2}{2}}\left[1 + \frac{(ka)^4}{6}\left(\log\left(\frac{1}{e^{0.5772}ka}\right) + \frac{19}{12}\right)\right] & \text{if } ka < 1 \\ \sqrt{\pi ka}\,e^{-ka}\left[1 + \frac{3}{32}\frac{1}{(ka)^2}\right] & \text{if } 1 < ka < 3.83 \end{cases}$$

where $\delta(ka)$ is a tabulated phase correction function. The implemented impedance models does not use the exact curve from Ref. 33. The tabulated function $\delta(ka)$ is depicted in Figure 2-18 together with the absolute value of the reflection coefficient $|R|$. A comparison between a full simulation and the impedance model can be seen in the *Open Pipe* application library verification model. Note that, Figure 2-18 also shows the theoretical low frequency limit for the reflection coefficient

$$|R|(\omega \to \infty) = 1 - \frac{1}{2}(ka)^2 .$$

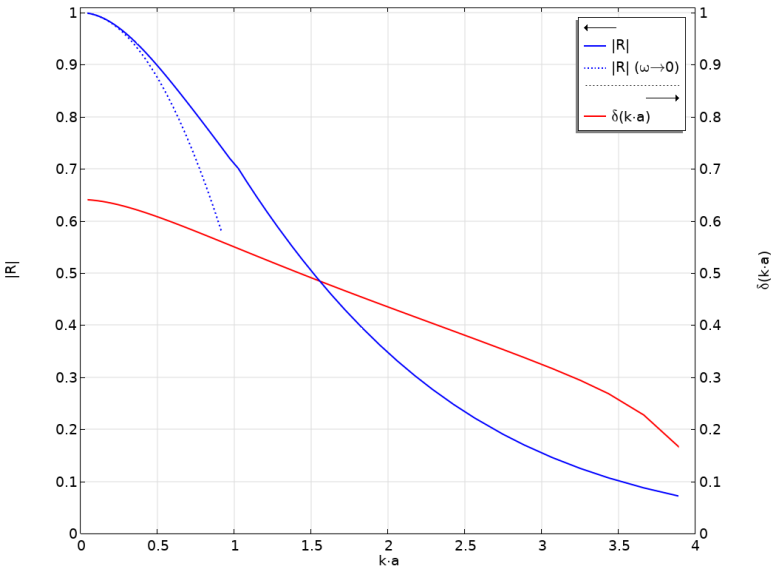


Figure 2-18: Absolute value of reflection coefficient as well as phase correction factor for the unflanged pipe end-impedance model.

Porous Layer Models

A boundary impedance model is implemented to specifically handle cases investigating the acoustical properties of a porous layer of a given thickness d backed by a sound-hard wall. This can be applied instead of explicitly modeling the porous layer, as long as the incident acoustic field is normal to the boundary; as for all the other boundary impedance models, the tangential components of the acoustic field are ignored by this model.

For a porous layer with a user-specified thickness d, the impedance from the porous layer backed by a sound-hard wall is given by (see Ref. 9)

$$Z_n = -i\rho_c c_c \cot(kd) \qquad k = \frac{\omega}{c_c}.$$

Here ρ_c and c_c are the equivalent fluid descriptions of the porous model. This impedance model applies to any type of porous model which can be written as an equivalent fluid model. All porous models implemented in COMSOL are available for

this impedance boundary condition. See details of the poroacoustics equivalent fluid models in the section Theory for the Equivalent Fluid Models.

Characteristic Specific Impedance Models

For well-defined wave types in infinite domains, an impedance condition exists in every point whereby the pressure and normal velocity are related, that is, the specific characteristic acoustic impedance. Thus, these impedances can be imposed on a boundary to model an infinite, open domain in cases where the wave source inside the domain is either a direction (plane wave), a point (spherical wave) or a line (cylindrical wave). However, be aware that only the boundary-normal component of the velocity is used in the impedance boundary condition while the tangential component is ignored; in cases of nonnegligible tangential components it is recommended to instead use the options Plane Wave Radiation, Spherical Wave Radiation, and Cylindrical Wave Radiation.

Plane Wave

The impedance is given by

$$Z_n = \rho_c c_c ,$$

see Ref. 6. This is given solely by material parameters and has no user input.

Spherical Wave

This impedance corresponds to the wave from a point source. It is calculated from the expression (given in Ref. 6)

$$Z_n = \frac{\rho_c c_c}{1 + \frac{1}{ikr_b}} \qquad r_b = |\mathbf{x}_0 - \mathbf{x}| \qquad k = \frac{\omega}{c_c}$$

where $\mathbf{x}_0$ is the user-specified location of the point source generating the spherical waves, $\mathbf{x}$ is the position of the boundary, and r_b is the distance between the two.

Cylindrical Wave

This classic infinite-domain wave impedance from an acoustic line source with the user-specified direction $\mathbf{n}_0$ and position $\mathbf{x}_0$ is presented in, for example, Ref. 6. The expression for the impedance is

$$Z_{\mathrm{n}} = i\rho_{\mathrm{c}}c_{\mathrm{c}}\frac{H_0^{(2)}(kr_b)}{H_1^{(2)}(kr_b)} \qquad k = \frac{\omega}{c_{\mathrm{c}}}$$

$$r_{\mathrm{b}} = \left|(\mathbf{x}_0 - \mathbf{x}) - \{(\mathbf{x}_0 - \mathbf{x}) \cdot \mathbf{e}_{\mathrm{sa}}\}\mathbf{e}_{\mathrm{sa}}\right| = \frac{\left|(\mathbf{x}_0 - \mathbf{x}) \times \mathbf{e}_{\mathrm{sa}}\right|}{\left|\mathbf{e}_{\mathrm{sa}}\right|}$$

where $H_m^{(2)}(x) = J_m(x) - iY_m(x)$ is the Hankel function of the second kind of order m given in terms of the Bessel functions of order m of the first and second kind, $J_{\mathrm{m}}(x)$ and $Y_{\mathrm{m}}(x)$, respectively. Notice that the source axis vector $\mathbf{e}_{\mathrm{sa}}$ is automatically normalized in this implementation.

Theory for the Interior Impedance Models

In this section:

- Interior Perforated Plate Models

Interior Perforated Plate Models

The transfer impedance of an interior boundary is defined as the ratio of the pressure drop across the boundary to the normal velocity on the boundary:

$$Z_n = \frac{\Delta p_t}{v_n} = \frac{p_{t,up} - p_{t,down}}{v_n} \tag{2-38}$$

The resulting value of Z_n can be treated as a superposition of several contributions which are derived separately. Note that the theory below is only valid for the perforates with circular-shaped holes. Other types of holes can lead to significantly different results, which will make the models that are considered here inadequate and unreliable.

TRANSFER IMPEDANCE OF A HOLE

Let the z-coordinate axis be directed along the axis of a cylindrical hole of the height t_p (see Figure 2-19). Let the variation of the pressure and the velocity along the z-axis have the following pattern:

$$p(z) = pe^{-ik_c z}, \mathbf{v}(z) = \mathbf{v}e^{-ik_c z} \tag{2-39}$$

The substitution of Equation 2-39 into Equation 2-38 results in

$$Z_n = \frac{p(z+t_p) - p(z)}{v_n(z+t_p/2)} = \frac{p(z)}{v_n(z)}(e^{-ik_c t_p/2} - e^{ik_c t_p/2}) = -2iZ_c \sin\frac{k_c t_p}{2},$$

where Z_c is the characteristic impedance and k_c is the complex wave number defined according to the low reduced frequency (LRF) models from the Narrow Region Acoustics for Slits, Circular Ducts, Rectangular Ducts, and Equilateral triangular Ducts. That is,

$$Z_c = \rho c([\gamma-(\gamma-1)\Upsilon_{th}]\Upsilon_v)^{-\frac{1}{2}}, \qquad k_c = \frac{\omega}{c}\left([\gamma-(\gamma-1)\Upsilon_{th}]\frac{1}{\Upsilon_v}\right)^{\frac{1}{2}}.$$

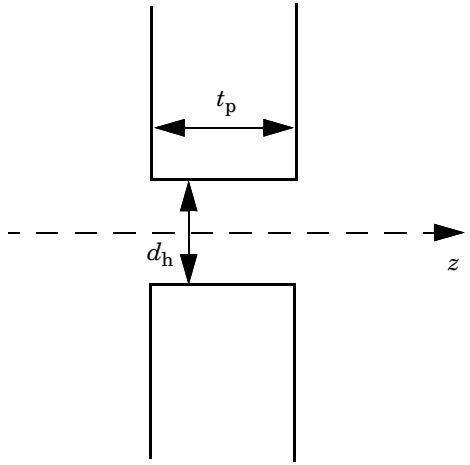


Figure 2-19: To the derivation of the transfer impedance of a one-hole perforate.

The expression for the normalized transfer impedance of an orifice reads

$$z_i = \frac{Z_i}{\rho c} = -\frac{2i\sin\left(\frac{k_c t_p}{2}\right)}{\sqrt{[\gamma-(\gamma-1)\Upsilon_{th}]\Upsilon_v}}. \tag{2-40}$$

For thinner plates ($kt_p << 1, k = \omega/c$) the approximation $\sin\left(\frac{k_c t_p}{2}\right) \approx \frac{k_c t_p}{2}$ is valid, which yields

$$z_n = \frac{Z_n}{\rho c} = -\frac{i\omega}{c}\frac{t_p}{\Upsilon_v}. \tag{2-41}$$

Note that Equation 2-41 coincides with the expression that follows from Crandall's formula for an infinite tube with the circular cross section (see Ref. 36). Equation 2-40 accounts for both viscous and thermal effects inside the hole, while the simplified Equation 2-41 contains the viscous part only (thermal effects are negligible for the thin plate limit).

VENA CONTRACTA AND AREA POROSITY

Equation 2-40 and Equation 2-41 are exact if the streamlines of a flow through the hole are parallel to the z-axis throughout the orifice area. In reality, there is a radial component of the flow, which leads to the reduction (contraction) of this area. The minimum area where the streamlines remain parallel to the z-axis is called *vena contracta*. The flow velocity at the vena contracta is also different from that of the ideal flow. A coefficient that accounts for these effects is called the discharge coefficient, C_D. The value of the discharge coefficient can be obtained from measurements as function of the plate thickness and the orifice shape and diameter.

Another parameter that used to express the interior impedance of a perforate is the porosity, σ. The holes are usually uniformly distributed over the plate, and the porosity accounts for the distribution as the ratio of the hollow area to the area of the plate. Depending on the pattern used when the holes are strewn over the plate, the porosity is defined as follows

$$\sigma = \frac{\pi d_h^2}{4a^2} \qquad \text{and} \qquad \sigma = \frac{\pi d_h^2}{2\sqrt{3}a^2}$$

for a square and a triangular pattern, respectively. The parameter a determines the hole spacing.

The resulting expression for the interior impedance of a perforate comes out from dividing z_i by the product σC_D:

$$z_{\text{orifice}} = \frac{z_n}{\sigma C_D}. \tag{2-42}$$

Either Equation 2-40 or Equation 2-41 can be used for the substitution for z_i in Equation 2-42. The resulting models will further be referred to as the thick and the thin plate models, respectively.

END CORRECTION AND HOLE-HOLE INTERACTION

The subscript "orifice" in Equation 2-42 means that the expression accounts for the transfer impedance of a perforate caused by the presence of the holes; that is, a piston of fluid of the length t_p. However, the actual mass of fluid affected by an incident wave is larger than that inside the hole. The effective mass of the fluid can be taken into account by the piston which is on each side longer by δ than the initial one (see Figure 2-20). This results in adding two extra terms (for each side of the perforate) of

the form Equation 2-41 with t_p replaced by δ. The choice of Equation 2-41 is due to the absence of highly conducting walls in the end corrections area.

If two holes are located relatively close to one another, the actual masses of the attached fluid can become overlapped. This makes the total mass less than just the sum of those for the separate hole. In order to take the hole-hole interaction into account, the end correction is reduced by a factor f_{int}.

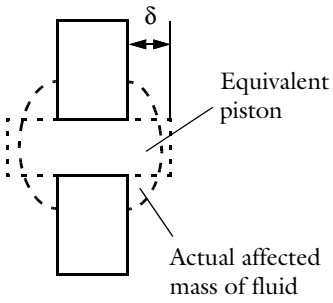


Figure 2-20: To the definition of end corrections.

The last is a function of the porosity and the most often expressed by the Fok function:

$$\psi(\sigma) = \sum_{n=0}^{N} a_n (\sqrt{\sigma})^n .$$

The end correction is usually considered as a function of the hole diameter and can in practice be different for the resistive and the reactive parts of the transfer impedance. For this reason, it is useful to split δ into two parts: δ_{resist} and δ_{react}. The resulting term that accounts for the end correction and the hole-hole interaction reads

$$z_{\text{end}} = -\text{Re}\left(\frac{i\omega}{c\sigma C_D}\frac{2\delta_{\text{resist}}}{\Upsilon_v} f_{\text{int}}\right) - i\,\text{Im}\left(\frac{i\omega}{c\sigma C_D}\frac{2\delta_{\text{react}}}{\Upsilon_v} f_{\text{int}}\right) . \qquad (2\text{-}43)$$

Equation 2-43 is only acceptable if the media is the same on both sides of the perforate. If the plate is backed by a porous layer on one side, the values of δ and f_{int} can differ significantly from those on the other side. For example, the interaction between holes is hampered by porous media, which results in neglecting of f_{int} on one side of the plate and leads to the following correction factor: $(\delta + 1)f_{\text{int}}$.

NONLINEAR AND MEAN FLOW EFFECTS

At medium and high sound pressure levels, the displacement of acoustic particles becomes comparable to the diameter of the holes. This causes flow separation and vortex shedding at the entrance and the exit of the hole. This results in the acoustic energy dissipation and increases the acoustic resistance of the perforate Ref. 37. That is, an extra resistance term should be added to the resulting transfer impedance expression.

Different forms of the contribution to the resistance are similar in the following sense:

- The resistance term is directly proportional to the acoustic particle velocity, which makes it nonlinear;
- The term is frequency dependent through the acoustic particle velocity.

The expression incorporated into the Interior Perforated Plate boundary condition reads

$$z_{\text{nl}} = \frac{1-\sigma^2 f_{\text{nl}}}{\sigma^2 C_{\text{D}}^2}\frac{|v_{\text{n}}|}{2c}, \tag{2-44}$$

where f_{nl} is a correction factor (equals 1 by default) and v_{n} is the acoustic particle velocity component normal to the plate. Other expressions for the nonlinear resistance term can be found in Ref. 37.

The presence of a mean flow also changes of the transfer impedance. In order to account for these and other possible effects, the Interior Perforated Plate boundary condition feature includes an option to enable user-defined resistance and reactance:

$$z_{\text{user}} = \theta^{(\text{user})} + i\chi^{(\text{user})}. \tag{2-45}$$

Relative to the mean flow, the contribution of a grazing or a bias flow to the resistance can be expressed through the flow Mach number as shown in Ref. 38.

Combining the expressions Equation 2-42–Equation 2-45 together yields the full expression for the transfer impedance of a perforate for the thick

$$\frac{Z_n}{\rho c} = -\frac{2i\sin\left(\frac{k_c t_p}{2}\right)}{\sqrt{[\gamma-(\gamma-1)\Upsilon_{th}]\Upsilon_v}} - \mathrm{Re}\left(\frac{i\omega}{c\sigma C_D^{(lin)}}\frac{2\delta_{resist}}{\Upsilon_v}f_{int}\right) - i\,\mathrm{Im}\left(\frac{i\omega}{c\sigma C_D^{(lin)}}\frac{2\delta_{react}}{\Upsilon_v}f_{int}\right)$$

$$+\frac{1-\sigma^2}{(\sigma C_D^{(nl)})^2}\frac{f_{nl}}{2c}|v_n| + \theta^{(user)} + i\chi^{(user)}$$

and the thin

$$\frac{Z_n}{\rho c} = -\mathrm{Re}\left(\frac{i\omega}{c\sigma C_D^{(lin)}}\frac{t_p+2\delta_{resist}}{\Upsilon_v}f_{int}\right) - i\,\mathrm{Im}\left(\frac{i\omega}{c\sigma C_D^{(lin)}}\frac{t_p+2\delta_{react}}{\Upsilon_v}f_{int}\right)$$

$$+\frac{1-\sigma^2}{(\sigma C_D^{(nl)})^2}\frac{f_{nl}}{2c}|v_n| + \theta^{(user)} + i\chi^{(user)}$$

plate models. The discharge coefficient C_D is taken different for the linear and the nonlinear parts for the sake of flexibility.

Theory for the Equivalent Fluid Models

In this section:

- Introduction to the Equivalent Fluid Models
- About the Pressure Acoustics Fluid Models
- About the Poroacoustics Models
- About the Narrow Region Acoustics Models

Introduction to the Equivalent Fluid Models

It is possible to define the properties of a fluid in several ways in pressure acoustics. In a Pressure Acoustics domain feature attenuation properties for the bulk fluid may be specified. Acoustic losses in porous materials are modeled by homogenizing the porous matrix and saturating fluid, and are defined by the Poroacoustics domain feature (frequency domain only). The viscous and thermal losses that occur in the acoustic boundary layer can be modeled in a homogenized way using the Narrow Region Acoustics domain feature (frequency domain only). The different ways of defining the properties of a fluid are called *fluid models*. They are also often referred to as *equivalent fluid models*.

Losses and damping occur when acoustic waves propagate in a porous material (where material refers to the homogenization of a fluid and a porous solid) because of bulk viscous and thermal properties, or because of thermal and viscous losses in the acoustic boundary layer at walls in narrow ducts. The purpose of the fluid model is to mimic a special loss behavior by defining a complex-valued density ρ_c and speed of sound c_c. These are often frequency dependent.

In a Pressure Acoustics domain feature, the default Linear elastic fluid model (see Linear Elastic Fluid Model) enables you to specify a linearly elastic fluid using either the density ρ and speed of sound c, the impedance Z and wave number k, or the equivalent bulk modulus K and the density ρ. When any of these material parameters are complex-valued, damping is introduced.

It is always necessary to specify a set of two parameters (for example Z and k) or conditions in order to calculate the complex speed of sound and complex density

needed to specify a fluid model. The choice of parameters typically depend on the application and which equivalent fluid is being modeled. For example:

- It is possible to determine the complex wave number k and impedance Z from directly measuring it in an impedance tube in order to produce curves of the real and imaginary parts (the resistance and reactance, respectively) as functions of frequency. These data can be used directly as input to COMSOL Multiphysics interpolation functions to define k and Z.
- The option to define the equivalent bulk modulus K and density ρ is often used when characterizing the propagation of acoustic waves in a porous material. These parameters may be determined from measurements or by defining an analytical model expression.

The linear elastic fluid model enables the user to enter any desired fluid models with the desired combination of fluid properties. It is the most general fluid model. You may enter any user-defined analytical expressions for your favorite equivalent fluid model or use measurement data to represent the lossy behavior of the fluid.

PROPAGATION IN GENERAL FLUID WITH BULK LOSSES

The options are (see About the Pressure Acoustics Fluid Models and the settings for the Pressure Acoustics node):

- Linear elastic: define density and speed of sound, impedance and wave number, or equivalent bulk modulus and density.
- Linear elastic with attenuation: define an attenuation parameter for the fluid.
- Ideal gas is also available but not described here. This fluid model is used to specify the fluid properties by selecting a gas constant type and selecting between entering the heat capacity at constant pressure or the ratio of specific heats. See Ideal Gas Fluid Model for details.
- Viscous and/or thermally conducting fluids (bulk losses): the losses are due to viscous losses, thermal conduction, or the combined thermal conduction and viscous losses in the bulk of the fluid.

PROPAGATION IN POROUS MATERIALS

A series of fluid models exist for describing the propagation of pressure waves in porous materials. These range from fully empirical models to semianalytical/empirical models with varying degree of complexity. See Poroacoustics and About the Poroacoustics Models for more detail.

PROPAGATION IN NARROW REGIONS, NARROW TUBES, OR WAVEGUIDES

See Narrow Region Acoustics and About the Narrow Region Acoustics Models. The losses are due to absorption/dissipation in the acoustic boundary layer (thermal and viscous losses). The losses are smeared on the domain in a homogenized way.

About the Pressure Acoustics Fluid Models

USER-DEFINED ATTENUATION FLUID MODEL

Use the user-defined attenuation fluid model to specify an attenuation defined through an attenuation coefficient α. The attenuation can be an analytical expression or given by measurement data imported using an interpolation function. This results in a complex-valued wave number k. This defines the complex speed of sound.

$$c_c = \frac{\omega}{k}$$

There are different attenuation types to select from: attenuation coefficient Np/m, attenuation coefficient dB/m, or attenuation coefficient dB/λ. Select **Attenuation coefficient Np/m** to define an attenuation coefficient in Np/m (nepers per meter), to define:

$$k = \frac{\omega}{c} - i\alpha \qquad \rho_c = \rho\left(\frac{c}{c_c}\right)^2$$

This results in a real valued bulk modulus of the fluid $K = \rho c^2$.

Select **Attenuation coefficient dB/m** to define an attenuation coefficient in dB/m (decibel per meter), to define:

$$k = \frac{\omega}{c} - i\ln(10)\frac{\alpha}{20} \qquad \rho_c = \rho\left(\frac{c}{c_c}\right)^2$$

This results in a real valued bulk modulus of the fluid $K = \rho c^2$.

Select **Attenuation coefficient dB/λ** to define an attenuation coefficient in dB/λ (decibel per wavelength). This option is often used in under water acoustics. The definition of the complex valued wavenumber and (real) density follows:

$$k = \frac{\omega}{c}\left(1 - i\ln(10)\frac{\alpha}{2\pi \cdot 20}\right) \qquad \rho_c = \rho$$

Notice the different definition of the complex density. This definition is commonly used in underwater acoustics, where the density is kept as a real value, see Ref. 63.

Feature description: User-Defined Attenuation Fluid Model

VISCOUS FLUID MODEL

The viscous model is an equivalent-fluid model that mimics the propagation of sound in a fluid including viscous losses occurring in the bulk of the fluid. The elastic fluid model with viscous losses is defined by:

$$\rho_c = \rho\left(1 + \frac{i\omega\delta}{c^2}\right)^{-1} \qquad c_c = c\left(1 + \frac{i\omega\delta}{c^2}\right)^{\frac{1}{2}}$$

$$\delta = \frac{1}{\rho}\left(\frac{4}{3}\mu + \mu_B\right)$$

where μ is the dynamic viscosity and μ_B is the bulk viscosity (see Ref. 4 or Ref. 6). This choice is only appropriate for situations where the damping takes place in free space and is not related to interaction between the fluid and a solid skeleton or a wall. These losses, in most fluids, occur over long distances or at very high frequencies.

THERMALLY CONDUCTING FLUID MODEL

The thermally conducting model is an equivalent-fluid model that mimics the propagation of sound in a fluid including losses due to thermal conduction in the bulk. The elastic fluid model with thermal losses is defined by:

$$\rho_c = \rho\left(1 + \frac{i\omega\delta}{c^2}\right)^{-1} \qquad c_c = c\left(1 + \frac{i\omega\delta}{c^2}\right)^{\frac{1}{2}}$$

$$\delta = \frac{1}{\rho}\left(\frac{(\gamma - 1)k}{C_p}\right)$$

where γ is the ratio of specific heats, C_p is the specific heat at constant pressure, and k is the thermal conductivity (see Ref. 6 chapter 9). This choice is only appropriate for situations where the damping takes place in free space and is not related to interaction between the fluid and a solid skeleton or a wall.

THERMALLY CONDUCTING AND VISCOUS FLUID MODEL

The thermally conducting and viscous model is an equivalent-fluid model that mimics the propagation of sound in a fluid including losses due to thermal conduction and viscosity in the bulk of the fluid. The elastic fluid model with thermal and viscous losses is defined by:

$$\rho_c = \rho\left(1 + \frac{i\omega\delta}{c^2}\right)^{-1} \qquad c_c = c\left(1 + \frac{i\omega\delta}{c^2}\right)^{\frac{1}{2}}$$

$$\delta = \frac{1}{\rho}\left(\left(\frac{4}{3}\mu + \mu_B\right) + \frac{(\gamma - 1)k}{C_p}\right)$$

where μ is the dynamic viscosity and μ_B is the bulk viscosity, γ is the ratio of specific heats, C_p is the specific heat at constant pressure, and k is the thermal conductivity (see Ref. 6 chapter 9). This choice is only appropriate for situations where the damping takes place in free space and is not related to interaction between the fluid and a solid skeleton or a wall.

Feature description: Thermally Conducting and/or Viscous Fluid Model

About the Poroacoustics Models

The Poroacoustics node introduces several fluid models for modeling the propagation of acoustic waves in porous materials. After some general remarks about fluid models and the rigid and limp regime approximations, these models are discussed in this section:

- Porous Fluid Models
- Delany–Bazley–Miki
- Zwikker–Kosten
- Attenborough
- Wilson
- Johnson–Champoux–Allard (JCA)
- Johnson–Champoux–Allard–Lafarge (JCAL)
- Johnson–Champoux–Allard–Pride–Lafarge (JCAPL)
- Three-Parameter Approximation JCAL Model

- Williams EDFM (equivalent density fluid model)
- Wood (fluid suspension Model)

POROUS FLUID MODELS

The poroacoustics fluid models are equivalent fluid models that mimic the behaviors of a full Poroelastic Material model, which is defined by Biot's theory. A poroacoustics fluid model is based on describing the frequency-dependent effective fluid density $\rho(\omega)$ and the effective fluid bulk modulus $K(\omega)$ of the combined equivalent fluid-solid system (saturating fluid and porous matrix). The description of these models includes the losses associated with the propagation of acoustic waves in porous materials. An *equivalent fluid model* is computationally less demanding than the full poroelastic model. However, it is only physically correct for certain choices of material parameters. Most poroacoustic models are only valid in the rigid or limp porous matrix approximations.

Limp and Rigid Porous Matrix Models

In the *rigid porous matrix* limit, the matrix is assumed to be so stiff that it does not move (sometimes referred to as a *motionless skeleton model*). In this case it is assumed that in Biot's theory $\mathbf{u} = \mathbf{0}$, which yields a wave equation with complex density and bulk modulus. In a rigid porous material the Biot-Willis coefficient is equal to the porosity $\alpha_B = \varepsilon_p$. All the poroacoustic fluid models are based on defining the rigid effective density ρ_{rig} (see Ref. 9).

The *limp porous matrix* limit is the opposite of the rigid assumption. It is used to model materials where the stiffness of solid phases (the porous matrix) is so weak that it cannot support free, structure-borne wave propagation (neither longitudinal nor transverse). That is, the stiffness (in vacuo bulk stiffness) is very small compared to air such that the solid phase motion becomes acoustically significant. If it is light enough, the solid phase still moves because it is "dragged along" by the fluid motion; a limp porous material model is also an equivalent fluid model because it only features a single longitudinal wave type. Typically, the limp assumption can be applied to very light weight fibrous materials (less than 10 kg/m^3) if these are not stiffened by the injection of binder material. In the limp case, it is assumed that the stress tensor vanishes and that in Biot's theory, the Biot–Willis coefficient is $\alpha_B = 1$. The limp density is related to the rigid density by a mixture model (see Ref. 10)

$$\rho_{limp} = \frac{\rho_{rig}\rho_{av} - \rho_f^2}{\rho_{av} + \rho_{rig} - 2\rho_f} \qquad \rho_{av} = \rho_d + \varepsilon_p\rho_f$$

where ρ_f is the fluid density, ρ_d is the drained porous matrix density, ρ_{av} is the average effective density, and ρ_{limp} is the resulting effective limp density.

Fluid Parameters

Defining the equivalent density and bulk modulus results in the usual complex-valued speed of sound and density (the propagation parameters):

$$k_{eq}^2 = \left(\frac{\omega}{c_c}\right)^2 - k_z^2 - \left(\frac{m}{r}\right)^3$$

$$c_c = \sqrt{\frac{K}{\rho_{rig}}} \quad \text{or} \quad c_c = \sqrt{\frac{K}{\rho_{limp}}}$$

$$\rho_c = \rho_{rig} \quad \text{or} \quad \rho_c = \rho_{limp}$$

Ideal Gas and General Fluid Options

We show below that all implemented models of poroacoustics apply for all fluids (gases and liquids) except the purely empirical Delany–Bazley–Miki model which has been obtained from fitting to measurement data.

The applicability of the poroacoustics models to general fluids is not described in the literature, but follows directly from repeating the published derivations without assuming the saturating fluid to be an ideal gas. The only difference is that one needs to use the following general expression for the ratio of specific heats γ (express γ in terms of other material parameters) which is valid for any fluid (for gases γ is typically a known parameter)

$$\gamma - 1 = \frac{\alpha_0^2 T_0}{\rho_0 C_p \beta_0} = \frac{c^2 \alpha_0^2 T_0}{C_p}$$

$$\alpha_0 = -\frac{1}{\rho_0}\frac{d\rho}{dT}\bigg|_p \quad , \tag{2-46}$$

$$\beta_0 = \frac{1}{\rho_0}\frac{d\rho}{dp}\bigg|_T = \frac{1}{K_T}$$

where α_0 is the isobaric thermal expansion coefficient, T_0 is the background quiescent temperature, ρ_0 is the background quiescent density, C_p is the specific heat at constant pressure, β_0 is the isothermal compressibility (inverse isothermal bulk modulus K_T), and c is the speed of sound. This result comes from thermodynamics.

To show the general applicability of the poroacoustics models, we here rederive the equivalent bulk modulus for the simplest case of the Zwikker-Kosten model for a general fluid. All other implemented models (again except the Delany-Bazley-Miki

model) are extensions of this result (see Ref. 9), and the equivalent bulk moduli can therefore in general be written as

$$K(\omega) = \frac{K_0}{\gamma - (\gamma - 1)\psi(\omega)} \tag{2-47}$$

for any of these models, where the frequency response $\Psi(\omega)$ is specific to each model. Consequently, the following derivation is sufficient to show the general result.

Derivation of the Equivalent Bulk Modulus Valid for Any Fluid in the Zwikker–Kosten Theory

In general, the (isentropic) bulk modulus is given by

$$K_0 = \rho \frac{dp}{d\rho}. \tag{2-48}$$

A relationship between the pressure p and the density ρ is needed to define the effective bulk modulus, and the equation of state provides such a relationship. Instead of relying on the ideal gas equation, we here use the following general equation of state

$$\rho = \rho(p, T) \tag{2-49}$$

which merely stipulates that the density ρ is a function of both pressure p and temperature T. The temperature-dependence is nonnegligible since we are dealing with small pores, and acoustics in porous materials belong to thermoviscous acoustics (see Thermoviscous Acoustics Interfaces). Assuming the acoustic fields are small harmonic perturbations (denoted by a subscript 1) about a quiescent reference state (denoted by subscript 0)

$$\begin{aligned} \rho &= \rho_0 + \rho_1 e^{i\omega t} \\ p &= p_0 + p_1 e^{i\omega t} \\ T &= T_0 + T_1 e^{i\omega t} \end{aligned} \tag{2-50}$$

and Taylor-expanding the general equation of state Equation 2-49 to first order, we find

$$\rho_1 = \rho_0(\beta_T p_1 - \alpha_0 T_1) \tag{2-51}$$

where β_T is the isothermal compressibility.

In order to calculate the effective bulk modulus from Equation 2-48 using the equation of state Equation 2-51, a relationship between T_1 and p_1 is derived from the

linearized energy equation (see Theory Background for the Thermoviscous Acoustics Branch)

$$0 = \left(\nabla^2 - \frac{i\omega}{D_{th}}\right) T_1 + \frac{\alpha_0 T_0}{k} i\omega p_1 + \frac{Q}{k}$$
$$D_{th} = \frac{k}{\rho_0 C_p} \tag{2-52}$$

where D_{th} is the thermal diffusivity, k is the thermal conductivity, and Q is a volumetric heat source. To proceed with the solution of this equation, it is generally assumed that (*i*) the pore can be modeled as a hollow cylinder, (*ii*) the contributions from radial variations dominate over axial variations so it suffices to solve the radial problem while assuming no axial variations Ref. 9, and (*iii*) the pore radius is much smaller than the wavelength, whereby the pressure does not change noticeably across the radius. By these assumptions the pressure p_1 can be treated as a constant in Equation 2-52 and the equation has only radial variations:

$$\nabla^2 T_1 \rightarrow \frac{1}{r}\frac{\partial}{\partial r} T_1 + \frac{\partial^2}{\partial r^2} T_1$$

The cross-sectional average solution under these assumptions, and with the boundary condition of zero acoustic temperature T_1 on the pore walls at $r = R$, is

$$\bar{T}_1(r) = \left[\frac{\alpha_0 T_0}{\rho_0 C_p} p_1 + \frac{Q}{\rho_0 C_p}\frac{1}{i\omega}\right]\left[1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-i}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-i})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-i})}\right]. \tag{2-53}$$

Here

$$\mathrm{Wo} = \sqrt{\frac{R^2 \omega \rho_0}{\mu}} \qquad \mathrm{Pr} = \frac{\mu}{\rho D_{th}} \tag{2-54}$$

where Wo is the Womersley number (see the note below), and Pr is the Prandtl number, measuring the relative width of the viscous boundary layer thickness to the thermal boundary layer thickness.

With the solution from Equation 2-53, the pressure can now be expressed as a function of the density using the first-order equation of state Equation 2-51:

$$p_1(\rho_1) = \frac{\rho_1 + \frac{Q\alpha_0}{C_p}\left[1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}\right]}{\rho_0\left(\beta_T - \frac{\alpha_0^2 T_0}{\rho_0 C_p}\left[1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}\right]\right)} \tag{2-55}$$

From this we obtain the following expression for the equivalent bulk modulus (see Equation 2-48)

$$K(\omega) = \rho_0\frac{\mathrm{d}p_1}{\mathrm{d}\rho_1} = \frac{K_0}{\gamma - (\gamma - 1)\left[1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}\right]} \tag{2-56}$$

where the ratio of specific heats γ is defined in Equation 2-46 for a general fluid and K_0 is the isentropic bulk modulus of the fluid. This formula Equation 2-56 is mathematically equivalent to the normal Zwikker–Kosten formula for the equivalent bulk modulus (see Ref. 15), but it has been *derived for a general fluid and not just an ideal gas.*

We emphasize that from this result for a general fluid follows the general formula Equation 2-47, in this particular case with

$$\psi(\omega) = 1 - \frac{2}{\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}}}\frac{J_1(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})}{J_0(\mathrm{Wo}\sqrt{\mathrm{Pr}}\sqrt{-\mathrm{i}})} \tag{2-57}$$

with the Womersley number Wo containing the frequency dependence.

The *Womersley number* Wo measures the influence of viscous effects relative to the oscillation frequency ω. For $\mathrm{Wo} \ll 1$ viscosity dominates and the velocity profile is the well-known Poiseuille parabola, while in the Helmholtz regime for $\mathrm{Wo} \gg 1$ the velocity profile is plug like with a very small boundary layer close to the walls.

DELANY–BAZLEY–MIKI

The Delany–Bazley–Miki model is an equivalent fluid model that mimics the bulk losses in certain porous/fibrous materials. The model represents a porous medium with the following complex propagation constants:

$$k_c = \frac{\omega}{c}[1 + C_1 X^{-C_2} - iC_3 X^{-C_4}]$$

$$Z_c = \rho_f c[1 + C_5 X^{-C_6} - iC_7 X^{-C_8}]$$

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

where ρ_f is the fluid density, f is the frequency, and R_f is the flow resistivity. Several predefined sets of the coefficients C_i exist. They are the classic Delany–Bazley model, the Miki model (see Ref. 9, section 2.5 and Ref. 22), the Qunli, several variants of the Mechel model for different configurations, the Komatsu model, and a so-called Modified Champoux and Allard model (see Ref. 23). Note the exception for the Komatsu model, where $X = 2 - \log_{10}(f/R_f)$, which is a fully empirical quantity. These are all empirical models based on fitting the two complex functions to measured data for the complex wave number k_c and complex specific acoustic impedance Z_c. All the models are applicable for materials with a porosity ε_p close to 1. The applicability of the different model parameters is listed in Table 2-10 and also Ref. 23 for further details.

TABLE 2-10: DELANY-BAZLEY-MIKI MODEL OPTIONS AND APPLICABILITY.

MODEL NAME	APPLICABILITY
Delany-Bazley	Glass and rock wool with: $0.01 \le X \le 1$ $10^3 \le R_f \le 50 \cdot 10^3$ Pasm^{-2}
Miki	Glass and rock wool with: $0.01 \le X \le 1$ $10^3 \le R_f \le 50 \cdot 10^3$ Pasm^{-2} The validity of the model using the Miki parameters is not well-established for $X < 0.01$, as it can violate the passivity condition, but the model is slightly better behaved mathematically below this limit using the Miki parameters rather than the Delany–Bazley parameters, see Ref. 22.

TABLE 2-10: DELANY-BAZLEY-MIKI MODEL OPTIONS AND APPLICABILITY.

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

ZWIKKER–KOSTEN

Zwikker–Kosten is one of the earliest equivalent fluid models for porous materials (Ref. 15). It is a rigid frame model defined by the complex (rigid) density

$$\rho_{rig} = \frac{\rho_f}{\varepsilon_p} \frac{1}{1 - \frac{2}{\mathrm{Wo}\sqrt{-i}} \frac{J_1(\mathrm{Wo}\sqrt{-i})}{J_0(\mathrm{Wo}\sqrt{-i})}} \qquad \mathrm{Wo} = \sqrt{\frac{\omega \rho_f H_r^2}{\mu}} = \sqrt{2}\frac{H_r}{\delta_v}$$

where H_r is the hydraulic radius of the pores (for straight cylindrical pores $H_r = a$ = radius) and Wo is the Womersley number (see Equation 2-54 and the note below). Wo is related to the ratio between viscous penetration depth δ_v and the hydraulic radius. δ_v gives the scale of the viscous boundary layer thickness (see Theory Background for the Thermoviscous Acoustics Branch for details). The bulk modulus is given by

$$K = \frac{\gamma p_A}{\varepsilon_p} \frac{1}{1 + (\gamma - 1)\frac{2}{\mathrm{Wo}\sqrt{-i\mathrm{Pr}}} \frac{J_1(\mathrm{Wo}\sqrt{-i\mathrm{Pr}})}{J_0(\mathrm{Wo}\sqrt{-i\mathrm{Pr}})}} \qquad \mathrm{Pr} = \frac{C_p \mu}{k}$$

where p_A denotes the ambient pressure, ρ_f the fluid density, γ the ratio of specific heat, Pr the Prandtl number, μ the dynamic viscosity, C_p the heat capacity at constant

pressure, and k the coefficient of thermal conduction. J_0 and J_1 are Bessel functions of the first kind. The factor γp_A is the isentropic bulk modulus ($K_0 = \gamma p_A$). The free parameters of the pores are the porosity ε_p and the hydraulic radius H_r.

ATTENBOROUGH

The Attenborough model is also based on the cylindrical-like pore assumption. It is a so-called four parameter semi-empirical model. The model is an extension of the Zwikker–Kosten model and adds two more input parameters. It accounts for the tortuosity (high-frequency limit) τ_∞, which is related to the orientation of the pores relative to the propagation direction. The hydraulic diameter of the pores is replaced by an expression that includes the flow resistivity R_f, and a fitting parameter b, (this parameter is related to the anisotropy of the pores). See Ref. 9 and Ref. 16. The equivalent density and bulk modulus are defined as

$$\rho_{rig} = \frac{\rho_f}{\varepsilon_p} \frac{\tau_\infty}{1 - \frac{2}{s'\sqrt{-i}} \frac{J_1(s'\sqrt{-i})}{J_0(s'\sqrt{-i})}}$$

and

$$K = \frac{\gamma p_A}{\varepsilon_p} \frac{1}{1 + (\gamma - 1)\frac{2}{s'\sqrt{Pr}\sqrt{-i}} \frac{J_1(s'\sqrt{Pr}\sqrt{-i})}{J_0(s'\sqrt{Pr}\sqrt{-i})}} \qquad Pr = \frac{C_p \mu}{k}$$

where p_A denotes the ambient pressure, ρ_f the fluid density, γ the ratio of specific heat, Pr the Prandtl number, μ the dynamic viscosity, C_p the heat capacity at constant pressure, and k the coefficient of thermal conduction. J_0 and J_1 are Bessel functions of the first kind. The variable s' (anisotropy factor) is derived from other material parameters and is related to the Womersley number (see Equation 2-54):

$$s' = b\sqrt{\frac{8\omega\rho_f\tau_\infty}{\varepsilon_p R_f}}$$

Here, ω denotes the angular frequency. The four parameters needed (when the fluid is air at room temperature) are the porosity ε_p, the tortuosity τ_∞, flow resistivity R_f, and

the fitting parameter b (dimensionless, close to 1). The fitting parameter b is tabulated for certain well-defined pore cross sections in Table 2-11.

TABLE 2-11: FITTING FACTOR b FOR DIFFERENT CROSS-SECTIONAL GEOMETRIES (SEE Ref. 9).

CROSS-SECTIONAL SHAPE	b
Circle	1
Square	1.07
Equilateral triangle	1.11
Rectangular slit	0.81

In this way, the hydraulic radius of the Attenborough model is formulated in terms of measurable intrinsic properties of the porous material which alleviates the need to know the pore radii.

The viscous characteristic length of the model L_v can also be defined by

$$L_v = b\sqrt{\frac{8\mu\tau_\infty}{\varepsilon_p R_f}} = s'\sqrt{\frac{\mu}{\omega\rho_f}} = s'\frac{\delta_v}{\sqrt{2}}$$

with δ_v the viscous penetration depth. This length is related to the pores' circular cross-section radius R (for a cylinder) and the thickness of the viscous boundary layer. The parameter $b = 1/s$, where s is the viscous characteristic length parameter. See the following models:

- Johnson–Champoux–Allard (JCA)
- Johnson–Champoux–Allard–Lafarge (JCAL)
- Johnson–Champoux–Allard–Pride–Lafarge (JCAPL)).

Note that the tortuosity is related to the angle θ between the cylindrical pores and the direction of propagation of the wave, by

$$\tau_\infty = \frac{1}{\cos^2(\theta)}$$

WILSON

The Wilson model is a generalization of the analytical models for porous materials with constant cross section and parallel pores. This model is intended to match the middle frequency behavior of a porous material (see Ref. 9, Ref. 17, and Ref. 18). It is not a good model for ω tending to 0 or infinity. The equivalent density and bulk modulus are given by

$$\rho_{rig} = \rho_\infty \frac{(1 + i\omega\tau_{vor})^{1/2}}{(1 + i\omega\tau_{vor})^{1/2} - 1}$$

$$K = K_\infty \frac{(1 + i\omega\tau_{ent})^{1/2}}{(1 + i\omega\tau_{ent})^{1/2} + \gamma - 1}$$

where τ_{vor} denotes the vorticity-mode relaxation time, τ_{ent} the entropy-mode relaxation time, ρ_∞ the infinity-frequency limit for the density, K_∞ the infinity-frequency limit for the bulk modulus, and γ is the ratio of specific heats. These are the four free parameters. With appropriate choices for the relaxation parameters, the Wilson model can be fitted to mimic all the models described here. For example, setting $\tau_{vor} = 2.54/R_f$ and $\tau_{ent} = 3.75/R_f$, the equations mimic the Delany–Bazley model (see Cox and D'Antonio Sec. 5.4.4, Ref. 18).

Approximate expressions based on nonacoustic parameters (properties of the porous matrix) also exist for the relaxation times:

$$\tau_{vor} = \frac{\rho_f l^2}{2\mu} \approx \frac{2\rho_f \tau_\infty}{\varepsilon_p R_f} \qquad \tau_{ent} \approx Pr\,\tau_{vor}$$

$$\rho_\infty \approx \frac{\rho_f \tau_\infty}{\varepsilon_p} \qquad K_\infty \approx \frac{\gamma p_A}{\varepsilon_p}$$

Here τ_∞ denotes the (high-frequency limit) tortuosity (it is called q^2 in the Wilson's paper Ref. 17), ε_p the porosity, ρ_f the fluid density, l a characteristic pore dimension, and Pr is the Prandtl number.

JOHNSON–CHAMPOUX–ALLARD (JCA)

The Johnson–Champoux–Allard (JCA) porous matrix model is defined by the following equivalent rigid densities $\rho_{rig}(\omega)$ and equivalent bulk modulus $K(\omega)$:

$$\rho_{rig} = \frac{\tau_\infty \rho_f}{\varepsilon_p}\left[1 + \frac{R_f \varepsilon_p}{i\omega\rho_f\tau_\infty}\sqrt{1 + \frac{4i\omega\tau_\infty^2\mu\rho_f}{R_f^2 L_v^2 \varepsilon_p^{\,2}}}\right]$$

$$K = \frac{\gamma p_A}{\varepsilon_p}\left[\gamma - (\gamma - 1)\left(1 + \frac{8\mu}{i\omega L_{th}^2 Pr\rho_f}\sqrt{1 + \frac{i\omega L_{th}^2 Pr\rho_f}{16\mu}}\right)^{-1}\right]^{-1}$$

Here τ_∞ is the tortuosity factor (high-frequency limit), ρ_f is the fluid density, ε_p is the porosity, R_f is the flow resistivity, μ is the dynamic viscosity, p_A is the quiescent

pressure, γ is the ratio of specific heats, L_v is the viscous characteristic length, L_{th} is the thermal characteristic length, and Pr is the Prandtl number. The viscous characteristic length is related to the viscous characteristic length parameter s by

$$L_v = \frac{1}{s}\sqrt{\frac{8\mu\tau_\infty}{\varepsilon_p R_f}}$$

Here s is a pore geometry dependent factor between 0.3 and 3.0 (for example 1 for circular pores, 0.78 for slits)

The expression given for the geometry dependent pore factor s is only valid for values of s close to 1. If this is not the case, enter the viscous characteristic length L_v directly into the model (the default selection).

The viscous L_v and thermal L_{th} characteristic lengths are also sometimes denoted by Λ and Λ', respectively.

Porous Absorber: Application Library path **Acoustics_Module/Building_and_Room_Acoustics/porous_absorber**

JOHNSON–CHAMPOUX–ALLARD–LAFARGE (JCAL)

The Johnson–Champoux–Allard–Lafarge (JCAL) model introduces corrections to the bulk modulus thermal behavior at low frequencies that is not captured by the JCA model (see Ref. 13). The equivalent density is the same as in the JCA model. The correction is to the bulk modulus and is given by

$$K = \frac{\gamma p_A}{\varepsilon_p}\left[\gamma-(\gamma-1)\left(1+\frac{\varepsilon_p\mu}{i\omega\kappa'_0\mathrm{Pr}\rho_f}\sqrt{1+\frac{4i\omega(\kappa'_0)^2\mathrm{Pr}\rho_f}{\mu L_{th}^2\varepsilon_p^2}}\right)^{-1}\right]^{-1}$$

with the introduction of the new parameter, κ'_0, which is the static thermal permeability (SI unit: m^2). For measurements and details of this parameter see Ref. 11 and Ref. 12 for examples.

JOHNSON–CHAMPOUX–ALLARD–PRIDE–LAFARGE (JCAPL)

The Johnson–Champoux–Allard–Pride–Lafarge (JCAPL) model further extends the JCAL models by introducing a static viscous τ_0 and thermal τ'_0 tortuosity, which both

introduce low frequency corrections to the JCAL and JCA models. See Ref. 14 and Ref. 9.

This model has the complex rigid density given by

$$\rho_{rig} = \frac{\rho_f \tilde{\tau}(\omega)}{\varepsilon_p}$$

$$\tilde{\tau}(\omega) = \tau_\infty \left[1 + \frac{1}{i\bar{\omega}}\tilde{F}(\omega)\right] \qquad \tilde{F}(\omega) = 1 - P + P\sqrt{1 + \frac{M}{2P^2} i\bar{\omega}}$$

$$i\bar{\omega} = \frac{i\omega\rho_f \kappa_0 \tau_\infty}{\mu\varepsilon_p} \qquad M = \frac{8\kappa_0\tau_\infty}{\varepsilon_p L_v^2}$$

$$P = \frac{M}{4(\tau_0/\tau_\infty - 1)} = \frac{2\kappa_0\tau_\infty^2}{\varepsilon_p L_v^2(\tau_0 - \tau_\infty)}$$

where the new parameter is the static viscous tortuosity τ_0 (dimensionless). The viscous permeability is defined as $\kappa_0 = \mu/R_f$ (SI unit: m^2).

In Ref. 9 (equation 5.32), P is called b, κ_0 is called q_0, and Pr is called B^2.

The complex bulk modulus K is given by

$$K = \frac{\gamma p_A}{\varepsilon_p} \frac{1}{\tilde{\beta}(\omega)}$$

$$\tilde{\beta}(\omega) = \gamma - (\gamma - 1)\left[1 + \frac{1}{i\bar{\omega}'}\tilde{F}'(\omega)\right]^{-1} = \gamma - (\gamma - 1)\tilde{\tau}'(\omega)^{-1}$$

$$\tilde{F}'(\omega) = 1 - P' + P'\sqrt{1 + \frac{M'}{2P'^2} i\bar{\omega}'} \qquad i\bar{\omega}' = \frac{i\omega\rho_f \mathrm{Pr}\kappa'_0}{\mu\varepsilon_p}$$

$$M' = \frac{8\kappa'_0}{\varepsilon_p L_{th}^2} \qquad P' = \frac{M'}{4(\tau'_0 - 1)}$$

where the new parameter is the static thermal tortuosity τ'_0 (dimensionless).

In Ref. 9 (equation 5.35), $\tau'(\omega)$ is called $\alpha'(\omega)$, and $P' = 1$.

The JCA model is recovered by setting $M' = P = P' = 1$ and the JCAL is recovered by setting $P = P' = 1$.

THREE-PARAMETER APPROXIMATION JCAL MODEL

The Three-Parameter Approximation JCAL Model was developed by Horoshenkov *et al.* (see Ref. 58, 59, 60, 61, and 62) to simplify modeling of porous materials. Many of the porous material models, like the JCAL model, require the measurement of material data that typically are measured acoustically. This makes them hard to use outside the acoustic community.

The three-parameter approximation is valid for granular and fibrous materials as well as foams. The material should have a pore size (s) distribution that is close to log-normal. This means that

$$-\log_2(s) \sim N(-\log_2(\bar{s}), \sigma^2)$$

Such a material can be acoustically described by three parameters: the porosity ε_p, the median pore size $\bar{s}$ and the standard deviation of the pore size σ_s. Five of the parameters of the JCAL model (viscous characteristic length, thermal characteristic length, flow resistivity, static thermal permeability, and tortuosity) can then be approximated by

$$L_v = \bar{s}e^{-(5/2)b} \qquad L_{th} = \bar{s}e^{(3/2)b} \qquad b = (\sigma_s \ln 2)^2$$

$$R_f = \frac{8\mu\tau_\infty}{\bar{s}^2\varepsilon_p}e^{6b} \qquad \kappa'_0 = \frac{\bar{s}^2\varepsilon_p}{8\tau_\infty}e^{6b} \qquad \tau_\infty = e^{4b}$$

Two different versions of the model exists. One where these expressions are entered into the JCAL model, and one where the three parameters are used in Padé approximations of the density and the compressibility.

WILLIAMS EDFM (EQUIVALENT DENSITY FLUID MODEL)

The Williams EDFM model is a so-called equivalent density fluid model. The model is derived under the assumption that the bulk and shear moduli of the frame of the porous material are negligible, see Ref. 24 for further details. This is why the model pertains to sediments. The effective bulk modulus K_{eff} and an effective density ρ_{eff} defined my the model are given by

$$K_{\text{eff}} = \left(\frac{1-\varepsilon_p}{K_{\text{gr}}} + \frac{\varepsilon_p}{K_f}\right)^{-1}$$

$$\rho_{\text{eff}}(\omega) = \rho_f \left(\frac{\tau_\infty(1-\varepsilon_p)\rho_{\text{gr}} + \varepsilon_p(\tau_\infty - 1)\rho_f + \dfrac{i\varepsilon_p \rho_{\text{mix}} F(\text{Wo})\mu}{\rho_f \omega \kappa}}{\varepsilon_p(1-\varepsilon_p)\rho_{\text{gr}} + (\tau_\infty - 2\varepsilon_p + \varepsilon_p^2)\rho_f + \dfrac{i\varepsilon_p F(\text{Wo})\mu}{\omega\kappa}} \right)$$

$$\rho_{\text{mix}} = \varepsilon_p \rho_f + (1-\varepsilon_p)\rho_{\text{gr}}$$

where the subscript "gr" pertains to the grains and the subscript "f" to the saturating fluid. The porosity is denoted ε_p, the observed mixture density ρ_{mix}, the tortuosity τ_∞, the dynamic viscosity μ, the angular frequency ω, and the permeability of the sediments κ. The function F is a function of the Womersley number Wo defined as

$$F(\text{Wo}) = \frac{\frac{\text{Wo}}{4}T(\text{Wo})}{1 - \frac{2i}{\text{Wo}}T(\text{Wo})} \qquad T(\text{Wo}) = -\frac{\sqrt{i}J_1(\sqrt{i}\,\text{Wo})}{J_0(\sqrt{i}\,\text{Wo})}$$

$$\text{Wo} = \sqrt{\frac{\rho_f \omega H_r^2}{\mu}}$$

where $J_n(x)$ is the Bessel function of the first kind of order n and H_r is the hydraulic radius. Based on these the effective complex speed of sound and density are given as

$$c_c = \sqrt{\frac{K_{\text{eff}}}{\rho_{\text{eff}}(\omega)}} \qquad \rho_c = \rho_{\text{eff}}(\omega)$$

WOOD (FLUID SUSPENSION MODEL)

In a fluid mixture or a fluid suspension (solid inclusions completely surrounded by fluid), the Wood formula can be used to determine the effective speed of sound for the mixture. It is determined by calculating the effective bulk modulus of the suspension and the volume average density. As the Williams EDFM, this model gives effective values for the mixture. This result is exact for low frequencies (when the wavelength is much larger than the size of the inclusions) since the effective bulk modulus in the quasistatic limit. The Wood model defines

$$\frac{1}{K_{\text{eff}}} = \frac{\theta_f}{K_f} + \sum_i^N \frac{\theta_i}{K_i}$$

$$\rho_{\text{eff}} = \theta_f \rho_f + \sum_i^N \theta_i \rho_i$$

where θ_f, K_f, and ρ_f, are the fluid's volume fraction, adiabatic bulk modulus, and density, respectively; and θ_i, K_i, and ρ_i, are the inclusion's volume fractions, adiabatic bulk moduli, and densities, respectively. And again the complex speed of sound and density are defined as

$$c_c = \sqrt{\frac{K_{\text{eff}}}{\rho_{\text{eff}}}} \qquad \rho_c = \rho_{\text{eff}}$$

About the Narrow Region Acoustics Models

The Narrow Region Acoustics fluid models are used to mimic the thermal and viscous losses that exist in narrow tubes where the tube cross-section length-scale is comparable to the thermal and viscous boundary layer thickness (boundary-layer absorption). It is essential to include these losses in order to get correct results.

These models are commonly used in situations where solving a full detailed thermoviscous acoustic model is computationally costly; for example when analyzing long narrow ducts/tubes of constant cross section. Here it is possible to add or smear the losses associated with the boundary layer onto the bulk of the fluid: an equivalent fluid model. For many geometries, analytical expressions exist for the losses associated with the acoustic boundary layers. The models can be applied under different assumptions. The models and assumptions are discussed in this section.

In more complex geometries where thermal and viscous losses are important, see The Thermoviscous Acoustics, Frequency Domain Interface, which is more fundamental and detailed.

Narrow Region Acoustics

Several fluid models exist:

- The *wide duct approximation* can be used for any duct cross section in the limit where the duct width is significantly larger than the acoustic boundary layer thickness. See Wide Ducts.
- The *very narrow circular ducts (isothermal)* can only be used when the duct width is so small that isothermal conditions apply. This is when the duct width is much smaller than the acoustic thermal boundary layer thickness. See Very Narrow Circular Ducts (Isothermal).
- The *slit*, *circular duct*, *rectangular duct*, and *equilateral triangular duct* models are based on an analytical solution of the thermoviscous acoustic equations in the limit where the acoustic wavelength is much larger than both the duct cross section (below the cutoff frequency) and the boundary layer thickness. This is the case in most engineering applications. See Slits, Circular Ducts, Rectangular Ducts, and Equilateral triangular Ducts.
- Finally, selecting the *user defined* option enables you to enter expressions for the complex wave number and the complex acoustic impedance. These may be analytical expressions, interpolated values, or values extracted from a detailed boundary mode analysis using the full formulation of The Thermoviscous Acoustics, Boundary Mode Interface.

WIDE DUCTS

For a relatively wide duct, the losses introduced in the acoustic boundary layer may be studied by adding these as an effective wall shear force. This approach is used in Blackstock (Ref. 6) and results in equivalent fluid complex wave number k_c defined by

$$k_c = \frac{\omega}{c}\frac{1}{\sqrt{1-B\sqrt{\frac{\pi}{i\omega}}}} \cong \frac{\omega}{c}\left(1+\frac{B}{2}\sqrt{\frac{\pi}{i\omega}}\right)$$
$$B = \frac{4}{H_d}\sqrt{\frac{\mu}{\pi\rho}}\left(1+\frac{\gamma-1}{\sqrt{\mathrm{Pr}}}\right) \qquad \mathrm{Pr} = \frac{\mu C_p}{k} \qquad H_d = 4\frac{S}{C} \tag{2-58}$$

where H_d is the hydraulic diameter of the duct, S is the duct cross-section area, C is the duct circumference, μ is the dynamic viscosity, ρ is the fluid density, γ is the ratio of specific heats, C_p is the specific heat at constant pressure, k is the fluid thermal conductivity, and Pr is the Prandtl number. For a cylindrical duct, $H_d = 2a$ where a is the radius.

The approximation in Equation 2-58 is only valid for systems where the effective radius $H_d/2$ is larger than the boundary layer, but not so small that mainstream thermal and viscous losses are important. Thus requiring

$$\delta_v < \frac{H_d}{2} < \frac{c^2}{\omega^2}\frac{1}{\delta_v} \qquad \delta_v = \sqrt{\frac{2\mu}{\omega\rho}}$$

where δ_v is the characteristic thickness of the viscous boundary layer (the viscous penetration depth), c is the speed of sound, and ω is the angular frequency. The complex wave number is related to the complex density and speed of sound by the equation (assuming a real valued bulk modulus)

$$c_c = \frac{\omega}{k_c} \qquad \rho_c = \rho\left(\frac{c}{c_c}\right)^2$$

VERY NARROW CIRCULAR DUCTS (ISOTHERMAL)

In the other limit where the duct diameter is sufficiently small or the frequency sufficiently low, the thermal boundary layer thickness becomes much larger than the duct cross section a. This is the case when

$$a \ll \delta_{th} \Rightarrow \frac{\omega\rho a^2 C_p}{2k} \ll 1$$

where δ_{th} is the characteristic thickness of the thermal boundary layer (thermal penetration depth), ρ is the density, C_p is the heat capacity at constant pressure, and k is the fluid thermal conductivity. In this case see Pierce (Ref. 5); the system may be seen as isothermal and the acoustic temperature variation is zero everywhere in the duct $T = 0$. The fluid complex wave number k_c is then defined by

$$k_c = \frac{\omega}{c_T}\left(\frac{4\mu}{\rho\omega a^2}\right)^{\frac{1}{2}} - i\left(\frac{4\mu\omega}{\rho c_T^2 a^2}\right)^{\frac{1}{2}} \qquad c_T = \frac{c}{\sqrt{\gamma}} \tag{2-59}$$

where c_T is the isothermal speed of sound, a is the duct radius, μ is the dynamic viscosity, and ω is the angular frequency. The theory is derived for ducts of circular cross section — the model is therefore only applicable for systems with small variations away from a circular cross section. The complex wave number is related to the complex density and speed of sound by the equation (here the bulk modulus is defined in the isothermal limit)

$$c_c = \frac{\omega}{k_c} \qquad \rho_c = \rho\left(\frac{c_T}{c_c}\right)^2$$

SLITS, CIRCULAR DUCTS, RECTANGULAR DUCTS, AND EQUILATERAL TRIANGULAR DUCTS

The slit, circular duct, rectangular duct, and equilateral triangular duct models are based on the so-called *low reduced frequency* (LRF) model that describes the propagation of acoustic waves in small waveguides (ducts and slits) including thermal and viscous losses. Details about these models are in Ref. 19, Ref. 20, and Ref. 21. The models cover the range from fully isothermal conditions (very low frequencies or very narrow tubes) to large ducts where the boundary layer only represents a fraction of the duct size. The models apply as long as the cross section of the duct is much smaller than the acoustic wavelength (the model is below the cutoff frequency) and when the cross section is constant or only very slowly varying.

In a narrow waveguide the complex wave number, k_c, and complex specific acoustic impedance, Z_c, are given by

$$k_c^2 = k_0^2\left(\frac{\gamma-(\gamma-1)\Upsilon_{th}}{\Upsilon_v}\right) \qquad k_0 = \frac{\omega}{c}$$

$$Z_c^2 = \frac{Z_0^2}{\Upsilon_v(\gamma-(\gamma-1)\Upsilon_{th})} \qquad Z_0 = \rho c$$

where Υ_v and Υ_{th} are the mean value (cross section averaged) of the scalar viscous and thermal field functions, respectively. The functions are geometry and material-dependent functions (specified below) and γ is the ratio of specific heats. The fluid density ρ, the speed of sound is c, and the angular frequency ω define the free space wave number k_0 and the specific acoustic impedance Z_0. The subscripts "v" and "th" stand for viscous and thermal (heat) fields, respectively. Once these are known, the complex speed of sound and complex density are given by

$$c_c = \frac{\omega}{k_c}$$

$$\rho_c = \frac{k_c Z_c}{\omega} = \frac{k_0 Z_0}{\omega}\frac{1}{\Upsilon_v} = \frac{\rho}{\Upsilon_v}$$

The values of the Υ_j functions can be derived by solving the full set of linearized Navier–Stokes equations (the equations solved by the thermoviscous acoustics interfaces, see Theory Background for the Thermoviscous Acoustics Branch) by

splitting these into an isentropic (adiabatic), a viscous, and a thermal part. Doing this introduces the viscous and the thermal wave numbers for the system

$$k_v^2 = -i\omega\frac{\rho}{\mu} \qquad k_{th}^2 = -i\omega\frac{\rho C_p}{k}$$

The equations may be solved analytically under the following assumptions used for the LRF models:

- The viscous and thermal wave numbers must be much larger than the acoustic wave number (the acoustic wavelength must be much larger than the boundary layer thickness):

$$\left|\frac{k_j}{k_0}\right| \gg 1 \quad \Rightarrow \quad \lambda_0 \gg \delta_j$$

- The cross section of the waveguide must be much smaller than the acoustic wavelength (the model is below the cutoff frequency).
- The cross section of the waveguide must be constant or at most slowly varying in the propagation direction.
- The length of the waveguide in the propagation direction should be larger than the boundary layer thickness.

The resulting analytical expressions, for the viscous and thermal Y functions, are for the given geometry (these results are reviewed in Ref. 19):

- Slit of height h

$$\Upsilon_j = 1 - \frac{\tan(k_j h/2)}{k_j h/2}$$

- Circular duct of radius a

$$\Upsilon_j = -\frac{J_2(k_j a)}{J_0(k_j a)}$$

- Rectangular duct of side lengths W and H

$$\Upsilon_i = k_j^2 \sum_{m=0}^{\infty} \left[(\alpha_m m')^{-2} \left(1 - \frac{\tan(\alpha_m W/2)}{\alpha_m W/2} \right) + (\beta_m m')^{-2} \left(1 - \frac{\tan(\beta_m H/2)}{\beta_m H/2} \right) \right]$$

$$m' = (m + 1/2)\pi$$

$$\alpha_m = \sqrt{k_j^2 - \left(\frac{2m'}{W}\right)^2} \qquad \beta_m = \sqrt{k_j^2 - \left(\frac{2m'}{H}\right)^2}$$

- Equilateral triangular duct of side length d

$$\Upsilon_i = 1 - 3\frac{\tan\left(\frac{3}{2}(k_j l)\right) - \frac{3}{2}(k_j l)}{\left(\frac{3}{2}(k_j l)\right)^2 \tan\left(\frac{3}{2}(k_j l)\right)} \qquad l = \frac{d}{2\sqrt{3}}$$

User Defined

The user defined option in the Narrow Region Acoustics domain feature can be used to define a LRF model for a waveguide of an arbitrary cross sections. Enter values for the complex wave number k_c and the characteristic complex impedance Z_c. This can be a user defined analytical expression or values derived from a mode analysis study.

To determine the complex propagation constants for a waveguide, of arbitrary cross section, use The Thermoviscous Acoustics, Boundary Mode Interface. Apply it on the cross-section geometry of the waveguide. The interface solves for the propagating modes and includes all losses in detail. The complex wave number k_c is then given by the plane wave mode solved for. This is the variable `tabm.kn`. The predefined variable `tabm.Zc` gives the (lumped) complex characteristic impedance Z_c. Search for the mode nearest to the (lossless) plane wave mode.

Theory for the Perfectly Matched Layers in the Time Domain

In this section:

- Introduction to Perfectly Matched Layers
- Perfectly Matched Layers in the Time Domain

Introduction to Perfectly Matched Layers

The concept of a perfectly matched layer (PML) as an absorbing boundary condition was introduced by Bérenger (Ref. 39) with regard to the system of Maxwell's equations. The PML provided absorption of propagating waves without introducing reflections from the interface between the PML and the physical domain. This made the PML technique attractive for treating open domain problems for acoustic, elastic, and electromagnetic wave propagation.

Bérenger's PML formulation is usually referred to as the split PML, because the unknowns are split into a sum of unphysical components in PML domains. Another formulation that does not require splitting the variables is based on the coordinate transformation in PML domains, where the real coordinate is mapped onto the complex plane:

$$\tilde{x} = f(x) \in C\,. \tag{2-60}$$

For example, the mapping Equation 2-60 will transform the 1D Helmholtz equation as follows:

$$\frac{(i\omega)^2}{c^2}p - \frac{1}{f'(x)}\frac{\partial}{\partial x}\left(\frac{1}{f'(x)}\frac{\partial p}{\partial x}\right) = 0\,. \tag{2-61}$$

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

For the sake of brevity, consider the 1D Equation 2-61. In order to derive the PML formulation in the time domain, the following steps are taken (Ref. 40). First, consider a special form of the mapping Equation 2-60:

$$\tilde{x} = x + \frac{1}{i\omega}\int_{x_0}^{x} \sigma(\xi)d\xi\,, \tag{2-62}$$

which yields $d\tilde{x} = f'(x)dx = (1 + \sigma(x)/i\omega)dx$.

Then, taking Equation 2-62 into account, multiply Equation 2-61 by $1 + \sigma(x)/i\omega$. Equation 2-61 transforms to the following form:

$$\left(\frac{(i\omega)^2}{c^2} + \frac{i\omega\sigma}{c^2}\right)p - \frac{\partial}{\partial x}\left(\frac{1}{1 + \frac{\sigma}{i\omega}}\frac{\partial p}{\partial x}\right) = 0\,. \tag{2-63}$$

The transformation to the time domain is performed according to the rule $i\omega p \rightarrow \partial p/\partial t$. Its direct application to Equation 2-63 would result in a time integral of p. To avoid this, an auxiliary variable u is introduced:

$$\frac{1}{1 + \frac{\sigma}{i\omega}}\frac{\partial p}{\partial x} = \frac{\partial p}{\partial x} - \frac{\sigma}{\sigma + i\omega}\frac{\partial p}{\partial x} = \frac{\partial p}{\partial x} + u\,. \tag{2-64}$$

Equation 2-63 and Equation 2-64 yield a system of partial differential equations in the time domain equivalent to the frequency domain Equation 2-61:

$$\begin{gathered}\frac{1}{c^2}\left(\frac{\partial^2 p}{\partial t^2} + \sigma\frac{\partial p}{\partial t}\right) - \frac{\partial}{\partial x}\left(u + \frac{\partial p}{\partial x}\right) = 0 \\ \frac{\partial u}{\partial t} + \sigma\left(u + \frac{\partial p}{\partial x}\right) = 0,\end{gathered} \tag{2-65}$$

The derivation of the model in 3D space, where more auxiliary variables are required, is given in Ref. 40.

In the axisymmetric cases, the coordinate stretching in PML domains is expressed in curvilinear coordinates. A thorough study of the use of PMLs for such problems in the frequency and the time domains is given in Ref. 41.

References for the Pressure Acoustics Branch

1. D. Givoli and B. Neta, "High-order Non-reflecting Boundary Scheme for Time-dependent Waves," *J. Comput. Phys.*, vol. 186, pp. 24–46, 2004.

2. A. Bayliss, M. Gunzburger, and E. Turkel, "Boundary Conditions for the Numerical Solution of Elliptic Equations in Exterior Regions," *SIAM J. Appl. Math.*, vol. 42, no. 2, pp. 430–451, 1982.

3. A.B. Bauer, "Impedance Theory and Measurements on Porous Acoustic Liners," *J. Aircr.*, vol. 14, pp. 720–728, 1977.

4. S. Temkin, *Elements of Acoustics*, Acoustical Society of America, 2001.

5. A.D. Pierce, *Acoustics: An Introduction to its Physical Principles and Applications*, Acoustical Society of America (second print), 1991.

6. D.T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.

7. P.M. Morse and K.U. Ignard, *Theoretical Acoustics*, Princeton University Press, 1986.

8. L.D. Landau and E.M. Lifshitz, *Fluid Mechanics, Course of Theoretical Physics, Volume 6*, Butterworth-Heinemann, 2003.

9. J.F. Allard and N. Atalla, *Propagation of Sound in Porous Media, Modeling Sound Absorbing Materials*, 2nd Edition, John Wiley & Sons, 2009.

10. R. Panneton, "Comment on the Limp Frame Equivalent Fluid Model for Porous Media," *J. Acoust. Soc. Am.*, vol. 122, no. 6, pp. EL217–EL222, 2007.

11. M. Sadouki, M. Fellah, Z.E. Fellah, E. Ogam, N. Sebaa, F.G. Mitri, and C. Depollier, "Measuring static thermal permeability and inertial factor of rigid porous materials," *J. Acoust. Soc. Am.*, vol. 130, p. 2627, 2011.

12. X. Olny and R. Panneton, "Acoustical determination of the parameters governing thermal dissipation in porous media," *J. Acoust. Soc. Am.*, vol. 123, pp. 814–824, 2008.

13. D. Lafarge, P. Lemarinier, J.-F. Allard, and V. Tarnow, "Dynamic compressibility of air in porous structures at audible frequencies," *J. Acoust. Soc. Am.*, vol. 102, no. 4, pp. 1994–2006, 1997.

14. S.R. Pride, F.D. Morgan, and A.F. Gangi, "Drag forces of porous-medium acoustics," *Phys. Rev. B*, vol. 47, pp. 4964–4978, 1993.

15. C. Zwikker and C. W. Kosten, *Sound Absorbing Materials*, Elsevier Publishing, New York, 1949.

16. K. Attenborough, "On the acoustic slow wave in air filled granular media," *J. Acoust. Soc. Am.*, vol. 81, no. 1, pp. 93–102, 1987.

17. K. Wilson, "Relaxation-matched modeling of propagation through porous media, including fractal pore structure," *J. Acoust. Soc. Am.*, vol. 94, no. 2, pp. 1136–1145, 1993.

18. T.J. Cox and P. D'Antonio, *Acoustic Absorbers and Diffusers, Taylor and Francis*, 2nd ed., 2009. https://apmr.matelys.com/index.html.

19. R. Kampinga, *Viscothermal Acoustics Using Finite Elements, Analysis Tools for Engineers*, PhD thesis, University of Twente, The Netherlands, 2010.

20. M.R. Stinson, "The propagation of plane sound waves in narrow and wide circular tubes, and generalization to uniform tubes of arbitrary cross-sectional shapes," *J. Acoust. Soc. Am.*, vol. 89, p. 550, 1990.

21. H. Tijdeman, "On the propagation of sound waves in cylindrical tubes," *J. Sound Vib.*, vol. 39, p. 1, 1975.

22. Y. Miki, "Acoustical properties of porous materials - modifications of Delany-Bazley models," *J. Acoust. Soc. Jpn (E)*, vol. 11, no. 1, 1990.

23. D. Oliva and V. Hongisto, "Sound absorption of porous materials – Accuracy of prediction methods," *Appl. Acoust.*, vol. 74, issue 12, pp. 1473–1479, 2013.

24. K.L Williams, "An effective density fluid model for acoustic propagation in sediments derived from Biot theory," *J. Acoust. Soc. Am.*, vol. 110, p. 2276, 2001.

25. A.B. Wood, *A Textbook of Sound*, The Macmillan company, New York, 1941.

26. W.M. Leach, Jr, *Electroacoustics & Audio Amplifier Design*, Kendall Hunt publishing company, 2010.

27. B. Håkansson, P. Carlsson, and A. Tjellström, "The mechanical point impedance of the human head, with and without skin penetration," *J. Acoust. Soc. Am.*, vol. 80, issue 4, pp. 1065–1075, 1986.

28. H. Hudde and A. Engel, "Measuring and modeling basic properties of the human middle ear and ear canal. Part I: Model structure and measure techniques," *ACOUSTICA acta acoustica*, vol. 84, pp. 720–738, 1998.

29. H. Hudde and A. Engel, "Measuring and modeling basic properties of the human middle ear and ear canal. Part II: Ear canal, middle ear cavities, eardrum, and ossicles," *ACOUSTICA acta acoustica*, vol. 84, pp. 894–913, 1998.

30. H. Hudde and A. Engel, "Measuring and modeling basic properties of the human middle ear and ear canal. Part III: Eardrum impedances, transfer functions and model calculations", *ACOUSTICA acta acoustica*, vol. 84, pp. 1091–1109, 1998.

31. M. R. Stinson and B. W. Lawton, "Specification of the geometry of the human ear canal for the prediction of sound pressure level distribution," *J. Acoust. Soc. Am.*, vol. 85, pp. 2492–2503, 1989.

32. D.H. Keefe, "Acoustical wave propagation in cylindrical ducts: Transmission line parameter approximations for isothermal and nonisothermal boundary conditions," *J. Acoust. Soc. Am.*, vol. 75, issue 1, pp. 58–62, 1984.

33. H. Levine and J. Schwinger, "On the radiation of sound from an unflanged circular pipe", *Phys. Rev.*, vol. 73, no. 4, pp. 383–406, 1948.

34. R. Courant, K.O. Friedrichs, and H. Lewy, "On the Partial Difference Equations of Mathematical Physics," IBM Journal, vol. 11, pp. 215–234, 1956.

35. Y. Saad, "A Flexible Inner-outer Preconditioned GMRES Algorithm," *SIAM J. Sci. Statist. Comput.*, vol. 14, pp. 461–469, 1993.

36. M.R. Stinson and E.A.G. Shaw, "Acoustic impedance of small, circular orifices in thin plates," *J. Acoust. Soc. Am.*, vol. 77, issue 6, pp. 2039–2042, 1985.

37. M.A. Temiz, J. Tournadre, and I.L. Arteaga, "Non-linear acoustic transfer impedance of micro-perforated plates with circular orifices," *J. Sound and Vibration*, vol. 366, pp. 418–428, 2016.

38. T. Elnady, "Modeling and characterization of perforates in lined ducts and mufflers," *PhD Thesis*, The Royal Institute of Technology (KTH), 2004.

39. J.-P. Bérenger, "A perfectly matched layer for the absorption of electromagnetic waves," *J. Comp. Phys.*, vol. 114, issue 2, pp. 185–200, 1994.

40. B. Kaltenbacher, M. Kaltenbacher, and I. Sim, "A modified and stable version of a perfectly matched layer technique for the 3-d second order wave equation in time domain with an application to aeroacoustics," *J. Comp. Phys.*, vol. 235, pp. 407–422, 2013.

41. F. Collino and P. Monk, "The perfectly matched layer in curvilinear coordinates," *SIAM J. Sci. Comput.*, vol. 19, no. 6, pp. 2061–2090, 1998.

42. M. Kaltenbacher, *Numerical Simulation of Mechatronic Sensors and Actuators*, 3rd edn., Springer: Berlin, 2015.

43. M.F. Hamilton and D.T. Blackstock, eds., *Nonlinear Acoustics*, Academic Press, San Diego, CA, 1998.

44. H.E. Bass, L.C. Sutherland, A.J. Zuckerwar, D.T. Blackstock, and D.M. Hester, "Atmospheric absorption of sound: Further developments," *J. Acoust. Soc. Am.*, vol. 97, pp. 680–683, 1995; "Erratum," *J. Acoust. Soc. Am.*, vol. 99, p. 1259. 1996.

45. ANSI S1.26-2014 (supersedes ANSI S1.26-1995) "American National Standard method for calculation of the absorption of sound by the atmosphere" (Acoustical Society of America, New York, 2014).

46. National Physics Lab (NPL), "NPL Acoustics: Calculation of absorption of sound by the atmosphere," http://resource.npl.co.uk/acoustics/techguides/absorption/

47. R.E. Francois and G.R. Garrison, "Sound absorption based on ocean measurements: Part I: Pure water and magnesium sulfate contributions," *J. Acoust. Soc. Am.*, vol. 72, pp. 896–907, 1982.

48. R.E. Francois and G.R. Garrison, "Sound absorption based on ocean measurements: Part II: Boric acid contribution and equation for total absorption," *J. Acoust. Soc. Am.*, vol. 72, pp. 1879–1890, 1982.

49. M.A. Ainslie and J.G. McColm, "A simplified formula for viscous and chemical absorption in sea water," *J. Acoust. Soc. Am.*, vol. 103, pp. 1671–1672, 1998.

50. F.H. Fisher and V.P. Simmons, "Sound absorption in seawater," *J. Acoust. Soc. Am.*, vol. 62, pp. 558–564, 1977.

51. National Physics Lab (NPL), "Calculation of absorption of sound in seawater," http://resource.npl.co.uk/acoustics/techguides/seaabsorption/

52. M.F. Hamilton and D.T. Blackstock, "On the coefficient of nonlinearity in nonlinear acoustics," *J. Acoust. Soc. Am.*, vol. 83, pp. 74–77, 1988.

53. V. Nikolic, *On Certain Mathematical Aspects of Nonlinear Acoustics: Well-Posedness, Interface Coupling, and Shape Optimization*, PhD thesis, University of Klagenfurt, 2015.

54. M. Berggren, A. Bernland, and D. Noreland, "Acoustic Boundary Layers as Boundary Condition," *J. Comp. Phys.*, vol. 371, pp. 633–650, 2018.

55. J.S. Bach and H. Bruus, "Theory for Acoustics with Viscous Boundary Layers and Streaming in Curved Elastic Cavities," *J. Acoust. Soc. Am.,* vol. 144, pp. 766–784, 2018.

56. J.H. Joergensen and H. Bruus, "Theory of pressure acoustics with thermoviscous boundary layers and streaming in elastic cavities," *J. Acoust. Soc. Am.*, vol. 149, pp. 3599–3610, 2021.

57. IOC, SCOR and IAPSO, 2010: *The international thermodynamics equation of seawater - 2010: Calculation and use of thermodynamic properties.* Intergovernmental Oceanographic Commission, Manuals and Guides No. 56, UNESCO (English), 196 pp.

58. Horoshenkov, A. Hurrell, and J.P. Groby, "A three-parameter analytical model for the acoustical properties of porous media," *J. Acoust. Soc. Am.*, vol. 145, no. 4, pp. 2512–2517, 2019

59. K.V. Horoshenkov, J.P. Groby, and O. Dazel, "Asymptotic limits of some models for sound propagation in porous media and the assignment of the pore characteristic lengths," *J. Acoust. Soc. Am.*, vol. 139, no. 5, pp. 2463–2474, 2016.

60. K.V. Horoshenkov, K. Attenborough, and S.N. Chandler-Wilde, "Padé approximants for the acoustical properties of rigid frame porous media with pore size distributions," *J. Acoust. Soc. Am.,* vol. 104, pp. 1198–1209, 1998.

61. K.V. Horoshenkov, A. Hurrell, and J.P. Groby, "Erratum: A three-parameter analytical model for the acoustical properties of porous media [J. Acoust. Soc. Am. 145(4), 2512–2517 (2019)]," *J. Acoust. Soc. Am.*, vol. 147, no. 1, p. 146, 2020.

62. K.V. Horoshenkov, J.P. Groby, and O. Dazel, "Erratum: Asymptotic limits of some models for sound propagation in porous media and the assignment of the pore characteristic lengths [J. Acoust. Soc. Am. 139(5), 2463–2474 (2016)]," *J. Acoust. Soc. Am.*, vol. 147, no. 1, p. 205, 2020.

63. G.B. Deane, "Internal friction and boundary conditions in lossy fluid seabeds," *J. Acoust. Soc. Am.*, vol. 101, p. 233, 1997.

64. M.J. Lighthill, “On sound generated aerodynamically I. General theory,” *Proc. Roy. Soc. Lond.*, vol. 211, pp. 564–587, 1952.

65. M.J. Lighthill, “On sound generated aerodynamically II. Turbulence as source of sound,” *Proc. Roy. Soc. Lond.*, vol. 222, pp. 1–22, 1954.

66. M. Kaltenbacher, *Numerical Simulation of Mechatronic Sensors and Actuators, Finite Elements for Computational Multiphysics*, Springer, Third Edition, 2015.

67. M. Kaltenbacher (Editor), *Computational Acoustics*, CISM, Courses and Lectures Volume 579, Springer, 2018.

68. S. Schroder and M. Kaltenbacher, “Hybrid Aeroacoustic Computations: State of Art and New Achievements,” *J. Theor. Comput. Acoust.*, vol. 27, p. 1950020, 2019.

69. S. Glegg and W. Devenport, *Aeroacoustics of Low Mach Number Flows*, Elsevier, Academic Press, 2017.

70. D.W. Herrin, F. Martinus, T.W. Wu, and A.F. Seybert, “An assessment of the high frequency boundary element and Rayleigh integral approximations,” *Applied Acoustics*, vol. 67, pp. 819-833, 2006.

71. J.A. Fawcett, “Modeling of high frequency scattering from objects using a hybrid Kirchhoff/diffraction approach,” *J. Acoust. Soc. Am.,* vol. 109, pp. 1312-1319, 2001.

72. K.G. Foote and D.T.I. Francis, “Comparing Kirchhoff-approximation and boundary-element models for computing gadoid target strengths,” *J. Acoust. Soc. Am.,* vol. 111, pp. 1644–1654, 2002.

73. J.H. Joergensen, *Theory and modeling of thermoviscous acoustofluidics*, PhD Thesis, Technical University of Denmark, 2022.

74. S. Erlandsson, *Evaluation, adaption and implementations of Perfectly Matched Layers in COMSOL Multiphysics*, MSc Thesis, KTH, Stockholm, 2020.

75. F. Nicoud, L. Benoit, C. Sensiau, and T. Poinsot, “Acoustic modes in combustors with complex impedances and multidimensional active flames,” *AIAA J.*, vol. 45, no. 2,pp. 426–441, 2007.

3

Elastic Waves Interfaces

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

- The Solid Mechanics (Elastic Waves) Interface
- The Poroelastic Waves Interface
- The Elastic Waves, Time Explicit Interface
- The Piezoelectric Waves, Time Explicit Interface
- Modeling with the Elastic Waves Branch
- Theory for the Poroelastic Waves Interfaces
- Theory for the Elastic Waves, Time Explicit Interface

The Solid Mechanics (Elastic Waves) Interface

The **Solid Mechanics (Elastic Waves)** interface ( ), found under the **Acoustics** > **Elastic Waves** branch ( ) when adding a physics interface, is a shortcut to add the Solid Mechanics interface which is used to compute the displacement field in solids with propagating elastic waves. The Solid Mechanics interface supports studies in many forms, specifically the dynamic Navier's equation is solved in the solid in the frequency domain. Dedicated Multiphysics Couplings exist to couple fluid, solid, and porous domains.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linear Elastic Material**, **Free**, and **Initial Values**. For 2D axisymmetric components, an **Axial Symmetry** node is also added. Features and boundary conditions are described in the Solid Mechanics documentation.

For information about meshing see Meshing and Solving Wave Problems Solved with Solid Mechanics in the Modeling with the Elastic Waves Branch section.

Two examples on scattered-field formulation for elastic waves:

- *Scattered-Field Formulation for Elastic Waves*. Application Library path **Acoustics_Module/Elastic_Waves/scattered_field_elastic_waves**
- *Elastic Cloaking with Polar Material*. Application Library path **Acoustics_Module/Elastic_Waves/elastic_cloaking**

THE PORT CONDITION

The **Solid Mechanics** interface has functionality and features that are specifically tailored for elastic wave applications. The Port condition allows setting up conditions at the inlet/outlet of elastic waveguide structures that are mathematically consistent. Use the **Port** to set up nonreflecting conditions, which perform better than a PML, as every propagating modes are captured explicitly.

When the port condition is used, a Boundary Mode Analysis can be performed for the given cross section. The equations for solving the propagating modes are automatically

added to the boundary. This analysis solves an eigenvalue problem for the out-of-plane wavenumber for a given frequency. The out-of-plane extension is added analytically to the governing equations. The analysis is also sometimes referred to as semianalytical finite elements or SAFE.

See the tutorial *Mechanical Multiport System: Elastic Wave Propagation in a Small Aluminum Plate*. Application Library path **Acoustics_Module/Elastic_Waves/mechanical_multiport_system**

SOLVING ELASTIC WAVE PROBLEMS IN THE TIME DOMAIN

When solving elastic wave problems with the **Solid Mechanics** interface in the time domain, it is important to make changes to the **Transient Solver Setting** section. Expand the section and select the checkbox **Maximum frequency to resolve**, then enter the maximum frequency to resolve in the model $f_{\mathrm{max,sol}}$. The maximum frequency is dictated by the frequency content of the source as well as the eigenmodes of the structure that can be excited. Setting this correctly will results in a transient solver tuned specifically for modeling wave problems.

The Poroelastic Waves Interface

The **Poroelastic Waves (pelw)** interface ( ), found under the **Acoustics** > **Elastic Waves** branch ( ) when adding a physics interface, is used to compute the displacement field and acoustic pressure fluctuation in porous materials with propagating poroelastic waves using Biot's mixed p-u formulation. Dedicated Multiphysics Couplings exist that define the couplings between fluid, solid, and porous domains. Anisotropic poroelastic properties can be modeled using the Anisotropic Poroelastic Material material model.

Examples of applications include the propagation of elastic waves in rocks and soils, modeling the acoustic attenuation properties of particulate filters, characterizing sound absorbers and liners, and modeling the porous foams in headphones.

The physics interface is valid for modeling the propagation of the coupled linear elastic and linear acoustic waves in the frequency domain. Harmonic variation of the pressure, displacement field, and sources is assumed and given by $e^{i\omega t}$, that is, using the $+i\omega$ convention. Biot's equations are solved accounting for the coupled propagation of elastic waves in the elastic porous matrix and pressure waves in the saturating pore fluid. This includes the damping effect of the pore fluid due to the combined effect of viscous and thermal losses (the Biot–Allard option where the poroacoustic properties are described by the JCA model), typically when the saturating fluid is air. An option also exists for liquids where only the viscous losses are important (the classical Biot loss model), this is typically with a saturating liquid like water or oil. The interface includes options for modeling both isotropic and anisotropic porous materials.

- For information about modeling strategies and meshing see Meshing Poroelastic Waves Models and Solving Large Poroelastic Wave Models in the Modeling with the Elastic Waves Branch section.
- See the Theory for the Poroelastic Waves Interfaces for details about the governing equations. The specifics of the Biot and the Biot–Allard models are also discussed here.

The boundary conditions are organizes such that conditions that apply to both the pressure field in the saturating fluid and the displacement field in the porous matrix are located in the main menu, while conditions that apply only to the porous matrix are in the **Porous Matrix Conditions** submenu, and conditions that apply to the saturating fluid are in the **Fluid Conditions** submenu. The latter two types of conditions can be combined on the same boundary.

When the **Poroelastic Waves** interface is added, these default nodes are also added to the **Model Builder**: **Poroelastic Material**, **Impervious Layer, Free**, and **Initial Values**. For 2D axisymmetric components, an **Axial Symmetry** node is also added.

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

- *Acoustics of a Particulate-Filter-Like System*: Application Library path **Acoustics_Module/Automotive/acoustics_particulate_filter**
- *Acoustic Reflections off a Water–Sediment Interface*: Application Library path **Acoustics_Module/Underwater_Acoustics/reflections_water_sediment**
- *Headphone on an Artificial Ear*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/headphone_artificial_ear**
- *Transverse Isotropic Porous Layer*: Application Library path **Acoustics_Module/Building_and_Room_Acoustics/transverse_isotropic_porous_layer**
- *Anisotropic Porous Absorber*: Application Library path **Acoustics_Module/Building_and_Room_Acoustics/anisotropic_porous_absorber**

SETTINGS

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

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to

different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

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

The rest of the physics interface settings sections are the same as for The Pressure Acoustics, Frequency Domain Interface and The Solid Mechanics Interface.

Domain, Boundary, and Pair Nodes for the Poroelastic Waves Interfaces

The Poroelastic Waves Interface has these domain, boundary, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

- Anisotropic Poroelastic Material
- Added Mass
- Boundary Load
- Body Load
- Edge Load
- Fixed Constraint
- Free
- Free, Sound Soft
- Initial Stress and Strain[1]
- Initial Values
- Interior Impervious Layer
- Impervious Layer
- Impervious Mass Layer
- Periodic Condition
- Poroelastic Material
- Prescribed Acceleration
- Prescribed Displacement
- Prescribed Velocity
- Pressure
- Point Load
- Point Load (on Axis)
- Rigid Connector[1]
- Ring Load
- Roller
- Spring Foundation[1]
- Symmetry
- Symmetry Plane
- Thin Elastic Layer

[1] These features are described for the Solid Mechanics interface in the *Structural Mechanics Module User's Guide*

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

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Poroelastic Material

Use the **Poroelastic Material** node to define the poroelastic material and fluid properties of an *isotropic* porous material, that is the properties of the porous matrix and the saturating fluid.

POROELASTIC MODEL

Select the **Model** used to describe the losses to include in the saturating fluid of the porous material:

- **Biot (viscous losses)**, primarily used in earth sciences when the saturating fluid is a liquid like water or oil. The model is based on Biot's original work and only includes the effects of viscous losses in the pores.
- **Biot–Allard (thermal and viscous losses)** (the default), intended for simulating porous materials where the saturating fluid is air. This is for modeling sound absorbers, liners, foams used in headphones and loudspeakers, cloth, and much more. The model still relies on the mixed u-p formulation of Biot, but includes the poroacoustic losses using the JCA (Johnson-Champoux-Allard) model.

When selecting a specific model, the required material input will change in order to align with the data normally available for the intended applications.

Three different waves can propagate in a poroelastic material; fast pressure waves, slow pressure waves, and shear waves. Their speed can be evaluated through the variables `pelw.cp_fast`, `pelw.cp_slow`, and `pelw.cs_poro`, respectively. These should not be confused with the material properties for the constituting solid (`pelw.cp` and `pwel.cs`) and fluid (`pelw.c`). All three poroelastic wave types should be resolved by the computational mesh.

POROUS MATRIX PROPERTIES

The default **Porous elastic material** uses the **Domain material** (the material defined for the domain). This setting should be different from the one for the **Fluid material** selected in the **Fluid Properties** section. Typically, select another material (a material defined in the **Materials** node) for the porous elastic material, and use the default **Domain material** option for the **Fluid material**.

First, enter the mechanical properties of the isotropic drained porous matrix. The drained parameters are also sometimes referred to as the in-vacuo elastic parameters;

they are in principle measured without the presence of the saturating fluid. Then (below the dividing line) enter the porous (or poroacoustic) parameters of the material:

- Porous Parameters for Biot (Viscous Losses)
- Porous Parameters for Biot–Allard (Thermal and Viscous Losses)

For the isotropic drained porous matrix, from the **Specify** list, select:

- **Young's modulus and Poisson's ratio** to specify drained Young's modulus (elastic modulus) E_d (SI unit: Pa) and Poisson's ratio ν_d (dimensionless). For an isotropic material, Young's modulus is the spring stiffness in Hooke's law, which in 1D form is $\sigma = E_d\varepsilon$, where σ is the stress and ε is the strain. Poisson's ratio defines the normal strain in the perpendicular direction, generated from a normal strain in the other direction and follows the equation $\varepsilon_\perp = -\nu\varepsilon_\parallel$.
- **Shear modulus and Poisson's ratio** (the default for the Biot–Allard model) to specify drained shear modulus G_d (SI unit: Pa) and Poisson's ratio ν_d (dimensionless).
- **Young's modulus and Shear modulus** to specify drained Young's modulus (elastic modulus) E_d (SI unit: Pa) and drained shear modulus G_d (SI unit: Pa).
- **Bulk modulus and shear modulus** (the default for the Biot model) to specify the drained bulk modulus K_d (SI unit: Pa) and the drained shear modulus G_d (SI unit: Pa). The bulk drained modulus is a measure of the solid porous matrix's resistance to volume changes. The shear modulus is a measure of the solid porous matrix's resistance to shear deformations.
- **Lamé parameters** to specify the drained Lamé parameters λ_d (SI unit: Pa) and μ_d (SI unit: Pa).
- **Pressure-wave and shear-wave speeds** to specify the drained pressure-wave speed c_p (SI unit: m/s) and the shear-wave speed c_s (SI unit: m/s).

Then also enter:

- **Drained density of porous material** to specify the drained density of the porous material in vacuum ρ_d (SI unit: kg/m^3). The drained density ρ_d is equal to $(1 - \varepsilon_p)\rho_s$, where ρ_s is the density of the solid material from which the matrix is made and ε_p is the porosity.
- **Isotropic structural loss factor** (only for the Biot-Allard model) to specify the loss factor of the porous matrix η_s (dimensionless). This value introduces the damping due to losses in the porous structure by transform the elastic moduli into complex valued quantities. This quantity can be frequency dependent if necessary.

For each pair of properties, select from the applicable list to use the value **From material** or enter a **User defined** value or expression. Each of these pairs define the drained elastic properties and it is possible to convert from one set of properties to another.

Porous Parameters for Biot (Viscous Losses)

Enter the following (remaining) parameters necessary to define the properties of a **Biot (viscous losses)** porous material model. The defaults use values **From material**. For **User defined**, enter other values or expressions as needed.

- **Porosity** to specify the porosity of the material ε_p (dimensionless). It defines the amount of void volume inside the porous matrix and takes values between 0 (fully solid material no fluid) and 1 (no porous material only fluid).
- **Permeability** to specify the permeability of the porous material κ_p (SI unit: m^2). The permeability is a measure of the ability of the porous material to let fluid pass through it. It hence gives some measure of the pore size and thus correlates to the viscous damping experienced by pressure waves propagating in the saturating fluid.
- **Biot–Willis coefficient** to specify the Biot–Willis coefficient α_B (dimensionless). This coefficient relates the bulk modulus (compressibility) of the drained porous matrix to a block of solid material. It is defined as

$$\alpha_B = 1 - \frac{K_d}{K_s}$$

  where K_d is the drained bulk modulus and K_s is the bulk module of a block of solid material (made of the matrix material). The drained bulk modulus is related to the stiffness of the porous matrix, while the solid bulk modulus is related to the compressibility of the material or grains from which the porous matrix is made. The Biot–Willis coefficient is bound by $\varepsilon_p \le \alpha_B \le 1$. A rigid porous matrix (Voigt upper bound) has $\alpha_B = \varepsilon_p$ and a soft or limp porous matrix (Reuss lower bound) has $\alpha_B = 1$.
- **Tortuosity factor** (high frequency limit) or the structural form factor τ_∞ (dimensionless). This is a purely geometrical factor that depends on the microscopic geometry and distribution of the pores inside the porous material. It is independent of the fluid and solid properties and is normally >1. The default is 2. The more complex the propagation path through the material, the higher is the absorption. The tortuosity partly represents this complexity.

Porous Parameters for Biot–Allard (Thermal and Viscous Losses)

Enter the following (remaining) parameters necessary to define the properties of a **Biot–Allard (thermal and viscous losses)** porous material model. The defaults use values **From material**. For **User defined** enter other values or expressions as needed.

- **Porosity** to specify the porosity of the material ε_p (dimensionless). It defines the amount of void volume inside the porous matrix and takes values between 0 (fully solid material no fluid) and 1 (no porous material only fluid).
- **Flow resistivity** to specify the (static) flow resistivity of the porous material R_f (SI unit: $Pa \cdot s/m^2$). The flow resistivity is a measure of the ability of the porous material to let fluid pass through it. It hence gives some measure of the pore size and thus correlates to the viscous damping experienced by pressure waves propagating in the saturating fluid. The flow resistivity is also sometimes denoted σ (using the unit $N \cdot s/m^4$) and it is related to the permeability through $\kappa_p = \mu/R_f$.
- **Tortuosity factor** (high frequency limit) or the structural form factor τ_∞ (dimensionless). This is a purely geometrical factor that depends on the microscopic geometry and distribution of the pores inside the porous material. It is independent of the fluid and solid properties and is normally >1. The default is 2. The more complex the propagation path through the material, the higher the absorption. The tortuosity partly represents this complexity.
- **Viscous characteristic length** to specify the viscous length scale L_v (SI unit: m). This value is sometimes denoted Λ and replaces the hydraulic radius used in simpler models to account for the viscous losses that appear in the acoustic boundary layer at pore walls.
- **Thermal characteristic length** to specify the thermal length scale L_{th} (SI unit: m). This value is sometimes denoted Λ' and replaces the hydraulic radius used in simpler models to account for the thermal losses that appear in the acoustic boundary layer at pore walls.
- **Biot–Willis coefficient** how to calculate the Biot–Willis coefficient α_B by selecting **From material**, **Rigid assumption** (the default), **General model**, or **User defined**. This coefficient relates the bulk modulus (compressibility) of the drained porous matrix to a block of solid material.
  - **From material** to pick up the value from the domain material.
  - **Rigid assumption** (the default) and the model defines a rigid porous matrix (Voigt upper bound) where $\alpha_B = \varepsilon_p$.
  - **General model** to define the Biot–Willis coefficient α_B according to its general definition

$$\alpha_B = 1 - \frac{K_d}{K_s}$$

where K_d is the drained bulk modulus and K_s is the bulk module of the skeleton material (bulk modulus of a block of solid material made of the matrix material). When this option is selected, also enter the **Bulk modulus of skeleton material** K_s (taken from material as default). The drained bulk modulus K_d is related to the stiffness of the porous matrix, while the skeleton bulk modulus K_s is related to the compressibility of the material or grains from which the porous matrix is made. The Biot–Willis coefficient is bound by $\varepsilon_p \le \alpha_B \le 1$. A rigid porous matrix (Voigt upper bound) has $\alpha_B = \varepsilon_p$ and a soft or limp porous matrix (Reuss lower bound) has $\alpha_B = 1$.

- **User defined** enter a value for the Biot–Willis coefficient α_B (dimensionless). In the limp limit when $\alpha_B = 1$ it is recommended to use the Poroacoustics feature of The Pressure Acoustics, Frequency Domain Interface instead of the Poroelastic Waves interface. In poroacoustics, the limp limit, when the structure is so "fluffy" that it moves with the fluid, has the assumption $K_d = 0$ and $\alpha_B = 1$ included explicitly.

FLUID PROPERTIES

The default **Fluid material** uses the **Domain material**. This is the material defined for the domain in the **Materials** node.

Define the properties of the saturating fluid in terms of its density, viscosity, and compressibility but also the viscosity model. The defaults use values for the material parameters are **From material**. For **User defined** enter other values or expressions as needed.

- Fluid Parameters for Biot (Viscous Losses) Model
- Fluid Parameters for Biot–Allard (Thermal and Viscous Losses) Model

Fluid Parameters for Biot (Viscous Losses) Model

- **Density** defines the density of the saturating fluid ρ_f (SI unit: kg/m^3).
- **Dynamic viscosity** to define the dynamic viscosity of the saturating fluid μ_f (SI unit: Pa·s). The parameter is important for the amount of viscous damping experienced by the acoustic waves.
- **Compressibility** of the saturating fluid χ_f (SI unit: 1/Pa). Remember that the fluid compressibility χ_f is related to the fluid bulk modulus K_f (SI unit: Pa) and the speed of sound c, through the relation

$$\chi_f = \frac{1}{K_f} = \frac{1}{c^2 \rho_f}$$

The compressibility of the fluid also enters the expression for Biot's module M, given by

$$M = \frac{K_s}{1 - \varepsilon_p - \frac{K_d}{K_s} + \varepsilon_p K_s \chi_f} \qquad K_s = \frac{K_d}{1 - \alpha_B}$$

It should be noted that Biot–Willis coefficient α_B only depends on the properties of the porous matrix while Biot's module M depends on both fluid and porous matrix properties.

Select a **Viscosity Model**, either **Biot's low frequency range** or **Biot's high frequency range**.

- **Biot's low frequency range** models damping at low frequencies where the acoustic boundary layer (the viscous penetration depth) is assumed to span the full width of the pores. This is also the so-called Poiseuille limit.
- For **Biot's high frequency range** also select **Specify** as **Reference frequency** or **Characteristic pore size**. Either enter a **Reference frequency** f_r (SI unit: Hz) or a **Characteristic pore size** a (SI unit: m). This model implements a correction factor to the viscosity that accounts for the relative scale difference between a typical pore diameter and the acoustic boundary layer thickness. The modified viscosity is of the form

$$\tilde{\mu}_f = \mu_f F\left(\sqrt{\frac{f}{f_r}}\right) \qquad f_r = \frac{\mu_f}{2\pi a^2 \rho_f}$$

where f_r is the reference frequency and a is a characteristic size of the pores. The expression for f_r is one typically used in the literature but it is often measured or empirically determined. The expression for f_r corresponds to finding the frequency at which the viscous boundary layer thickness is of the scale a.

See High Frequency Correction (Biot Model) for more details.

Fluid Parameters for Biot–Allard (Thermal and Viscous Losses) Model

- **Density** to define the density of the saturating fluid ρ_f (SI unit: kg/m^3).

- **Dynamic viscosity** to define the dynamic viscosity of the saturating fluid μ_f (SI unit: Pa·s). The parameter is important for the amount of viscous damping experienced by the acoustic waves.
- **Ratio of specific heats** to define the ratio of specific heats (adiabatic index) of the saturating fluid γ (dimensionless).
- **Heat capacity at constant pressure** to define the (specific) heat capacity at constant pressure of the saturating fluid C_p (SI unit: J/(kg·K)).
- **Thermal conductivity** to define the thermal conductivity of the saturating fluid $\mathbf{k}$ (SI unit: W/(m·K)). The parameter is important for the amount of thermal damping experienced by the acoustic waves.

Anisotropic Poroelastic Material

Use the **Anisotropic Poroelastic Material** node to define the poroelastic material and fluid properties of an anisotropic porous material. Anisotropic properties can here be defined for the elastic matrix material properties as well as the relevant poroacoustic properties (flow resistivity, tortuosity factor, and viscous characteristic length).

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains all applicable coordinate systems in the component. The coordinate system is used for interpreting directions of orthotropic and anisotropic material data and when stresses, strains, and porous properties are presented in a local system.

Coordinate systems with directions which change with time should not be used.

POROELASTIC MODEL

Select the Model as either **Biot (viscous losses)** or **Biot–Allard (thermal and viscous losses)** (the default). See the Poroelastic Material for details on the options.

POROUS MATRIX PROPERTIES

Select a **Porous model**: **Drained matrix, isotropic**; **Drained matrix, orthotropic**; or **Drained matrix, anisotropic**. Then enter or select the settings as described.

First enter the mechanical properties of the drained porous matrix. The drained parameters are also sometimes referred to as the in-vacuo elastic parameters; they are in principle measured without the presence of the saturating fluid:

- Porous Drained Matrix, Isotropic
- Porous Drained Matrix, Orthotropic
- Porous Drained Matrix, Anisotropic

Then (below the dividing line) enter the porous (or poroacoustic) parameters of the poroelastic material:

- Porous Parameters for Biot (Viscous Losses) (Anisotropic)
- Porous Parameters for Biot–Allard (Thermal and Viscous Losses) (Anisotropic)

Porous Drained Matrix, Isotropic

If **Drained matrix, isotropic** is selected (the default) from the **Porous model** list, select a pair of elastic properties to describe an isotropic drained porous material. The drained parameters are also sometimes referred to as the in-vacuo elastic parameters; they are in principle measured without the presence of the saturating fluid.

From the **Specify** list, select:

- **Young's modulus and Poisson's ratio** to specify drained Young's modulus (elastic modulus) E_d (SI unit: Pa) and Poisson's ratio ν_d (dimensionless). For an isotropic material, Young's modulus is the spring stiffness in Hooke's law, which in 1D form is $\sigma = E_d\varepsilon$, where σ is the stress and ε is the strain. Poisson's ratio defines the normal strain in the perpendicular direction, generated from a normal strain in the other direction and follows the equation $\varepsilon_\perp = -\upsilon\varepsilon_\parallel$.
- **Shear modulus and Poisson's ratio** (the default for the Biot–Allard model) to specify drained shear modulus G_d (SI unit: Pa) and Poisson's ratio ν_d (dimensionless).
- **Young's modulus and Shear modulus** to specify drained Young's modulus (elastic modulus) E_d (SI unit: Pa) and drained shear modulus G_d (SI unit: Pa).
- **Bulk modulus and shear modulus** (the default for the Biot model) to specify the drained bulk modulus K_d (SI unit: Pa) and the drained shear modulus G_d (SI unit: Pa). The bulk drained modulus is a measure of the solid porous matrix's resistance to volume changes. The shear modulus is a measure of the solid porous matrix's resistance to shear deformations.

- **Lamé parameters** to specify the drained Lamé parameters λ_d (SI unit: Pa) and μ_d (SI unit: Pa).
- **Pressure-wave and shear-wave speeds** to specify the drained pressure-wave speed c_p (SI unit: m/s) and the shear-wave speed c_s (SI unit: m/s).

Then also enter (also for the orthotropic and anisotropic drained matrix options):

- **Drained density of porous material** to specify the drained density of the porous material in vacuum ρ_d (SI unit: kg/m^3). The drained density ρ_d is equal to $(1 - \varepsilon_p)\rho_s$ where ρ_s is the density of the solid material from which the matrix is made and ε_p is the porosity.
- **Isotropic structural loss factor** (only for the Biot-Allard model) to specify the loss factor of the porous matrix η_s (dimensionless). This value introduces the damping due to losses in the porous structure by transform the elastic moduli into complex valued quantities. This quantity can be frequency dependent if necessary.

For each pair of properties, select from the applicable list to use the value **From material** or enter a **User defined** value or expression. Each of these pairs define the drained elastic properties and it is possible to convert from one set of properties to another.

Porous Drained Matrix, Orthotropic

When **Drained matrix, orthotropic** is selected from the **Porous model** list, the material properties of the solid porous matrix vary in orthogonal directions only.

Select the **Material data ordering** as **Standard (11, 22, 33, 12, 23, 13)** or **Voigt (11, 22, 33, 23, 13, 12)** (the default).

The default properties take values **From material**. For **User defined**, enter values or expressions for the drained **Young's modulus** $\mathbf{E}$ (SI unit: Pa), the drained **Poisson's ratio** ν (dimensionless), and the drained **Shear modulus** $\mathbf{G}$ (SI unit: Pa).

Porous Drained Matrix, Anisotropic

When **Drained matrix, anisotropic** is selected from the **Porous model** list, the material properties of the solid porous matrix vary in all directions, and the stiffness comes from the symmetric **Elasticity matrix**, $\mathbf{D}$ (SI unit: Pa). The default uses values **From material**. For **User defined** enter values in the 6-by-6 symmetric matrix that displays.

Select the **Material data ordering** as **Standard (11, 22, 33, 12, 23, 13)** or **Voigt (11, 22, 33, 23, 13, 12)** (the default).

Porous Parameters for Biot (Viscous Losses) (Anisotropic)

In the **Anisotropic Poroelastic Material** model the **Permeability** $[\kappa_p]_{ij}$ and the **Tortuosity factor** $[\tau_\infty]_{ij}$ can have anisotropic properties. The remaining (isotropic) parameters are described in the Porous Parameters for Biot (Viscous Losses) section.

For the two inputs select either **From material** (the default) or **User defined**. For the **User defined** option, select if the material property is entered as **Isotropic** or **Diagonal** (the properties only have nonzero components in the diagonal for i = j). When taken **From material**, the properties can now be defined as diagonal tensor data, when defining a material in the **Materials** node.

Porous Parameters for Biot–Allard (Thermal and Viscous Losses) (Anisotropic)

In the **Anisotropic Poroelastic Material** model the **Flow resistivity** $[R_f]_{ij}$, the **Tortuosity factor** $[\tau_\infty]_{ij}$, and the **Viscous characteristic length** $[L_v]_{ij}$ can have anisotropic properties. The remaining (isotropic) parameters are described in the Porous Parameters for Biot–Allard (Thermal and Viscous Losses) section.

For the three inputs select either **From material** (the default) or **User defined**. For the **User defined** option, select if the material property is entered as **Isotropic** or **Diagonal** (the properties only have nonzero components in the diagonal for i = j). When taken **From material**, the properties can now be defined as diagonal tensor data, when defining a material in the **Materials** node.

FLUID PROPERTIES

For details see the Fluid Properties in the (isotropic) Poroelastic Material model.

- *Transverse Isotropic Porous Layer*: Application Library path **Acoustics_Module/Building_and_Room_Acoustics/transverse_isotropic_porous_layer**
- *Anisotropic Porous Absorber*: Application Library path **Acoustics_Module/Building_and_Room_Acoustics/anisotropic_porous_absorber**

Body Load

The **Body Load** condition can be added from the **Porous Matrix Conditions** submenu. It has two options that are described in the Body Load for the The Solid Mechanics Interface.

Impervious Layer

The **Impervious Layer** acts as a wall or sound hard boundary for the pressure fluctuations in the saturating fluid. The condition is one of the two default boundary conditions for The Poroelastic Waves Interface. The condition can be added from the **Fluid Conditions** submenu and it can be combined with any of the conditions found in the **Porous Matrix Conditions** submenu.

Free

The **Free** condition lets the porous matrix move freely (not constrained) at the selected boundary. The condition is one of the two default boundary conditions for The Poroelastic Waves Interface. The condition can be added from the **Porous Matrix Conditions** submenu and it can be combined with any of the conditions found in the **Fluid Conditions** submenu.

Initial Values

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

INITIAL VALUES

Enter a value or expression for the **Pressure** p (SI unit: Pa) and **Displacement field** $\mathbf{u}$ (SI unit: m) initial values. The default is 0 Pa for the pressure and 0 m for the displacement field.

Impervious Mass Layer

The **Impervious Mass Layer** condition adds a mixed condition for the porous matrix as well as the saturating fluid. A distributed mass is applied on the boundary resulting an effective (frequency and displacement dependent) load for the elastic matrix, while an impervious (sound hard) condition is applied to the saturating fluid movement.

ADDED MASS

Select the **Mass type** as **Mass per reference area** or **Total mass**, then enter the desired mass density ρ_A or total mass $\mathbf{m}$. For both select if the mass distribution is **Isotropic**, **Diagonal**, or **Symmetric**.

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and physics interfaces being modeled.

PERIODICITY SETTINGS

Select a **Type of periodicity**: **Continuity** (the default), **Antiperiodicity, Floquet periodicity** (Bloch periodicity), **Cyclic symmetry**, or **User defined**. For **User defined**, select one or more of the following checkboxes; and then also select the **Type of periodicity** as **Continuity** or **Antiperiodicity** for the component:

- **Periodic displacement in the x direction**
- **Periodic displacement in the y direction**
- **Periodic displacement in the z direction** (3D only)
- **Periodic pressure**

For details, see the Periodic Condition description in Pressure Acoustics.

To get correct behavior of the **Periodic Condition**, in particular the **Floquet periodicity** and the **Cyclic symmetry** conditions, it is recommended that the source and destination meshes are identical. This can be achieved using the **Identical Mesh** mesh feature. When the **Periodic Condition** stretches across regions with a mix of default material models, PMLs, background pressure fields, or background acoustic fields, it is recommended to add one **Periodic Condition** for each set of such boundaries.

Free, Sound Soft

The **Free, Sound Soft** node adds a mixed condition, with no constraints and no loads on the porous matrix, and a sound-soft boundary for the saturating fluid pressure.

Porous, Free (Sound-Soft Boundary) Theory

Pressure

The **Pressure** node adds a mixed condition that acts as a pressure source/load at the boundary, which means a constant acoustic pressure $p = p_0$ is specified. The condition is applied to the saturating fluid as well as the porous matrix. In the frequency domain, p_0 is the amplitude of a harmonic pressure source.

PRESSURE

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

CONSTRAINT SETTINGS

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

Pressure Equations

Symmetry

The **Symmetry** node adds a boundary condition where there is symmetry in the pressure and displacement field. Use this condition to reduce the size of a model by cutting it in half where there are symmetries. Additional options apply to the displacement field normal component.

NORMAL DIRECTION CONDITION

You can allow a symmetry plane to move along its normal direction. This can be used to model situations where you want a plane to remain strictly planar but still relax the property of it being fixed.

From the list, select **No displacement** (the default), **Free Displacement**, **Prescribed force**, or **Prescribed displacement**.

- The value **No displacement** gives a standard symmetry condition.
- Select **Free Displacement** to allow the symmetry plane to translate in the normal direction. The displacement is determined by the criterion that there is no resulting reaction force in the normal direction.
- Select **Prescribed force** to prescribe the total reaction force acting on the direction normal to the symmetry plane. Enter the **Normal force** F_n. The force is defined as

positive when acting along the outward normal of the symmetry plane. Setting the prescribed force to zero gives the same effect as using **Free Displacement**.

- Select **Prescribed displacement** to prescribe the displacement in the direction normal to the symmetry plane. Enter the **Normal displacement** u_{n0}. Setting the prescribed displacement to zero gives the same effect as using **No displacement**.

CONSTRAINT SETTINGS

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

Symmetry Plane

The **Symmetry Plane** nodes adds the symmetry boundary condition in a 2D axisymmetric model. It only applies to the axial z-component of the displacement.

See the Normal Direction Condition settings for the Symmetry condition.

CONSTRAINT SETTINGS

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

Boundary Load

The **Boundary Load** condition can be added from the **Porous Matrix Conditions** submenu. It has four options that are described in the Boundary Load for the The Solid Mechanics Interface.

Fixed Constraint

The **Fixed Constraint** node adds a condition that makes the porous matrix fixed (fully constrained); that is, the displacements are zero in all directions. The condition can be added from the **Porous Matrix Conditions** submenu and it can be combined with any of the conditions found in the **Fluid Conditions** submenu.

Prescribed Displacement

The **Prescribed Displacement** node adds a condition where the displacements in one or more directions are prescribed for the porous matrix domain, boundary, edge, or point (depending on space dimension). The condition can be added from the **Porous Matrix**

Conditions submenu and it can be combined with any of the conditions found in the **Fluid Conditions** submenu.

If a displacement is prescribed in one direction, this leaves the porous matrix free to deform in the other directions. The condition can be added from the **Porous Matrix Conditions** submenu and it can be combined with any of the conditions found in the **Fluid Conditions** submenu.

- If a prescribed displacement is not activated in any direction, this is the same as a **Free** constraint.
- If a zero displacement is applied in all directions, this is the same as a **Fixed Constraint** for the porous matrix.

COORDINATE SYSTEM SELECTION

The constraints are applied in the directions of the coordinate system selected in this section. You should only use a coordinate system that has fixed spatial directions.

PRESCRIBED DISPLACEMENT

For the displacement in each direction, select **Free** or **Prescribed**. With **Free** (the default) the displacement component is left unconstrained. Use **Prescribed** to constrain the displacement to a given value.

CONSTRAINT SETTINGS

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

Prescribed Displacement Equations

Prescribed Velocity

The **Prescribed Velocity** node adds a boundary condition where the velocity of the porous matrix is prescribed in one or more directions. With this boundary condition, it is possible to prescribe a velocity in one direction, leaving the porous matrix free in the other directions. The condition can be added from the **Porous Matrix Conditions** submenu and it can be combined with any of the conditions found in the **Fluid Conditions** submenu.

COORDINATE SYSTEM SELECTION

	Coordinate systems with directions which change with time should not be used.

PRESCRIBED VELOCITY

	To define a porous, prescribed velocity for each space direction (x, y, and z for 3D), select one or all of the **Prescribed in x direction**, **Prescribed in y direction**, and **Prescribed in z direction** checkboxes. Then enter a value or expression for the components v_x, v_y, and v_z (SI unit: m/s).
	For 2D axisymmetric components and to define a prescribed velocity for each space direction (r and z), select one or both of the **Prescribed in r direction** and **Prescribed in z direction** checkboxes. Then enter a value or expression for v_r and v_z (SI unit: m/s).
	Prescribed Velocity Equations

Prescribed Acceleration

The **Prescribed Acceleration** node adds a boundary condition, where the acceleration of the porous matrix is prescribed in one or more directions. With this boundary condition, it is possible to prescribe an acceleration in one direction, leaving the porous matrix free in the other directions. The condition can be added from the **Porous Matrix Conditions** submenu and it can be combined with any of the conditions found in the **Fluid Conditions** submenu.

COORDINATE SYSTEM SELECTION

	Coordinate systems with directions which change with time should not be used.

PRESCRIBED ACCELERATION

To define a porous, prescribed acceleration for each space direction (x, y, and z for 3D), select one or all of the **Prescribed in x direction**, **Prescribed in y direction**, and **Prescribed in z direction** checkboxes. Enter a value or expression for the prescribed acceleration a_x, a_y, and a_z (SI unit: $\mathrm{m/s^2}$).

For 2D axisymmetric components and to define a porous, prescribed acceleration for each space direction (r and z), select one or both of the **Prescribed in r direction** and **Prescribed in z direction** checkboxes. Then enter a value or expression for the prescribed acceleration a_r and a_z (SI unit: $\mathrm{m/s^2}$).

Prescribed Acceleration Equations

Roller

The **Roller** node adds a roller (sliding wall) constraint as the boundary condition; that is, the porous matrix displacement is zero in the direction perpendicular (normal) to the boundary, but the porous matrix is free to move in the tangential direction. The condition can be added from the **Porous Matrix Conditions** submenu and it can be combined with any of the conditions found in the **Fluid Conditions** submenu.

CONSTRAINT SETTINGS

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

Roller Equations

Thin Elastic Layer

The **Thin Elastic Layer** node to apply elastic and damping conditions between two parts on an interior boundary. The condition can be added from the **Porous Matrix Conditions**

submenu and it can be combined with any of the conditions found in the **Fluid Conditions** submenu.

The **Thin Elastic Layer** and Spring Foundation nodes are similar, with the difference that a Spring Foundation connects the structural part on which it is acting to a fixed "ground".

COORDINATE SYSTEM SELECTION

The spring and damping constants are given with respect to the selected coordinate system, with the exception that when the stiffness is specified by **Use material data**, the coordinate system selection is not used.

SPRING

Select the **Spring type** and its associated spring constant or force using Table 3-1 as a guide. The default option is the spring type for the type of geometric entity and space dimension, and there are different combinations available based on this.

When the option is of the type "force as function of extension", then the built-in variables describing the spring extension must be used in the expression as described in Springs and Dampers. The spring matrix can be entered as **Isotropic**, **Diagonal**, **Symmetric**, or **Full**. For **Isotropic** the same spring constant is used in all the diagonal elements of the spring matrix.

When **Use material data** is selected as **Spring type**, the spring stiffness values are computed from material data and layer thickness. From the **Specify** list, select a pair of elastic properties — **Young's modulus and Poisson's ratio**, **Young's modulus and shear modulus**, or **Bulk modulus and shear modulus**. Each of these pairs define the elastic properties and it is possible to convert from one set of properties to another according to Table 3-3. For the chosen properties, select from the applicable list to use the value **From material** or enter a **User defined** value or expression. In order to use **From material**, you must have assigned a material to the selected boundaries.

Enter a **Thickness**, d_s, to specify the physical thickness of the elastic layer.

TABLE 3-1: SPRING TYPES FOR THE THIN ELASTIC LAYER FEATURE.

SPRING TYPE	VARIABLE	SI UNITS	SPACE DIMENSION
Total spring constant	$\mathbf{k}_{tot}$	N/m	3D, 2D, and 2D axisymmetric
Spring constant per reference area	$\mathbf{k}_A$	N/(m·m^2)	3D, 2D, and 2D axisymmetric
Spring constant per reference length	$\mathbf{k}_L$	N/(m·m)	2D

TABLE 3-1: SPRING TYPES FOR THE THIN ELASTIC LAYER FEATURE.

SPRING TYPE	VARIABLE	SI UNITS	SPACE DIMENSION
Total force as function of extension	$\mathbf{F}_{tot}(\mathbf{u}_s)$	N	3D, 2D, and 2D axisymmetric
Force per area as function of extension	$\mathbf{f}_A(\mathbf{u}_s)$	N/m^2	3D, 2D, and 2D axisymmetric
Force per length as function of extension	$\mathbf{f}_L(\mathbf{u}_s)$	N/m	2D
Use material data	N/A	N/A	3D, 2D, and 2D axisymmetric

ADDED MASS

Enter the added mass of the thin elastic layer. The type of input depends on the selected **Spring type**; see Table 3-1.

- For **Total spring constant** and **Total force as function of extension**, enter the **Total mass** m of the thin elastic layer.
- For **Spring constant per reference area** and **Force per area as function of extension**, enter the **Mass per reference area** ρ_A of the thin elastic layer.
- For **Spring constant per reference length** and **Force per length as function of extension**, enter the **Mass per reference length** ρ_L of the thin elastic layer.
- For **Use material data**, enter the **Density** ρ of the thin elastic layer. Either use the value **From material** or enter a **User defined** value or expression. In order to use **From material**, you must have assigned a material to the selected boundaries.

LOSS FACTOR DAMPING

From the **Loss factor type** list, select **Scalar (Same for all components)** or **Individual components**.

- For **Scalar (Same for all components)** enter a single **Loss factor for spring** η_s, which is used to multiply all values of the spring matrix or spring force vector.
- For **Individual components** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the table for the **Loss factor for spring** η_s or η_f based on space dimension. The loss factors act on the corresponding components of the spring matrix or spring force vector. If you select **Isotropic**, the effect is the same as when you select **Diagonal** and enter the same value for all diagonal elements.

VISCOUS DAMPING

Select the **Damping type** using Table 3-2 as a guide. The default option is the default damping type for the space dimension. The damping matrix can be entered as **Isotropic**, **Diagonal**, **Symmetric**, or **Full**. For **Isotropic** the same viscous constant is used in all the diagonal elements of the damping matrix.

TABLE 3-2: DAMPING TYPES FOR THE THIN ELASTIC LAYER FEATURE.

DAMPING TYPE	VARIABLE	SI UNITS	SPACE DIMENSION
Damping constant per reference area	$\mathbf{d}_A$	N·s/(m·m^2)	3D, 2D, and 2D axisymmetric
Total damping constant	$\mathbf{d}_{tot}$	N·s/m	3D, 2D, and 2D axisymmetric
Damping constant per reference length	$\mathbf{d}_L$	N·s/(m·m)	2D

ENERGY DISSIPATION

You can select to compute energy dissipation variables. Select the **Calculate dissipated energy** checkbox as needed to compute the energy dissipated by damping.

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

Added Mass

The **Added Mass** condition can be added from the **Porous Matrix Conditions** submenu. It has two options that are described in the Added Mass for the The Solid Mechanics Interface.

Interior Impervious Layer

The **Interior Impervious Layer** condition acts as an interior sound hard wall for the saturating fluid in the porous material. The condition adds a so-called slit to the pressure degree of freedom and makes it discontinuous at the boundary. Note that the condition does not add a slit to the displacement degree of freedom. The condition can be added from the **Fluid Conditions** submenu and it can be combined with any of the conditions found in the **Porous Matrix Conditions** submenu.

Edge Load

The **Edge load** condition can be added from the **Edges** submenu. It applies to the porous matrix. It has two options that are described in the Edge Load for the The Solid Mechanics Interface.

Point Load

The **Point Load** condition can be added from the **Points** submenu. It applies to the porous matrix. It has two options that are described in the Point Load for the The Solid Mechanics Interface.

Ring Load

The **Ring Load** condition can be added from the **Points** submenu. It applies to the porous matrix and exists only in 2D axisymmetric models. It has two options that are described in the Ring Load for the The Solid Mechanics Interface.

Point Load (on Axis)

The **Point Load (on Axis)** condition can be added from the **Points** submenu. It applies to the porous matrix and exists only in 2D axisymmetric models. It has one option that are described in the Point Load (on Axis) for the The Solid Mechanics Interface.

The Elastic Waves, Time Explicit Interface

The **Elastic Wave, Time Explicit** interface ( ), found under the **Acoustics** > **Elastic Waves** branch ( ) when adding a physics interface, is used to model the transient propagation of elastic waves in linear elastic materials. It is used to compute the velocity and strain field in solids with propagating elastic waves. It is suited for time-dependent simulations with arbitrary time-dependent sources and fields. In general, the interface is suited for modeling the propagation of elastic waves over large distances relative to the wavelength, for example, ultrasound propagation for nondestructive testing (NDT), or seismic waves. The dedicated Fracture condition is useful for many of these applications. The interface includes absorbing layers that are used to set up effective nonreflecting like boundary conditions (sponge layers).The interface exists in 2D, 2D axisymmetry, and 3D.

The interface is based on the discontinuous Galerkin (dG or dG-FEM) method and uses a time explicit solver. The method is very memory efficient and can solve problems with many million degrees of freedom (DOFs). The method is also well suited for distributed computing on clusters.

The **Elastic Waves, Time Explicit** interface can be coupled to The Pressure Acoustics, Time Explicit Interface and The Nonlinear Pressure Acoustics, Time Explicit Interface, using either the Acoustic–Structure Boundary, Time Explicit or the Pair Acoustic–Structure Boundary, Time Explicit multiphysics couplings. It can also couple to the The Convected Wave Equation, Time Explicit Interface using the Convected Acoustic–Structure Boundary, Time Explicit or the Pair Convected Acoustic–Structure Boundary, Time Explicit multiphysics couplings.

For modeling linear piezoelectric wave problems, the **Elastic Waves, Time Explicit** interface can be coupled to the **Electrostatics** interface; see The Piezoelectric Waves, Time Explicit Interface.

The interface solves the equations of linear elasticity in the velocity-strain formulation. The dependent variables are the structural velocity $\mathbf{v}$ (SI unit: m/s) and the strain $\mathbf{E}$ (SI unit: 1). Structural damping can be introduced using a Rayleigh damping model. The displacement $\mathbf{u}$ (SI unit: m) can be computed by solving additional equations, see Computing the Displacement in the Elastic Waves, Time Explicit.

When the **Elastic Wave, Time Explicit** interface is added, these default nodes are also added to the **Model Builder**: **Elastic Wave, Time Explicit Model**, **Free**, and **Initial Values**.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Elastic Wave, Time Explicit** to select physics features from the context menu.

SETTINGS

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

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

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

- *Isotropic-Anisotropic Sample: Elastic Wave Propagation*. Application Library path **Acoustics_Module/Elastic_Waves/isotropic_anisotropic_sample**
- *Ground Motion After Seismic Event: Scattering off a Small Mountain*. Application Library path **Acoustics_Module/Elastic_Waves/ground_motion_seismic_event**
- *Propagation of Seismic Waves Through Earth*. Application Library path **Acoustics_Module/Elastic_Waves/seismic_waves_earth**

FILTER PARAMETERS FOR ABSORBING LAYERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. In the **Filter Parameters for Absorbing Layers** section you can change and control the values set for the filter used in the Absorbing Layers in Elastic Waves, Time Explicit. The values of the filter parameters defined here are used in all absorbing layers added to the model and they override the value of filter parameters enabled in the material model (Elastic Waves, Time Explicit Model). The default values of the filter parameters α, η_c, and s are set to 0.2, 0.01, and

2, respectively. Inside the absorbing layer, it is important to use a filter that is not too aggressive since this will result in spurious reflections.

	For general information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

2D APPROXIMATION

The out-of-plane components of the velocity and the strain are not computed per default in 2D. This can be changed by selecting **Include out-of-plane components** checkbox. It can be relevant in some cases, for example for anisotropic materials, where the out-of-plane velocity may become different from zero for in-plane loads. This formulation is often referred to as 2.5D. Note that this comes at an extra computational cost.

This section is only visible in 2D components.

THICKNESS

Enter a value for the thickness d. The default value is `1`, which represents a unit-thickness slice. Make sure that the same thickness assumption is used everywhere when you manually combine the Elastic Waves, Time Explicit interface with other physics interfaces that have an explicit thickness property. When using other unit systems than the default SI system, you need to pay special attention to the values of this property.

This section is only visible in 2D components.

TRANSIENT MESH SETTINGS

Enter the **Maximum frequency to resolve** $f_{\max}$ (the default is `1000[Hz]`). This value is used to set up the physics-controlled mesh for the Elastic Waves, Time Explicit interface.

DISCRETIZATION

In this section you can select the discretization for the **Structural velocity** and **Strain tensor, Voigt notation**. Per default both are set to **Quartic** (4th order). Using quartic

elements together with a mesh size equal to approximately half the wavelength to be resolved, leads to the best performance when using the dG method.

When solving **Elastic Waves, Time Explicit**, it is important to have consistent settings for the Geometry Shape Function and the Discretization of the physics. The **Automatic** setting for the **Geometry shape function** (in the Curved Mesh Elements section on the Components node's settings) may results in a linear geometry representation, if other physics are present in the model. This can lead to numerical errors when solving as the default is to use fourth-order (**Quartic**) spatial discretization of the dependent variables. To remedy this change the **Geometry shape function** to **Quadratic Lagrange**.

If the settings in a model are inconsistent, a warning is given in the solver with the text:

```
The geometry shape function order is not consistent with
the discretization of the dependent variables.
Consider increasing the geometry shape function order to
Quadratic Lagrange in the Curved Mesh Element section in
the Component node's settings.
```

DEPENDENT VARIABLES

The dependent variables (field variables) are the **Structural velocity** and **Structural velocity, components**, and the **Strain tensor, Voigt notation** and **Strain tensor, Voigt notation, components**. The names can be changed, but the names of fields and dependent variables must be unique within a model.

- For information about modeling strategies, meshing, the absorbing layer and more, see Modeling with the Elastic Waves Branch.
- For the theoretical background of the model solved, see Theory for the Elastic Waves, Time Explicit Interface.

Domain, Boundary, Edge, Point, and Pair Nodes for the Elastic Waves, Time Explicit Interface

The Elastic Waves, Time Explicit Interface has these domain, boundary, edge, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context

menu (all users). Continuity in velocity and stress is the default condition on interior boundaries. If a boundary separates two different materials, the Material Discontinuity or Continuity (the pair version of material discontinuity) condition has to be applied to ensure the continuity in velocity and stress.

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

- Axial Symmetry
- Antisymmetry
- Body Load
- Boundary Load
- Compute Displacement
- Continuity
- Damping
- Elastic Waves, Time Explicit Model
- Initial Values
- Fixed
- Fracture
- Free
- General Flux/Source
- General Interior Flux
- Low-Reflecting Boundary
- Material Discontinuity
- Prescribed Velocity
- Roller
- Symmetry

The Piezoelectric Material feature and the Mechanical Damping subfeature are describe in The Piezoelectric Waves, Time Explicit Interface section.

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Elastic Waves, Time Explicit Model

The **Elastic Waves, Time Explicit Model** node adds the equations for modeling the propagation of linear elastic waves. Define the properties of a general isotropic or

anisotropic material. The model solves the governing equations for a general linear elastic material in a velocity-strain formulation

$$\rho\frac{\partial \mathbf{v}}{\partial t} - \nabla \cdot \mathbf{S} = \mathbf{F}_v$$

$$\frac{\partial \varepsilon}{\partial t} - \frac{1}{2}[\nabla \mathbf{v} + (\nabla \mathbf{v})^T] = \mathbf{0}$$

$$\mathbf{S} = \mathbf{C}:\varepsilon$$

where $\mathbf{v}$ is the velocity, ρ the density, $\mathbf{S}$ the stress tensor, ε the strain tensor, $\mathbf{C}$ is the elasticity tensor (or stiffness tensor), and $\mathbf{F}_v$ is a possible body force. The equations are valid for both isotropic and anisotropic material data. Bulk dissipation can be added by using the Damping subnode.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains all applicable coordinate systems in the component. The coordinate system is used for interpreting directions of orthotropic and anisotropic material data and when stresses or strains are presented in a local system.

Coordinate systems with directions which change with time should not be used.

LINEAR ELASTIC MATERIAL

Define the **Solid model** representation of the material. Choose:

- **Isotropic** for a linear elastic material that has the same properties in all directions.
- **Orthotropic** for a linear elastic material that has different material properties in orthogonal directions, so that its stiffness depends on the properties E_i, ν_{ij}, and G_{ij}.
- **Anisotropic** for a linear elastic material that has different material properties in different directions, and the stiffness comes from the symmetric elasticity matrix, D

If orthotropic or anisotropic is selected then also select the **Material data ordering** to **Voigt** (the default) or **Standard** notation. This option defines what numbering and notation is used for entering user defined material data and when defining material properties in the **Materials** node.

Specification of Elastic Properties for Isotropic Materials

For an **Isotropic Solid model**, from the **Specify** list select a pair of elastic properties for an isotropic material — **Young's modulus and Poisson's ratio**, **Young's modulus and shear**

modulus, **Bulk modulus and shear modulus**, **Lamé parameters**, or **Pressure-wave and shear-wave speeds**. For each pair of properties, select from the applicable list to use the value **From material** or enter a **User defined** value or expression.

Each of these pairs define the elastic properties and it is possible to convert from one set of properties to another according to Table 3-3.

TABLE 3-3: EXPRESSIONS FOR THE ELASTIC MODULI.

DESCRIPTION	VARIABLE	$D(E,\nu)$	$D(E,G)$	$D(K,G)$	$D(\lambda,\mu)$
Young's modulus	E =	E	E	$\frac{9KG}{3K+G}$	$\mu\frac{3\lambda+2\mu}{\lambda+\mu}$
Poisson's ratio	ν =	ν	$\frac{E}{2G}-1$	$\frac{1}{2}\left(1-\frac{3G}{3K+G}\right)$	$\frac{\lambda}{2(\lambda+\mu)}$
Bulk modulus	K =	$\frac{E}{3(1-2\nu)}$	$\frac{EG}{3(3G-E)}$	K	$\lambda+\frac{2\mu}{3}$
Shear modulus	G =	$\frac{E}{2(1+\nu)}$	G	G	μ
Lamé parameter λ	λ =	$\frac{E\nu}{(1+\nu)(1-2\nu)}$	$\frac{G(E-2G)}{3G-E}$	$K-\frac{2G}{3}$	λ
Lamé parameter μ	μ =	$\frac{E}{2(1+\nu)}$	G	G	μ
Pressure-wave speed	c_p =	$\sqrt{\frac{E(1-\nu)}{\rho(1+\nu)(1-2\nu)}}$	$\sqrt{\frac{G(4G-E)}{\rho(3G-E)}}$	$\sqrt{\frac{K+4G/3}{\rho}}$	$\sqrt{\frac{\lambda+2\mu}{\rho}}$
Shear-wave speed	c_s =	$\sqrt{\frac{E}{2\rho(1+\nu)}}$	$\sqrt{G/\rho}$	$\sqrt{G/\rho}$	$\sqrt{\mu/\rho}$

The individual property parameters are:

- **Young's modulus** (elastic modulus) E.
- **Poisson's ratio** ν.
- **Shear modulus** G.
- **Bulk modulus** K.
- **Lamé parameter** λ and **Lamé parameter** μ.
- **Pressure-wave speed** (longitudinal wave speed) c_p.
- **Shear-wave speed** (transverse wave speed) c_s. This is the wave speed for a solid continuum.

Specification of Elastic Properties for Orthotropic Materials

When **Orthotropic** is selected from the **Solid model** list, the material properties vary in orthogonal directions only. Select the **Transversely isotropic** checkbox to define a material with one principal direction that differs from the others.

The **Material data ordering** can be specified in either **Standard** or **Voigt** notation. When **User defined** is selected in 3D, enter three values in the fields for **Young's modulus** $\mathbf{E}$, **Poisson's ratio** ν, and the **Shear modulus** $\mathbf{G}$. This defines the relationship between engineering shear strain and shear stress. It is applicable only to an *orthotropic material* and follows the equation

$$\varepsilon_{ij} = \frac{\tau_{ij}}{G_{ij}}$$

ν_{ij} is defined differently depending on the application field. It is easy to transform among definitions, but check which one the material uses.

For the **Transversely isotropic** option, one principal direction in the material is different from two others that are equivalent. This special direction is assumed to be the first axis of the selected coordinate system. Because of the symmetry, the following relations hold:

$$E_3 = E_2$$

$$\nu_{13} = \nu_{12}$$

$$G_{23} = \frac{E_2}{2(1+\nu_{23})}$$

$$G_{12} = G_{13}$$

Thus, only five elasticity moduli need to be entered when the **User defined** option is selected.

Specification of Elastic Properties for Anisotropic Materials

When **Anisotropic** is selected from the **Solid model** list, the material properties vary in all directions, and the stiffness comes from the symmetric **Elasticity matrix**, $\mathbf{D}$. You can also choose to specify the compliance matrix $\mathbf{D}^{-1}$ instead. The **Material data ordering**

can be specified in either **Standard** or **Voigt** notation. When **User defined** is selected, a 6-by-6 symmetric matrix is displayed.

Note that the 6-by-6 **Elasticity matrix**, $\mathbf{D}$ has to be complete regardless of the spatial dimension of the problem. This is due to the generalized plane strain formulation of the governing equations solved in 2D.

Density

The default **Density** ρ uses values **From material**. For **User defined**, enter another value or expression. If any material in the model has a temperature dependent mass density, and **From material** is selected, the **Volume reference temperature** list will appear in the **Model Input** section. As a default, the value of T_{ref} is obtained from a **Common model input**. You can also select **User defined** to enter a value or expression for the reference temperature locally.

LAX–FRIEDRICHS FLUX PARAMETER

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog. In this section, you specify the value of the *Lax–Friedrichs flux parameter* τ_{LF} (default value: 0.2). This value controls the numerical flux between the elements (nodal discontinuous Lagrange elements) used with the discontinuous Galerkin (dG) method. The numerical flux defines how adjacent elements are connected and how continuous $\mathbf{v}$ and $\mathbf{E}$ are. Different definitions of the numerical flux lead to different variants of the dG method. The flux implemented here is the so-called global Lax–Friedrichs numerical flux. The value of the parameter τ_{LF} should be between 0 and 0.5. For $\tau_{\text{LF}} = 0$ a so-called central flux is obtained. Setting $\tau_{\text{LF}} = 0.5$ gives a maximally dissipative global Lax–Friedrichs flux.

For general information about the numerical flux see the Numerical Flux section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

FILTER PARAMETERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. By default, the filter parameters α, η_c, and s are not active. Select the **Activate** checkbox to activate the filter. The filter provides higher-order smoothing for the dG formulation and can be used to stabilize

the solution. Inside absorbing layers, the settings given here are overridden by the Filter Parameters for Absorbing Layers.

Enter values for the filter parameters in the corresponding text fields (default values: 36, 0.6, and 3). α must be positive and lie between 0 and 36. $\alpha = 0$ means no dissipation and $\alpha = 36$ means maximal dissipation. η_c should be between 0 and 1, where 0 means maximal filtering and 1 means no filtering (even if filtering is active). The s parameter should be larger than 0 and controls the order of the filtering (a dissipation operator of order $2s$). For $s = 1$, you get a filter that is related to the classical 2nd-order Laplacian. A larger s gives a more pronounced low-pass filter.

For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

ESTIMATE OF MAXIMUM WAVE SPEED

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. The maximum wave speed enters the expression for the *cell wave time scale* used by the solver to estimate a stable internal time step. Select **Automatic** (the default) or **User defined**. With the **Automatic** option the maximum speed is set equal to the pressure wave speed, for **User defined** enter another value for c_{max}. The estimate for isotropic materials is correct, but for certain configurations of anisotropic materials it can be necessary to enter a larger value to ensue numerical stability.

Damping

Use the **Damping** subnode to add Rayleigh damping to the material model (the Elastic Waves, Time Explicit Model node). The only option for the **Damping type** is **Rayleigh damping**.

In this damping model, the damping parameter ξ is expressed in terms of the mass m and the stiffness k as

$$\xi = \alpha_{dM} m + \beta_{dK} k$$

That is, Rayleigh damping is proportional to a linear combination of the stiffness and mass; there is no direct physical interpretation of the *mass damping parameter* α_{dM} and the stiffness damping parameter β_{dM}.

This feature adds a right hand side to the momentum equation of the form:

$$-\alpha_{dM}\rho\mathbf{v} + \beta_{dK}\left(\nabla \cdot \frac{\partial \mathbf{S}}{\partial t}\right)$$

DAMPING SETTINGS

Select **Input parameters** — **Alpha and beta** — to enter the damping parameters explicitly, or **Damping ratios** to derive the damping parameters from the damping ratio at two frequencies.

When **Alpha and beta** is selected, enter the **Mass damping parameter** α_{dM} and the **Stiffness damping parameter** β_{dK}.

When **Damping ratios** is selected, enter two frequencies, f_1 and f_2, and the corresponding damping ratios ζ_1 and ζ_2 at these frequencies. The Rayleigh damping parameters are computed as

$$\alpha_{dM} = 4\pi f_1 f_2 \frac{\zeta_1 f_2 - \zeta_2 f_1}{f_2^2 - f_1^2}$$

$$\beta_{dK} = \frac{\zeta_2 f_2 - \zeta_1 f_1}{\pi(f_2^2 - f_1^2)}$$

FILTER PARAMETERS

When the **Damping** subnode is added, filtering is automatically enabled on the selected domains. To display the section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. The filtering is added in order to stabilize the underlying equations. This is necessary because damping is added as a right-hand-side contribution to the governing equations in the dG-FEM formulation. In general, the values for the damping parameters should represent physical behavior. If unrealistically large values are entered, the method becomes unstable even with the filtering turned on. Note that if only very small amounts of damning are added, the filter will contribute with an additional small amount of numerical damping.

Compute Displacement

Use the **Compute Displacement** subnode (to the Elastic Waves, Time Explicit Model node) to add additional equations for computing the displacement in a point, along an edge, on a surface, or in a domain. The feature is a pure postprocessing type and the

variables can be used to visualize displacements. The feature adds and solves an ODE and thus requires additional computational resources. This is necessary since the Elastic Waves, Time Explicit interface uses the velocity-strains formulation of the governing equations. It is recommended to only use the feature where necessary.

Isotropic-Anisotropic Sample: Elastic Wave Propagation. Application Library path **Acoustics_Module/Elastic_Waves/isotropic_anisotropic_sample**

Initial Values

The **Initial Values** node adds initial values for the velocity field and strain field that can serve as an initial condition for a transient simulation. In addition to the default **Initial Values** node always present in the interface, you can add more **Initial Values** nodes if needed.

Body Load

The **Body Load** node adds a general volumetric load to the selected domain. The feature can be used to approximate sources of complex external processes.

FORCE

Select a **Load type** — **Force per volume**, **Total force**, or for 2D components, **Force per area**.

Then enter values or expressions for the components in the force vector based on the space dimension. These can be general time dependent expressions or data from interpolation functions.

For **Total force** COMSOL Multiphysics divides the total force by the volume of the domains where the load is active.

Axial Symmetry

In a 2D axisymmetric geometry, the **Axial Symmetry** condition is automatically added on the axis of symmetry at $r = 0$. On this boundary, the radial component of the velocity is set equal to zero as well as the tangential component of the traction:

$$\mathbf{e}_r \cdot \mathbf{v} = 0$$
$$\mathbf{e}_r \times \mathbf{Sn} = \mathbf{0}$$

Free

The **Free** node is the default boundary condition. It means that there are no constraints and no loads acting on the boundary. When the physics interfaces is added, a default **Free** node is added. If you look at the selections for this node, it will show all boundaries which do not have any boundary conditions applied.

Fixed

The **Fixed** node adds a condition that makes the geometric entity fixed; that is, the velocities are zero in all directions on the selected geometrical entities.

Prescribed Velocity

The **Prescribed Velocity** node adds a boundary condition where the velocity is prescribed in one or more directions. It is possible to prescribe a velocity in one or more directions, leaving the solid fixed in the other directions (the default 0 value).

PRESCRIBED VELOCITY

Enter the velocity vector $\mathbf{v}_0$ components. These can be general time dependent expressions or data from interpolation functions.

Roller

The **Roller** node adds a roller constraint as the boundary condition; that is, the displacement is zero in the direction perpendicular (normal) to the boundary, but the boundary is free to move in the tangential direction. A **Roller** condition is mathematically similar to a Symmetry condition.

Boundary Load

Use a **Boundary Load** to apply tractions or pressure to boundaries.

FORCE

Select a **Load type** — **Force per area**, **Pressure**, **Total force**, or for 2D components, **Force per length**. Then enter values or expressions for the components in the force vector based on the space dimension.

- For **Force per length**, the boundary load as force per unit area is the value of the force per length divided by the thickness.
- For **Total force**, COMSOL Multiphysics divides the total force by the area of the surfaces where the load is active.
- For **Pressure**, a scalar input is given, and the orientation of the load is given by the normal to the boundary. The pressure is positive when directed toward the solid. In a geometrically nonlinear analysis, the current surface normal and area are used.

Low-Reflecting Boundary

Use the **Low-Reflecting Boundary** node to let elastic waves pass out from the model with minimal reflections. As a default, it takes material data from the domain in an attempt to create a perfect impedance match for both pressure waves and shear waves. The condition is sensitive to the direction of the incoming wave and works best for normal incidence.

The **Low-Reflecting Boundary** should be used in combination with the Absorbing Layers in Elastic Waves, Time Explicit to create efficient absorbing conditions (sponge layers) that work for most angles of incidence. The condition should be added at the outside of the absorbing layer.

DAMPING

Select a **Damping type** — **P and S waves** or **User defined**. For **User defined** enter values or expressions for the **Mechanical impedance** $\mathbf{d}_i$. The defaults for all values are `0.5*elte.rho*(elte.cp+elte.cs)` (on the diagonal for the **Diagonal** and **Symmetric** options).

See the Low-Reflecting Boundary Condition in the theory chapter of the *Structural Mechanics Module User's Guide*.

Symmetry

The **Symmetry** node adds a boundary condition that represents symmetry in the geometry and in the loads. A symmetry condition is free in the plane and fixed in the out-of-plane direction.

Antisymmetry

The **Antisymmetry** node adds a boundary condition for an antisymmetry boundary. An antisymmetry condition is fixed in the plane and free in the out-of-plane direction.

Continuity

The **Continuity** node, exists as an option at interfaces between parts in an assembly, from the **Pairs** submenu. The condition is similar to the Material Discontinuity condition that applies to interior boundaries. The difference is that the **Continuity** condition should also be applied both when the same material exists on the two sides of a pair and when the material is different. The advantage of using a pair feature is that the mesh does not need to be conforming on the two sides of the interface (the two parts of the assembly). This is especially advantageous for the time explicit discontinuous Galerkin method as the time step depends on mesh size and local speed of sound.

The condition implements continuity in velocity and in the normal stress over the pair:

$$\mathbf{v}_{\text{src}} = \mathbf{v}_{\text{dst}}$$
$$\mathbf{S}_{\text{src}}\mathbf{n}_{\text{src}} = -\mathbf{S}_{\text{dst}}\mathbf{n}_{\text{dst}}$$

Subscripts “src” and “dst” in the equation refer to the source and destination in the pair; that is, the two sides in the pair.

General Flux/Source

The **General Flux/Source** node, found under the **More** submenu, adds the most general boundary condition for the elastic waves equation, as it is formulated (in the conservative form) for the discontinuous Galerkin method (dG). The condition defines the normal flux $\mathbf{g}$ at an exterior boundary. Care should be taken when using this advanced condition as the method is sensitive to proper formulation of boundary conditions. Only one characteristic can enter the domain at any boundary at the time, meaning that it is easy to over-specify the problem.

GENERAL FLUX/SOURCE

Enter the expression for the components of the **Flux vector g**.

General Interior Flux

The **General Interior Flux** node, found under the **More** submenu, adds the most general interior boundary condition for the Elastic Waves, Time Explicit model, as it is formulated (in the conservative form) for the discontinuous Galerkin method (dG). The condition defines the normal flux **g** at an interior boundary. It should include any necessary numerical flux (as it overrides the Lax–Friedrichs flux). Care should be taken when using this advanced condition as the method is sensitive to proper formulation of boundary conditions.

GENERAL INTERIOR FLUX

Enter the expression for the components of the **Flux vector g**.

Fracture

The **Fracture** condition is used to treat two elastic domains with imperfect bonding. The fracture can, for example, be a thin elastic layer, a fluid-filled layer, or it can be a discontinuity in elastic materials (an interior boundary). Typical applications are for modeling nondestructive testing (NDT) applications, like the response of delamination regions or other defects, or for modeling wave propagation in fractured solid media in the oil and gas industry. Add this node from the **Interior Conditions** submenu or from the **Pairs** submenu.

Angle Beam Nondestructive Testing. Application Library path **Acoustics_Module/Ultrasound/angle_beam_ndt**

COORDINATE SYSTEM

Select the **Coordinate system** used to define vector and tensor quantities in the feature (the spring constants). The **Boundary System (sys1)** is used per default.

FRACTURE PROPERTIES

Select the **Stiffness type** to define the properties of the fracture, as **Stiffness per unit area**, **Stiffness per unit length**, **Total stiffness**, or **Use material data**.

For **Stiffness per unit area**, **Stiffness per unit length**, and **Total stiffness** enter the value of the spring constant ($\mathbf{k}_A$, $\mathbf{k}_L$, or $\mathbf{k}_{tot}$, respectively) with the appropriate units. Then select the tensor or matrix representation of the spring constant as **Isotropic**, **Diagonal**, or **Symmetric**. For **Isotropic,** the same spring constant is used in all the diagonal elements of the spring matrix.

When **Use material data** is selected as **Stiffness type**, the spring constant stiffness values are computed from material data and layer thickness. From the **Specify** list, select a pair of elastic properties — **Young's modulus and Poisson's ratio**, **Young's modulus and shear modulus**, or **Bulk modulus and shear modulus**. Each of these pairs define the elastic properties and it is possible to convert from one set of properties to another according to Table 3-3. For the chosen properties, select from the applicable list to use the value **From material** or enter a **User defined** value or expression. In order to use **From material**, you must have assigned a material to the selected boundaries.

Enter a **Thickness** d_s to specify the physical thickness of the elastic layer.

TIME STEPPING

Select the time stepping **Method** used to solve the system of ODEs that describes the elastic properties of the fracture. Select **Boundary ODEs** (the default) to solve the ODEs using the same method as the physics solver specified in the Study; or select **Backward Euler** to solve the ODEs using first order backward Euler.

APPLICABLE PAIR REGION

This section only exists for the interior version of the **Fracture** feature. To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options**. Select the **Allowed region**. Here you can modify how a particular features act as a fallback feature on the nonoverlapping parts, using the **Allowed region** list with the options **Fallback and nonpair regions** (the default), **Nonpair region**, and **All regions**. For the first and default option, the feature has the fallback behavior described above. Select the **Nonpair region** option if you do not want the feature to act as a fallback feature at all. All parts of the feature's selection that overlaps with any pair feature will then be excluded from the feature's selection. Choose the **All regions** option if you want the feature to contribute to the entire selection, including the parts where the pair is in contact. The feature's conditions will then contribute also with pair features conditions.

See The Fracture Boundary Condition in the Theory for the Elastic Waves, Time Explicit Interface for details.

Material Discontinuity

The **Material Discontinuity** node adds an interior boundary condition to handle jumps in material properties. As the interface is based on the discontinuous Galerkin method, special handling is necessary when a jump exists in the material properties. Add this node from the **Interior Conditions** submenu.

The condition implements continuity in velocity and normal stress over the interior boundary:

$$\mathbf{v}_{\text{down}} - \mathbf{v}_{\text{up}} = \mathbf{0}$$
$$(\mathbf{S}_{\text{down}} - \mathbf{S}_{\text{up}})\mathbf{n} = \mathbf{0}$$

Subscripts “up” and “down” in the equation refer to the up and down sides of the interior boundary, relative to the normal $\mathbf{n}$.

The Piezoelectric Waves, Time Explicit Interface

The **Piezoelectric Waves, Time Explicit** multiphysics interface ( ), found under the **Acoustics** > **Elastic Waves** branch ( ) when adding a physics interface. This interface combines The Elastic Waves, Time Explicit Interface and The Electrostatics Interface together with a dedicated multiphysics coupling and material models. These add the constitutive relationships required to model piezoelectric phenomena in the time domain for wave propagation. Both the direct and inverse piezoelectric effects can be modeled and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms.

The interface is based on the discontinuous Galerkin (dG or dG-FEM) method and uses a time explicit solver. The electrostatics problem is solved at every time step through an algebraic system of equations solved with classical FEM. The piezoelectric problem is solved fully coupled. The method is very memory efficient and can solve problems with many million degrees of freedom (DOFs). The method is also well suited for distributed computing on clusters.

When adding the **Piezoelectric Waves, Time Explicit** multiphysics interface a dedicated time explicit version of the Piezoelectric Effect multiphysics coupling, called Piezoelectricity, Time Explicit, is added. The feature is used to couple the Piezoelectric Material node in Elastic Waves, Time Explicit to the Charge Conservation, Piezoelectric node in Electrostatics.

	When setting up models using an *Assembly* geometry and the *Pair* multiphysics couplings it is recommended to always use imprints for assemblies. This will both increase performance and stability.

	• *Angle Beam Nondestructive Testing*. Application Library path **Acoustics_Module/Ultrasound/angle_beam_ndt** • *Ultrasonic Flowmeter with Piezoelectric Transducers*. Application Library path **Acoustics_Module/Ultrasound/flow_meter_piezoelectric_transducers**

Piezoelectric Material

The **Piezoelectric Material** node defines the piezoelectric material properties either in stress-charge form using the elasticity matrix and the coupling matrix, or in strain-charge form using the compliance matrix and the coupling matrix. It is normally used together with a Piezoelectricity, Time Explicit multiphysics coupling node and a corresponding **Charge Conservation, Piezoelectric** node in the **Electrostatics** interface. This node is added by default to the **Elastic Waves, Time Explicit** interface when adding a **Piezoelectric Waves, Time Explicit** multiphysics interface.

This material model requires the Structural Mechanics Module, MEMS Module, or Acoustics Module.

When the **Piezoelectric Material** node is added to the **Elastic Waves, Time Explicit** interface in the absence of an active **Piezoelectric Effect, Time Explicit** multiphysics coupling node, the material behaves similarly to an Elastic Waves, Time Explicit Model node. The elastic properties correspond to the elasticity or compliance matrix entered (see below). The piezoelectric effect is then not included in the equation system.

COORDINATE SYSTEM SELECTION

Select the **Coordinate system** that defines the polarization orientation of the piezoelectric crystal. The default is the **Global coordinate system**. The polarization is defined in the local x-axis direction in 2D, while it uses the local z-axis in 2D axisymmetry and 3D.

For entering the material properties matrices (see below), use the following order (Voigt notation), which is the common convention for piezoelectric materials: *xx*, *yy*, *zz*, *yz*, *xz*, *zy*.

PIEZOELECTRIC MATERIAL PROPERTIES

Select a **Constitutive relation** as **Stress-charge form** (the default) or **Strain-charge form**. For each of the following, the default uses values **From material**. For **User defined** enter other values in the matrix or field as needed.

- For **Stress-charge form**, select an **Elasticity matrix, Voigt notation** (c_{E}).
- For a **Strain-charge form**, select a **Compliance matrix, Voigt notation** (s_{E}).
- Select a **Coupling matrix, Voigt notation** (e_{ES} or d_{ET}).

- Select a **Relative permittivity** (ε_{rS} or ε_{rT}).
- Enter values for the **Remanent electric displacement** ($\mathbf{D}_r$).
- Select a **Density** (ρ).

LAX-FRIEDRICH FLUX PARAMETER

See Lax–Friedrichs Flux Parameter.

FILTER PARAMETERS

See Filter Parameters.

ESTIMATE OF MAXIMUM WAVE SPEED

See Estimate of Maximum Wave Speed.

Mechanical Damping

The **Mechanical Damping** is a subnode to the Piezoelectric Material used to define the mechanical damping properties for the parent piezoelectric material (the elastic part). See the Damping node in The Elastic Waves, Time Explicit Interface.

Modeling with the Elastic Waves Branch

In this section:

- Meshing and Solving Wave Problems Solved with Solid Mechanics
- Meshing Poroelastic Waves Models
- Solving Large Poroelastic Wave Models
- Meshing and Solving Elastic Waves, Time Explicit Models
- Absorbing Layers in Elastic Waves, Time Explicit
- Computing the Displacement in the Elastic Waves, Time Explicit

Meshing and Solving Wave Problems Solved with Solid Mechanics

In solids, several waves propagate depending on the configuration of the model. The wave with the shortest wavelength (the slowest wave) needs to be resolved with 5 to 6 mesh elements. In general, geometry details also need to be resolved by the mesh. For thin structures use at least two mesh elements in the thickness.

In the bulk, the pressure wave is typically the fastest wave and the shear wave is the slowest wave. The wave speeds can be evaluated through the variables `solid.cp` and `solid.cs`. They can be evaluated before solving a model fully, by using the **Get Initial Value** on the study, and then plot the variables or evaluate them in a point. If a free surface or a material discontinuity is present in the model, interfacial waves will also propagate. For example, the classical estimate for the Rayleigh wave speed v_R is

$$\frac{v_R}{c_s} \approx \frac{0{,}87 + 1{,}12\nu}{1+\nu}$$

where ν is Poisson's ratio and c_s is shear wave speed.

When solving elastic wave problems with the **Solid Mechanics** interface in the time domain, it is important to make changes to the **Transient Solver Setting** section (top interface node). Expand the section and select the checkbox **Maximum frequency to resolve**, then enter the maximum frequency to resolve in the model $f_{\mathrm{max,sol}}$ (remember to reset the solver if changes are made to this section). The maximum frequency is dictated by the frequency content of the source as well as the eigenmodes of the

structure that can be excited. Setting this correctly will results in a transient solver tuned specifically for modeling wave problems.

The correct setup of the solver (specifying the maximum frequency), when using the Solid Mechanics interface to model transient wave problems, ensures adequate resolution in time and space. The transient solver is based on the time implicit method. This is in contrast to the time explicit based physics interface, where the internal time step needs to ensure stability of the method.

Meshing Poroelastic Waves Models

Three different waves can propagate in a poroelastic material: fast pressure waves, slow pressure waves, and shear waves. Their speed can be evaluated through the variables `pelw.cp_fast`, `pelw.cp_slow`, and `pelw.cs_poro`, respectively. These should not be confused with the material properties for the constituting solid (`pelw.cp` and `pwel.cs`) and the fluid in the pores (`pelw.c`). The wave speed values can be evaluated before solving a model fully, by using the **Get Initial Value** on the study. Then plot the variables or evaluate them in a point.

All three poroelastic wave types should be resolved by the computational mesh. This means that the minimum mesh size is given by the wavelength of the slowest wave (the shortest wavelength) divided by 5 to 6. The mesh should of course also resolve geometry details such as highly curved boundaries and narrow regions. In general a mesh convergence study is always recommended.

Solving Large Poroelastic Wave Models

Large poroelastic waves models and especially large multiphysics models, that involve poroelastic waves physics, can be solved using an iterative solver (see the application library example below). For iterative approaches, tight tolerances are recommended. The preconditioner can be either the multigrid (geometric multigrid or GMG) solving for both pressure and displacement (as in the example). An alternative is to use hybridization and a direct preconditioner for the pressure DOFs and another direct

preconditioner for the displacement DOFs. Both should use PARDISO as the direct solver.

Headphone on an Artificial Ear: Application Library path
Acoustics_Module/Electroacoustic_Transducers/headphone_artificial_ear

Meshing and Solving Elastic Waves, Time Explicit Models

Meshing and solution time are closely linked when modeling physics based on the discontinuous Galerkin (dG) time explicit method. The computational mesh has to resolve the shortest wavelength in the model (the slowest wave), while it is the fastest wave speed and the smallest mesh element that dictate the internal time step of the solver.

In general, the maximal mesh size $h_{\max}$ is dictated by the smallest wavelength (the slowest wave) such that

$$h_{\max} = \frac{c_{\min}}{1.5 \cdot f_{\max}} \text{ to } \frac{c_{\min}}{2 \cdot f_{\max}}$$

for the default 4th (quartic) order discretization, where $f_{\max}$ is the maximal frequency to resolve in the model, given by the frequency content of the source, and $c_{\min}$ is the slowest waves speed in the model. This is typically the shear wave speed c_s, but for problems with a free interface or a material discontinuities, interfacial waves also exist. For example, the classical estimate for the Rayleigh wave speed v_R is

$$\frac{v_R}{c_s} \approx \frac{0{,}87 + 1{,}12\nu}{1 + \nu}$$

where ν is Poisson's ratio and c_s is shear wave speed.

Internally, the time step taken by the time explicit method is given by the global minimum of the local mesh size relative to the maximal local wave speed. This is also known as the *cell wave time scale*, the value can be visualized by plotting the variable `elte.wtc`. This means that small mesh elements should be avoided, see Meshing, Discretization, and Solvers for the *Convected Wave Equation* documentation, for more details.

The maximal wave speed used to predict the cell wave time scale uses the pressure wave speed c_p as default. The value is exact for isotropic materials but is an estimate for

materials with anisotropic properties. For anisotropic materials it can be necessary to use a larger value to ensure stability of the numerical method. This can be done in the **Estimate of Maximum Wave Speed** section for each material model. To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. Select **Automatic** (the default) or **User defined**. With the **Automatic** option the maximum speed is set equal to the pressure wave speed, for **User defined** enter another value for $c_{\max}$ (a larger value may be needed for stability).

When several materials are used in a model, the use of assemblies and pair features is recommended (Continuity and Pair Acoustic–Structure Boundary, Time Explicit). The advantage of using a pair feature is that the mesh does not need to be conforming on the two sides of the interface (the two parts pf the assembly). This is especially advantageous for the time explicit discontinuous Galerkin method as the time step depends on mesh size and local speed of sound.

If material properties result in large variations of the local *cell wave time scale*, then it can be advantageous to use the **Adam-Bashforth 3 (local)** method instead of the default **Runge-Kutta** method.

PHYSICS-CONTROLLED MESH

The Elastic Waves, Time Explicit interface supports physics-controlled mesh. To get an appropriate mesh, remember to set the value of the **Maximum frequency to resolve** in the Transient Mesh Settings section. The automatically generated mesh will use this value and an estimate of the slowest wave speed (the default being the shear wave speed), defined in the model, to set up an appropriate mesh that follows the best practices.

On the main **Mesh** node select the **Sequence Type** to be **Physics-controlled mesh** and then select the physics interface in the **Contributor** list. The **Maximum mesh element size control parameter** is either **From physics** (it will use the value from the Transient Mesh Settings section) or **Frequency**. For **Frequency** enter a value for the **Maximum frequency**. Select the **Number of mesh elements per wavelength**; for **Automatic** this is 1.5 elements per wavelength, or enter a **User defined** value. Finally, select the **Minimum wave speed** used to define the shortest wavelength; for **Automatic** the shear wave speed is used, for **User defined** enter another value, for example, a bending wave speed or a Rayleigh wave speed, if necessary.

Absorbing Layers in Elastic Waves, Time Explicit

The Elastic Waves, Time Explicit Interface includes *absorbing layers,* also often referred to as *sponge layers*. The layers work by combining three techniques: a scaling system, filtering, and a low-reflecting condition.

The layers are set up by adding the **Absorbing Layer** under the **Definitions** node. This adds a special scaled system. The scaling effectively slows down the propagating waves and ensures that they "align up" (normal) toward the outer boundary. This means that the waves will hit the outer boundary in a closer to normal direction. Filtering attenuates and filters out high frequency components of the wave. The filter parameters are controlled under the Filter Parameters for Absorbing Layers section at the main physics level. Finally, at the outer boundary of the layer add a Low-Reflecting Boundary condition, which will work well to remove all remaining waves, as normal incidence has been ensured.

For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

The physical thickness of the layer should be such that it contains 2 to 3 mesh elements. This in practice means that it should roughly have a thickness equal to the largest wavelength resolved in the model. You want to avoid to create small mesh elements in the absorbing layers. These can potentially slow down the solution procedure since the internal time stepping is controlled by the mesh size.

For the **Absorbing Layers** select the **Type** (**Cartesian**, **Cylindrical**, **Spherical**, or **User defined**) under the **Geometry** section. Enter values for the **Physical Width** and **Pole Distance** under the **Scaling** section.

For more details on the **Geometry** and **Scaling** see the Infinite Elements, Perfectly Matched Layers, and Absorbing Layers section in the *COMSOL Multiphysics Reference Manual*.

For the layers to work optimally, the filter should not be too aggressive. Moreover, the scaled coordinates in the layer domain should also vary smoothly. To inspect the scaled system you can, for example, plot the coordinate variables `x_absorb_ab1`, `y_absorb_ab1`, and `z_absorb_ab1`. In 2D axisymmetry, they are `r_absorb_ab1` and

`z_absorb_ab1`. Note that the variables are not in the plot menu under the **Replace Expression**.

Using the absorbing layers with the three combined techniques will enable the reduction of spurious reflections by a factor between 100 and 1000 compared to the incident amplitude.

- *Isotropic-Anisotropic Sample: Elastic Wave Propagation*. Application Library path **Acoustics_Module/Elastic_Waves/isotropic_anisotropic_sample**
- *Ground Motion After Seismic Event: Scattering off a Small Mountain*. Application Library path **Acoustics_Module/Elastic_Waves/ground_motion_seismic_event**

Computing the Displacement in the Elastic Waves, Time Explicit

The **Elastic Waves, Time Explicit** interface solves for the velocity $\mathbf{v}$ and the strain ε. It is sometimes useful to also compute the displacement $\mathbf{u}$. This comes at an additional computational cost as an additional equation needs to be solved. The cost depends on where the displacement evaluation is required, in a point, on an edge, on a boundary, or in a domain. In all cases the simple ODE related the displacement and the velocity:

$$\frac{d\mathbf{u}}{dt} = \mathbf{v}$$

This computation is predefined in the Compute Displacement features that can be added as a subnode to the Elastic Waves, Time Explicit Model node.

For an example where this feature is used in a point see the tutorial *Isotropic-Anisotropic Sample: Elastic Wave Propagation*. Application Library path **Acoustics_Module/Elastic_Waves/isotropic_anisotropic_sample**

Theory for the Poroelastic Waves Interfaces

The Poroelastic Waves Interface theory is described in this section:

- Elastic Waves Introduction
- Poroelastic Waves Theory
- Anisotropic Poroelastic Material Theory
- Boundary Conditions for Poroelastic Waves
- Postprocessing Variables
- References for the Poroelastic Waves Interfaces

Elastic Waves Introduction

The most general linear relation (more details are found in the Structural Mechanics Theory section of the *Structural Mechanics Module User's Guide*) between the stress and strain tensors in solid materials can be written as

$$\sigma_{ij} = \mathbf{c}_{ijkl}\varepsilon_{kl}$$

here, σ is the Cauchy's stress tensor, ε is the strain tensor, and $\mathbf{c}_{ijkl}$ is a fourth-order elasticity tensor. For small deformations, the strain tensor is defined as

$$\varepsilon(\mathbf{u}) = \frac{1}{2}(\nabla\mathbf{u} + \nabla\mathbf{u}^T)$$

where $\mathbf{u}$ represents the displacement vector.

The elastic wave equation is then obtained from Newton's second law

$$\rho\frac{\partial^2}{\partial t^2}\mathbf{u} - \nabla\cdot(\sigma(\mathbf{u}) - \mathbf{s}_0) = \mathbf{F}$$

here, ρ is the medium density, and $\mathbf{s}_0$ and $\mathbf{F}$ represent source terms.

An important case is the time-harmonic wave, for which the displacement varies with time as

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

with f (SI unit: Hz) denoting the frequency and $\omega = 2\pi f$ (SI unit: rad/s) the angular frequency. Assuming the same time-harmonic dependency for the source terms $\mathbf{s}_0$ and $\mathbf{F}$, the wave equation for linear elastic waves reduces to an inhomogeneous Helmholtz equation:

$$-\rho\omega^2\mathbf{u} - \nabla \cdot (\sigma(\mathbf{u}) - \mathbf{s}_0) = \mathbf{F} \tag{3-1}$$

Alternatively, treat this equation as an eigenvalue PDE to solve for eigenmodes and eigenfrequencies as described in the *Structural Mechanics Module User's Guide* in the Structural Mechanics Modeling chapter under Eigenfrequency Analysis. Also add damping as described in Mechanical Damping and Losses.

Poroelastic Waves Theory

In his seminal work, Biot extended the classical theory of linear elasticity to porous media saturated with fluids (Ref. 1, Ref. 2, and Ref. 3).

In Biot's theory, the bulk moduli and compressibilities are independent of the wave frequency, and can be treated as constant parameters. The porous matrix is described by linear elasticity and damping is introduced by considering the viscosity of the fluid in the pores, which can be frequency dependent. This description is adequate for the propagation of poroelastic waves in soils and rocks where the saturating fluid is a liquid, like oil or water. This formulation is referred to as the *Biot model* (this is in some sense the classical formulation).

- High Frequency Correction (Biot Model)

When the considered porous material is saturated by a gas, like air, thermal losses need to be included in order to properly model its behavior. This is the case when modeling sound absorbers, car cabin liners, or foams used in headsets or loudspeakers. The formulation of the equations where both the thermal and viscous losses are included is sometimes referred to as the *Biot–Allard model*. In this case, both the viscosity and the fluid compressibility are considered to be frequency dependent and complex valued (Ref. 7, Ref. 8, Ref. 9, Ref. 10, and Ref. 11).

- Biot–Allard Model (Viscous and Thermal Losses)

Consider Biot's expressions for poroelastic waves (Ref. 3, Ref. 4, and Ref. 6)

$$\rho_{av}\frac{\partial^2}{\partial t^2}\mathbf{u} + \rho_f\frac{\partial^2}{\partial t^2}\mathbf{w} - \nabla\cdot\sigma = 0$$
$$\rho_f\frac{\partial^2}{\partial t^2}\mathbf{u} + \frac{\mu_f}{\kappa}\frac{\partial}{\partial t}\mathbf{w} + \frac{\tau}{\varepsilon_p}\rho_f\frac{\partial^2}{\partial t^2}\mathbf{w} + \nabla p_f = 0 \quad (3\text{-}2)$$

Here, $\mathbf{u}$ is the displacement of the porous material, σ is the total stress tensor (fluid and porous material), $\mathbf{w}$ is the fluid displacement with respect to the porous matrix, ρ_f and μ_f are the fluid's density and viscosity, τ is the tortuosity, ε_p is the porosity, p_f is the fluid pore pressure, κ is the permeability and ρ_{av} the average density. The average density is the total density (porous material plus pore fluid) $\rho_{av} = \rho_d + \varepsilon_p\rho_f$.

Assuming a time-harmonic dependency for the variables, $\mathbf{u}(\mathbf{x},t) = \mathbf{u}(\mathbf{x})e^{i\omega t}$, $\mathbf{w}(\mathbf{x},t) = \mathbf{w}(\mathbf{x})e^{i\omega t}$, the time derivatives can be removed, so the system in Equation 3-2 becomes

$$-\rho_{av}\omega^2\mathbf{u} + \rho_f\omega^2\mathbf{w} - \nabla\cdot\sigma = 0$$
$$-\rho_f\omega^2\mathbf{u} - \omega^2\rho_c(\omega)\mathbf{w} + \nabla p_f = 0 \quad (3\text{-}3)$$

Here, the complex density $\rho_c(\omega)$ (Ref. 5) accounts for the tortuosity, porosity, and fluid density, and the viscous drag on the porous matrix

$$\rho_c(\omega) = \frac{\tau}{\varepsilon_p}\rho_f + \frac{\mu_f}{i\omega\kappa} \quad (3\text{-}4)$$

HIGH FREQUENCY CORRECTION (BIOT MODEL)

At low frequencies or for small pore sizes, the flow profile inside the pores can be assumed to be Poiseuille like. In this case, the viscosity in Equation 3-4 effectively has a constant value. For increasing frequency, the profile changes and a frequency-dependent correction factor needs to be taken into account. This is done by selecting the **Biot's high frequency range** option from the **Viscosity model** list. In this case Equation 3-4 is implemented with a frequency-dependent viscosity $\mu_c(f)$ (Ref. 2, Ref. 3, Ref. 5)

$$\mu_c(f) = \mu_f \cdot F_c\left(\sqrt{\frac{f}{f_r}}\right)$$

Here, f_r is a reference frequency (SI unit: Hz), which determines the low-frequency range $f << f_r$ and the high-frequency range $f >> f_r$.

The reference frequency f_r can be interpreted as the limit when viscous forces equal inertial forces in the fluid motion. In a pore with characteristic size a, this happens when the viscous penetration depth is equal to the pore radius.

$$\delta_v = \sqrt{\frac{\mu_f}{2\pi f_r \rho_f}} = a \quad \Rightarrow \quad f_r = \frac{\mu_f}{2\pi a^2 \rho_f}$$

In the low-frequency limit, viscous effects dominate, while in the high-frequency limit, inertial effects dominate fluid motion in the pores (losses occur in the viscous boundary layer). In Biot's low frequency range, $\omega \to 0$ and $F_c = 1$.

In order to account for a frequency dependence on the viscous drag, Biot defined the operator $F_c(\Theta)$ as

$$F_c(\Theta) = \frac{1}{4}\left(\frac{\Theta T(\Theta)}{1 + 2iT(\Theta)/\Theta}\right)$$

where, $T(\Theta)$ is related to the Kelvin functions $\mathrm{Ber}(\Theta)$ and $\mathrm{Bei}(\Theta)$

$$T(\Theta) = \frac{\mathrm{Ber}'(\Theta) + i\mathrm{Bei}'(\Theta)}{\mathrm{Ber}(\Theta) + i\mathrm{Bei}(\Theta)} = \frac{-\sqrt{-i}J_1(\sqrt{-i}\Theta)}{J_0(\sqrt{-i}\Theta)}$$

and J_0 and J_1 are Bessel functions of the first kind. This expression can be recognized as the loss terms in Zwikker–Kosten like equivalent fluid models (Derivation of the Equivalent Bulk Modulus Valid for Any Fluid in the Zwikker–Kosten Theory) or the loss models for cylindrical waveguides in the narrow region acoustics or LRF models (About the Narrow Region Acoustics Models) models.

U-P FORMULATION

The formulation in terms of the displacements $\mathbf{u}$ and $\mathbf{w}$ is not optimal from the numerical viewpoint, since it requires to solve for two displacement fields (Ref. 7, Ref. 8, Ref. 9). The Poroelastic Waves interface solves for the fluid pore pressure variable p_f instead of the fluid displacement field $\mathbf{w}$.

The second row in Equation 3-3 is simplified to

$$\mathbf{w} = \frac{1}{\omega^2 \rho_c(\omega)}(\nabla p_f - \rho_f \omega^2 \mathbf{u})$$

so the first row in Equation 3-3 becomes

$$-\rho_{av}\omega^2\mathbf{u}-\frac{\rho_f}{\rho_c(\omega)}(\nabla p_f-\rho_f\omega^2\mathbf{u})-\nabla\cdot\sigma = 0 \tag{3-5}$$

The total stress tensor σ is then divided into the contributions from the elastic porous (drained) matrix and from the pore fluid

$$\sigma(\mathbf{u}, p_f) = \sigma_d(\mathbf{u})-\alpha_B p_f\mathbf{I}$$

Here, the identity tensor $\mathbf{I}$ means that the pore pressure p_f only contributes to the diagonal of the total stress tensor σ. The parameter α_B is the so-called Biot–Willis coefficient. The drained, elastic stress tensor is written as $\sigma_d = \mathbf{c}:\varepsilon$ when ε is the strain tensor of the porous matrix, and the elasticity tensor $\mathbf{c}$ contains the drained porous matrix's elastic properties (see the Linear Elastic Material feature in the *Structural Mechanics Module User's Guide*).

Finally, arrange Equation 3-5 in terms of the variables $\mathbf{u}$ and p:

$$-\left(\rho_{av}-\frac{\rho_f^2}{\rho_c(\omega)}\right)\omega^2\mathbf{u}-\nabla\cdot(\sigma_d(\mathbf{u})-\alpha_B p_f\mathbf{I}) = \frac{\rho_f}{\rho_c(\omega)}\nabla p_f \tag{3-6}$$

The next Biot's equation comes from taking the divergence of the second row in Equation 3-3, previously divided by $-\rho_c(\omega)$

$$\omega^2\nabla\cdot\left(\frac{\rho_f}{\rho_c(\omega)}\mathbf{u}\right)+\omega^2\nabla\cdot\mathbf{w}+\nabla\cdot\left(-\frac{1}{\rho_c(\omega)}\right)\nabla p_f = 0 \tag{3-7}$$

Using the expressions for the volumetric strain $\varepsilon_{vol} = \nabla\cdot\mathbf{u}$ and fluid displacement (Ref. 3, Ref. 4),

$$-\nabla\cdot\mathbf{w} = \frac{p_f}{M}+\alpha_B\varepsilon_{vol}$$

Biot's modulus M is calculated from the porosity ε_p, fluid compressibility χ_f, Biot–Willis coefficient α_B and the drained bulk modulus of the porous matrix K_d

$$\frac{1}{M} = \varepsilon_p\chi_f+\frac{\alpha_B-\varepsilon_p}{K_d}(1-\alpha_B) \tag{3-8}$$

so Equation 3-7 simplifies to

$$\omega^2\nabla\cdot\left(\frac{\rho_f}{\rho_c(\omega)}\mathbf{u}\right)-\omega^2\left(\frac{1}{M}p_f+\alpha_B\varepsilon_{vol}\right)+\nabla\cdot\left(-\frac{1}{\rho_c(\omega)}\right)\nabla p_f = 0 \tag{3-9}$$

and Biot's wave equations (Equation 3-6 and Equation 3-9) can be written in terms of the variable $\mathbf{u}$ and p_f as

$$\begin{aligned} -\omega^2\left(\rho_{av}-\frac{\rho_f^2}{\rho_c(\omega)}\right)\mathbf{u}-\nabla\cdot(\sigma_d(\mathbf{u})-\alpha_B p_f\mathbf{I}) &= \frac{\rho_f}{\rho_c(\omega)}\nabla p_f \\ -\frac{\omega^2}{M}p_f+\nabla\cdot-\frac{1}{\rho_c(\omega)}(\nabla p_f-\omega^2\rho_f\mathbf{u}) &= \omega^2\alpha_B\varepsilon_{vol} \end{aligned} \tag{3-10}$$

The saturated (also called Gassmann) modulus can be obtained from the drained bulk modulus K_d, Biot modulus M, and Biot–Willis coefficient α_B as $K_{sat}=K_d+\alpha_B{}^2M$ (Ref. 5).

Further arranging the first row in Equation 3-10 to fit the formulation in the Elastic Waves interface (Equation 3-1) gives

$$-\omega^2\left(\rho_{av}-\frac{\rho_f^2}{\rho_c(\omega)}\right)\mathbf{u}-\nabla\cdot(\sigma_d(\mathbf{u})-\mathbf{s}_0)=\mathbf{F} \tag{3-11}$$

The body load $\mathbf{F}$ depends on the angular frequency and the gradient of fluid pressure and the fluid pressure acts as a spherical contribution to the diagonal of Cauchy stress tensor

$$\mathbf{F}=\frac{\rho_f}{\rho_c(\omega)}\nabla p_f$$
$$\mathbf{s}_0=\alpha_B p_f\mathbf{I}$$

Arranging the second row in Equation 3-10 to fit the implementation of the Pressure Acoustics, Frequency Domain interface gives (see Theory Background for the Pressure Acoustics Branch)

$$-\frac{\omega^2}{M}p_f+\nabla\cdot-\left(\frac{1}{\rho_c(\omega)}(\nabla p_f-\mathbf{q}_d)\right)=Q_m \tag{3-12}$$

The monopole domain source Q_m (SI unit: $1/s^2$) and the dipole domain source $\mathbf{q}_d$ (SI unit: N/m^3) depend on the angular frequency ω, the displacement of the porous matrix $\mathbf{u}$, the fluid density and Biot–Willis coefficient α_B

$$Q_m = \omega^2 \alpha_B \varepsilon_{vol}$$
$$\mathbf{q}_d = \omega^2 \rho_f \mathbf{u}$$

BIOT–ALLARD MODEL (VISCOUS AND THERMAL LOSSES)

When both thermal and viscous losses are included, the viscosity in Equation 3-4 and the fluid compressibility in Equation 3-8 are replaced by frequency-dependent expressions. The losses due to viscosity are considered by the viscosity expression and the losses due to thermal conduction by the fluid compressibility expression, see Ref. 9.

The frequency-dependent complex viscosity is given by

$$\mu(\omega) = \mu\left(1 + \frac{4i\omega\tau_\infty^2\mu\rho_f}{R_f^2 L_v^2 \varepsilon_p^{\ 2}}\right)^{\frac{1}{2}}$$

where the viscous characteristic length L_v has been introduced (it is sometimes referred to as Λ). The frequency-dependent complex fluid compressibility is given by

$$\chi_f(\omega) = \frac{1}{\gamma p_A}\left[\gamma - (\gamma - 1)\left(1 + \frac{8\mu}{i\omega L_{th}^2 \mathrm{Pr}\rho_f}\sqrt{1 + \frac{i\omega L_{th}^2 \mathrm{Pr}\rho_f}{16\mu}}\right)^{-1}\right]$$

where the thermal characteristic length L_{th} has been introduced (it is sometimes referred to as Λ'). The two expressions can be recognized as the JCA equivalent fluid model (Johnson–Champoux–Allard (JCA)) available in Pressure Acoustics.

Different loss models or formulations for the frequency-dependent viscosity and fluid compressibility can be entered manually. In order to do so, select the **Biot (viscous losses)** model and then set the fluid **Compressibility** and the fluid **Dynamic viscosity** to **User defined**. In these two fields, enter the desired model expression. It can, for example, depend on the frequency, using the variable `freq`.

Anisotropic Poroelastic Material Theory

When modeling many porous fibrous materials and others, several porous material properties (as well as the elastic properties of the drained porous matrix) will be

anisotropic. The effects of an anisotropic flow resistivity (or permeability), anisotropic tortuosity, and an anisotropic viscous characteristic length, are introduced through an anisotropic acoustics model that has an anisotropic (complex valued) density. This is much like the Anisotropic Acoustics model in The Pressure Acoustics, Frequency Domain Interface. Several formulations exist and can, for example, be found in Ref. 9, 12, 13, 14, 15, and 16.

In the formulation used in the **Anisotropic Porous Material,** the anisotropic effects of the porous properties are assumed to be oriented in the same direction. This assumption is true or a good assumption for most porous materials. In this way, the material data is always defined by a diagonal tensor. In the model, the (complex-valued) anisotropic density becomes:

$$\rho_{ii} = \frac{[\tau_\infty]_{ii}\rho_f}{\varepsilon_p} + \frac{[\tilde{\mu}_f]_{ii}}{i\omega[\kappa_p]_{ii}} \tag{3-13}$$

where the quantities in brackets are now tensor valued. The fluid compressibility remains isotropic as it does not depend on any of the porous properties that can exhibit anisotropic behavior.

Boundary Conditions for Poroelastic Waves

Although boundary conditions can be set up for the porous matrix and fluid independently of each other, there exist a few common boundary conditions which deserve special attention. The following sections refer to the boundary conditions for the system written in Equation 3-11 and Equation 3-12. See derivation in Ref. 7, Ref. 8, and Ref. 9.

POROUS, FREE (SOUND-SOFT BOUNDARY) THEORY

The Free, Sound Soft boundary condition defines a condition where the displacement of the porous matrix in Equation 3-11 is unconstrained, so it can move freely without experiencing any loads.

The *sound soft* boundary condition for acoustics creates a boundary condition for Equation 3-12 where the acoustic pressure vanishes, so it sets $p_f = 0$.

FIXED (SOUND-HARD BOUNDARY) EQUATIONS

For simulating a poroelastic medium bounded by a rigid impervious wall, impose a Fixed Constraint node for the porous matrix displacement in Equation 3-11, $\mathbf{u} = 0$, and a *sound-hard* boundary condition for the pore pressure in Equation 3-12:

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

PRESSURE EQUATIONS

For a given fluid pressure p_0 on the boundary, set the pressure in Equation 3-12 to $p_f = p_0$. Since the fluid pressure is set to p_0, the normal stress on the porous matrix in Equation 3-11 reduces to

$$\mathbf{n} \cdot \sigma_d(\mathbf{u}) = \mathbf{n}(\alpha_B - 1)p_0$$

For a rigid porous matrix $\alpha_B = \varepsilon_p$, the load is equivalent to

$$\mathbf{n} \cdot \sigma_d(\mathbf{u}) = \mathbf{n}(\varepsilon_p - 1)p_0$$

and for a soft porous matrix $\alpha_B = 1$, there is no load since

$$\mathbf{n} \cdot \sigma_d(\mathbf{u}) = \mathbf{0}$$

PRESCRIBED DISPLACEMENT EQUATIONS

For a prescribed displacement $\mathbf{u}_0$ at the boundary, set the displacement of the porous matrix in Equation 3-11 as $\mathbf{u} = \mathbf{u}_0$ and assume a sound-hard (impervious) boundary for the fluid pressure in Equation 3-12:

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

PRESCRIBED VELOCITY EQUATIONS

For a prescribed velocity $\mathbf{v}_0$ at the boundary, set the displacement of the porous matrix in Equation 3-11 as

$$\mathbf{u} = \frac{1}{i\omega}\mathbf{v}_0$$

and assume a sound-hard (impervious) boundary condition for the fluid pressure in Equation 3-12

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

PRESCRIBED ACCELERATION EQUATIONS

For a prescribed acceleration $\mathbf{a}_0$ at the boundary, set the displacement of the porous matrix in Equation 3-11 as

$$\mathbf{u} = \frac{1}{-\omega^2}\mathbf{a}_0$$

and assume a sound-hard (impervious) boundary condition for the fluid pressure in Equation 3-12

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

ROLLER EQUATIONS

The roller, or sliding wall boundary, means that the boundary is impervious (sound-hard) to fluid displacements, but it allows tangential displacements of the porous matrix.

The normal displacement of the porous matrix in Equation 3-11 is constrained, but the porous matrix is free to move in the tangential direction

$$\mathbf{n} \cdot \mathbf{u} = 0$$

The impervious boundary condition for the fluid pressure in Equation 3-12 is obtained from

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

SEPTUM BOUNDARY LOAD EQUATIONS

For a prescribed load $\mathbf{F}_A$ at the boundary, suppose that one side of the septum is fixed to the porous matrix and the other side bears the load.

A septum is a very limp and thin impervious layer with surface density ρ_{sep}. Since the septum can be seen as a boundary mass density, this boundary condition is achieved by setting an effective load $\mathbf{F}_S = \mathbf{F}_A + \rho_{sep}\omega^2\mathbf{u}$ on the porous matrix, so the normal stress in Equation 3-11 reduces to

$$\mathbf{n} \cdot (\sigma_d(\mathbf{u}) - \alpha_B p_f \mathbf{I}) = \mathbf{F}_S$$

and a sound-hard (impervious) boundary condition is applied for the fluid pressure in Equation 3-12

$$\mathbf{n} \cdot \left(\frac{1}{\rho_c(\omega)} (\nabla p_f - \rho_f \omega^2 \mathbf{u}) \right) = 0$$

To set up this condition combine the Boundary Load and the Impervious Mass Layer features.

Postprocessing Variables

Other well known formulations of the poroelastic waves equations exist which use other parameters and variables in their definition (see Ref. 10). To help users that are familiar with these formulations, some of the most common parameters have been added as postprocessing variables under the *Biot–Allard coefficients* menu (when clicking the **Replace Expressions** button in plots). The variables included are:

TABLE 3-4: BIOT–ALLARD COEFFICIENTS.

COMSOL NAME	DEFINITIONS	NAME
pelw.rho11	$\rho_{11} = \rho_d - \rho_{12}$	Mass coefficient (11)
pelw.rho22	$\rho_{22} = \varepsilon_p \rho_f - \rho_{12}$	Mass coefficient (22)
pelw.rho12	$\rho_{12} = -\rho_a = -\varepsilon_p \rho_f (\tau_\infty - 1)$	Inertial interaction mass coefficient (12)
pelw.rho11_c	$\tilde{\rho}_{11} = \rho_{11} + \tilde{b}(\omega)/(i\omega)$	Complex mass coefficient (11)
pelw.rho22_c	$\tilde{\rho}_{22} = \rho_{22} + \tilde{b}(\omega)/(i\omega)$	Complex mass coefficient (11)
pelw.rho12_c	$\tilde{\rho}_{12} = \rho_{12} - \tilde{b}(\omega)/(i\omega)$	Complex inertial interaction mass coefficient (12)
pelw.A_c	See Ref. 10	Complex elastic coefficient
pelw.Q_c	See Ref. 10	Complex elastic coupling coefficient
pelw.R_c	See Ref. 10	Complex bulk modulus coefficient
pelw.gamma_c	See Ref. 10	Diagonal stress coupling coefficient

References for the Poroelastic Waves Interfaces

1. M.A. Biot, “Theory of Propagation of Elastic Waves in a Fluid-saturated Porous Solid. I. Low-frequency Range”, *J. Acoust. Soc. Am.*, vol. 28, pp 168–178, 1956.

2. M.A. Biot, "Theory of Propagation of Elastic Waves in a Fluid-saturated Porous Solid. II. Higher Frequency Range", *J. Acoust. Soc. Am.*, vol. 28, pp 179–191, 1956.

3. M.A. Biot, "Generalized Theory of Acoustic Propagation in Porous Dissipative Media", *J. Acoust. Soc. Am.*, vol. 34, pp 1254–1264, 1962.

4. M.A. Biot, "Mechanics of Deformation and Acoustic Propagation in Porous Media", *J. Appl. Phys.*, vol. 33, pp 1482–1498, 1962.

5. G. Mavko and others, *The Rock Physics Handbook*, 2nd ed., Cambridge University Press, 2009.

6. J.M. Carcione, *Wave Fields in Real Media: Wave Propagation in Anisotropic, Anelastic, Porous and Electromagnetic Media*, 2nd ed. Elsevier (*Handbook of Geophysical Exploration*, vol. 38, Seismic Exploration), 2007.

7. P. Debergue, R. Panneton, and N. Atalla, "Boundary Conditions for the Weak Formulation of the Mixed (u,p) Poroelasticity Problem", *J. Acoust. Soc. Am.*, vol. 106, pp 2383–2390, 1999.

8. N. Atalla, M.A. Hamdi, and R. Panneton, "Enhanced Weak Integral Formulation for the Mixed (u,p) Poroelastic Equations", *J. Acoust. Soc. Am.*, vol. 109, pp 3065–3068, 2001.

9. J.F. Allard and N. Atalla, *Propagation of Sound in Porous Media*, 2nd ed., John Wiley & Sons, 2009.

10. N. Atalla, R. Panneton, and P. Debergue, "A mixed displacement–pressure formulation for poroelastic materials", *J. Acoust. Soc. Am.*, vol. 104, pp 1444, 1998.

11. N. Atalla, F. Sgard, and C.K. Amedin, "On the modeling of sound radiation from poroelastic materials", *J. Acoust. Soc. Am.*, vol. 120, pp 1990, 2006.

12. N.-E. Hörlin and P. Göransson, "Weal, anisotropic symmetric formulation of Biot's equations for vibro-acoustic modelling of porous elastic materials", *Int. J. Numer. Meth. Engng.*, vol. 84, pp. 1519–1540, 2010.

13. B. P. Semeniuk and P. Göransson, "Microstructure based estimation of the dynamic drag impedance of lightweight fibrous materials", *J. Acoust. Soc. Am.*, vol. 141, pp 1360, 2017.

14. B. P. Semeniuk, P. Göransson, and O. Dazel, "Dynamic equations of a transversely isotropic, highly porous, fibrous material including oscillatory heat transfer effects", *J. Acoust. Soc. Am.*, vol. 146, pp 2540-2551, 2019.

15. B. P. Semeniuk, E. Lundberg, and P. Göransson, "Acoustics modelling of open-cell foam materials from microstructure and constitutive properties", *J. Acoust. Soc. Am.*, vol. 149, pp 2016–2026, 2021.

16. B. Nenning, R. Binois, N. Dauchez, E. Perrey-Debain, and F. Foucart, "A transverse isotropic equivalent fluid model combining both limp and rigid frame behaviors for fibrous materials", *J. Acoust. Soc. Am.*, vol. 143, pp 2089–2098, 2018.

Theory for the Elastic Waves, Time Explicit Interface

The Elastic Waves, Time Explicit Interface theory is described in this section:

- Governing Equations
- The Fracture Boundary Condition
- References for the Elastic Waves, Time Explicit Interface

Governing Equations

The model solves the governing equations for a general linear elastic material in a velocity-strain formulation

$$\rho\frac{\partial \mathbf{v}}{\partial t} - \nabla \cdot \mathbf{S} = \mathbf{F}_v$$

$$\frac{\partial \varepsilon}{\partial t} - \frac{1}{2}[\nabla \mathbf{v} + (\nabla \mathbf{v})^T] = \mathbf{0}$$

$$\mathbf{S} = \mathbf{C}:\varepsilon$$

where $\mathbf{v}$ is the velocity, ρ the density, $\mathbf{S}$ the stress tensor, ε the strain tensor, $\mathbf{C}$ is the elasticity tensor (or stiffness tensor), and $\mathbf{F}_v$ is a possible body force. The equations are valid for both isotropic and anisotropic material data.

The unknowns solved for are three components of the velocity vector and six components of the strain tensor in 3D. The out-of-plane components of the velocity and the strain are not computed per default in 2D. This can be changed by selecting **Include out-of-plane components** checkbox. It can be relevant in some cases, for example for anisotropic materials, where the out-of-plane velocity may become different from zero for in-plane loads. This formulation is often referred to as 2.5D.

The Fracture Boundary Condition

The **Fracture** boundary condition relies on the linear slip interface theory proposed by Schoenberg in Ref. 2. The linear slip condition implements the concept of an

imperfectly bonded interface between two elastic domains. Imperfect bonding means that:

- The traction is continuous across the interface
- The displacement is discontinuous across the interface

The traction at the interface is expressed as a function of the displacement jump, which up to the first order terms reads

$$\mathbf{Sn} = -\mathbf{k}_{\mathrm{A}}(\mathbf{u}_{\mathrm{down}} - \mathbf{u}_{\mathrm{up}}),$$

where $\mathbf{k}_{\mathrm{A}}$ is the positive definite symmetric boundary stiffness matrix which has dimensions stress/length. For the velocity, which is the dependent variable in the velocity-strain formulation, the linear slip interface boundary condition reads

$$\frac{\partial \mathbf{S}}{\partial t}\mathbf{n} = -\mathbf{k}_{\mathrm{A}}(\mathbf{v}_{\mathrm{down}} - \mathbf{v}_{\mathrm{up}}).$$

A fracture can represent a thin elastic layer, a fluid filled crack, or a discontinuity (open gap or free surface) in the elastic material. The free surface is approached when $\mathbf{k}_{\mathrm{A}} = 0$. For the fluid filled crack, the tangential stiffness should vanish $\mathbf{k}_{T} = 0$.

References for the Elastic Waves, Time Explicit Interface

1. J.S. Hesthaven and T. Warburton, *Nodal Discontinuous Galerkin Methods: Algorithms, Analysis, and Applications*, Springer, 2008.

2. M. Schoenberg, "Elastic wave behavior across linear slip interfaces," *J. Acoust. Soc. Am.*, vol. 65, issue 5, 1980.

4

Acoustic–Structure Interaction Interfaces

This chapter describes the physics interfaces found under the **Acoustic–Structure Interaction** branch ( ).

- The Acoustic–Solid Interaction, Frequency Domain Interface
- The Acoustic–Solid Interaction, Transient Interface
- The Acoustic–Shell Interaction, Frequency Domain Interface
- The Acoustic–Shell Interaction, Transient Interface
- The Acoustic–Piezoelectric Interaction, Frequency Domain Interface
- The Acoustic–Piezoelectric Interaction, Transient Interface
- The Acoustic–Poroelastic Waves Interaction Interface
- The Acoustic–Solid–Poroelastic Waves Interaction Interface
- The Acoustic–Solid Interaction, Time Explicit Interface
- Modeling with the Acoustic–Structure Interaction Branch

The Acoustic–Solid Interaction, Frequency Domain Interface

The **Acoustic–Solid Interaction, Frequency Domain** multiphysics interface ( ), found under the **Acoustics** > **Acoustic–Structure Interaction** branch ( ) when adding a physics interface, combines the Pressure Acoustics, Frequency Domain and Solid Mechanics interfaces to connect the acoustics pressure variations in the fluid domain with the structural deformation in the solid domain. It can, for example, be used to determine the transmission of sound through an elastic structure or solve for the coupled vibroacoustics phenomena present in a loudspeaker.

Acoustic–structure interaction or ASI refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid domain, and the structural acceleration acts on the fluid domain as a normal acceleration across the fluid-solid boundary.

A dedicated multiphysics coupling condition is readily defined for the fluid-solid boundary and sets up the fluid loads on the solid domain and the effect of the structural accelerations on the fluid.

The physics interface is available for 3D, 2D, and 2D axisymmetric geometries and has the capability to model pressure acoustics and solid mechanics in the frequency domain, including a special acoustic-solid boundary condition for the fluid-solid interaction.

When a predefined **Acoustic–Solid Interaction, Frequency Domain** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain** and **Solid Mechanics** interfaces are added to the Model Builder.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the **Acoustic–Structure Boundary** multiphysics coupling.

See The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When a predefined multiphysics interface is used, for example **Acoustic–Solid Interaction, Frequency Domain**, the **Selection** on the multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Frequency Domain** on one side and **Solid Mechanics** on the other.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics couplings, but you need to manually select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Coupling

The Acoustic–Structure Boundary multiphysics coupling is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface.
- The available physics features for The Solid Mechanics Interface are listed in the *Structural Mechanics Module User's Guide*.

- Theory section in the *Structural Mechanics Module User's Guide*
- Theory Background for the Pressure Acoustics Branch

Acoustic–Structure Interaction: Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/acoustic_structure**

Suggestions for setting up the solvers for solving large acoustic–structure interaction problems are given in Solving Large Acoustic–Structure Interaction Models in the Modeling with the Acoustic–Structure Interaction Branch section.

The Acoustic–Solid Interaction, Transient Interface

The **Acoustic–Solid Interaction, Transient** multiphysics interface ( ), found under the **Acoustics** > **Acoustic–Structure Interaction** branch ( ) when adding a physics interface, combines the Pressure Acoustics, Transient and Solid Mechanics interfaces to connect the acoustics pressure variations in the fluid domain with the structural deformation in the solid domain. It can, for example, be used to determine the transmission of sound through an elastic structure or solve for the coupled vibroacoustics phenomena present in a loudspeaker.

Acoustic–structure interaction or ASI refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid domain, and the structural acceleration acts on the fluid domain as a normal acceleration across the fluid-solid boundary.

Special physics interface conditions are readily defined at the fluid-solid boundary and set up the fluid loads on the solid domain and the effect of the structural accelerations on the fluid. The physics interface allows for transient studies but also solves in the frequency domain with the available boundary conditions.

When a predefined **Acoustic–Solid Interaction, Transient** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Transient** and **Solid Mechanics** interfaces are added to the Model Builder.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the **Acoustic–Structure Boundary** multiphysics coupling.

See The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual.*

For modeling of acoustic–structure interaction in the frequency domain, The Acoustic–Solid Interaction, Frequency Domain Interface contains additional functionality that is not applicable for modeling in the time domain.

- Theory section in the *Structural Mechanics Module User's Guide*
- Theory Background for the Pressure Acoustics Branch

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When the predefined multiphysics interface is used, the **Selection** on the multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Transient** on one side and **Solid Mechanics** on the other.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics couplings, but you need to manually select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Coupling

The Acoustic–Structure Boundary multiphysics coupling is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Pressure Acoustics, Transient Interface are listed in the section Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface.
- The available physics features for The Solid Mechanics Interface are listed in the *Structural Mechanics Module User's Guide*.

- Theory section in the *Structural Mechanics User's Guide*
- Theory Background for the Pressure Acoustics Branch

The Acoustic–Piezoelectric Interaction, Frequency Domain Interface

The **Acoustic–Piezoelectric Interaction, Frequency Domain** multiphysics interface ( ), found when adding a physics interface under the **Acoustics** > **Acoustic–Structure Interaction** branch ( ), combines the Pressure Acoustics, Frequency Domain, Solid Mechanics, and Electrostatics interfaces to connect and solve for the acoustic pressure variations in fluids with the structural deformation in both solids and piezoelectric solid domains. The physics interface also includes features from Electrostatics to solve for the electric field in the piezoelectric material. It may be used for modeling piezoelectric transducers for sonar or medical applications and, for example, enhancing the impedance matching layers as well as the far-field radiation patterns of the transducer.

The Helmholtz equation is solved in the fluid domain and the structural equations in the solid together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled, and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms.

When a predefined **Acoustic–Piezoelectric Interaction, Frequency Domain** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch ( ) of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain**, **Solid Mechanics**, and **Electrostatics** interfaces are added to the Model Builder.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the **Acoustic–Structure Boundary** and **Piezoelectricity** multiphysics couplings.

See The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

The equations solved in the solid and fluid domains can be found in the *Structural Mechanics Module User's Guide* and in Theory Background for the Pressure Acoustics Branch, respectively.

Piezoacoustic Transducer: Application Library path **Acoustics_Module/Piezoelectric_Devices/piezoacoustic_transducer**

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When the predefined multiphysics interface is used, the **Selection** on the **Acoustic–Structure Boundary** multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Frequency Domain** on one side and **Solid Mechanics** on the other. Similarly, the **Selection** on the **Piezoelectricity** multiphysics coupling is automatically set to **All domains** and active in all valid domains.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics coupling, but you need to manually select on which boundaries or domains they need to be applied, or select **All boundaries** or **All domains** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Couplings

The Acoustic–Structure Boundary multiphysics coupling is described in the Multiphysics Couplings chapter. The Piezoelectricity multiphysics coupling is described for The Piezoelectricity Interface.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface.
- The available physics features for The Solid Mechanics Interface that are described in the *Structural Mechanics Module User's Guide*.
- The available physics features for The Electrostatics Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Electrostatics Interface in the *COMSOL Multiphysics Reference Manual*.
- Some additional piezoelectric features for the Solid Mechanics and Electrostatics interfaces are described for The Piezoelectricity Interface.

- Theory section in the *Structural Mechanics Module User's Guide*
- Theory of Electrostatics in the *COMSOL Multiphysics Reference Manual*
- Theory Background for the Pressure Acoustics Branch

The Acoustic–Piezoelectric Interaction, Transient Interface

The **Acoustic–Piezoelectric Interaction, Transient** multiphysics interface ( ), found when adding a physics interface under the **Acoustics** > **Acoustic–Structure Interaction** branch ( ), combines the Pressure Acoustics, Transient and Piezoelectricity interfaces to connect and solve for the acoustic pressure variations in fluids with the structural deformation in both solids and piezoelectric solid domains. The physics interface also includes features from Electrostatics to solve for the electric field in the piezoelectric material. Examples include modeling piezoelectric transducers for sonar or medical applications and, for example, enhancing the impedance matching layers.

The pressure wave equation is solved in the fluid domain and the structural dynamic equations in the solid together with the constitutive relationships required to model piezoelectrics. Both the direct and inverse piezoelectric effects can be modeled, and the piezoelectric coupling can be formulated using the strain-charge or stress-charge forms.

When a predefined **Acoustic–Piezoelectric Interaction, Transient** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain**, **Solid Mechanics**, and **Electrostatics** interfaces are added to the Model Builder.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the **Acoustic–Structure Boundary** and **Piezoelectricity** multiphysics couplings.

See The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When the predefined multiphysics interface is used, the **Selection** on the **Acoustic–Structure Boundary** multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Frequency Domain** on one side and **Solid Mechanics** on the other. Similarly, the **Selection** on the **Piezoelectricity** multiphysics coupling is automatically set to **All domains** and active in all valid domains.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics couplings, but you need to manually select on which boundaries or domains they need to be applied, or select **All boundaries** or **All domains** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Couplings

The Acoustic–Structure Boundary multiphysics coupling is described in the Multiphysics Couplings chapter. The Piezoelectricity multiphysics coupling is shortly described in The Piezoelectricity Interface.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Pressure Acoustics, Transient Interface are listed in the section Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface.
- The available physics features for The Solid Mechanics Interface are listed in the *Structural Mechanics Module User's Guide*.

- The available physics features for The Electrostatics Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Electrostatics Interface in the *COMSOL Multiphysics Reference Manual*.
- Some additional piezoelectric features for the Solid Mechanics and Electrostatics interfaces are described for The Piezoelectricity Interface.

- Theory section in the *Structural Mechanics Module User's Guide*
- Theory of Electrostatics in the *COMSOL Multiphysics Reference Manual*
- Theory Background for the Pressure Acoustics Branch

The Acoustic–Poroelastic Waves Interaction Interface

The **Acoustic–Poroelastic Waves Interaction** multiphysics interface ( ) combines Pressure Acoustics, Frequency Domain and Poroelastic Waves together with the Acoustic–Porous Boundary multiphysics coupling.

When a predefined **Acoustic–Poroelastic Waves Interaction** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch ( ) of the **Model Wizard** or the **Add Physics** windows, the **Poroelastic Waves** and **Pressure Acoustics, Frequency Domain** interfaces are added to the Model Builder.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the **Acoustic–Porous Boundary** multiphysics coupling.

See The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When the predefined multiphysics interface is used, for example **Acoustic–Poroelastic Waves Interaction**, the **Selection** on the multiphysics coupling is automatically set to **All boundaries**. In this way the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Frequency Domain** on one side and **Poroelastic Waves** on the other.

However, if physics interfaces are added one at a time, followed by the multiphysics coupling, these modified settings are not automatically included.

For example, if single **Poroelastic Waves** and **Pressure Acoustics, Frequency Domain** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics couplings, but you need manually to

select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior

	Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

	Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Coupling

The Acoustic–Porous Boundary multiphysics coupling is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Poroelastic Waves Interface are listed in the section Domain, Boundary, and Pair Nodes for the Poroelastic Waves Interfaces.
- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface.

The Acoustic–Solid–Poroelastic Waves Interaction Interface

The **Acoustic–Solid–Poroelastic Waves Interaction** multiphysics interface ( ) combines Pressure Acoustics, Frequency Domain, Solid Mechanics, and Poroelastic Waves together with the Acoustic–Structure Boundary, Porous-Structure Boundary, and Acoustic-Porous Boundary multiphysics couplings.

When a predefined **Acoustic–Solid–Poroelastic Waves Interaction** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch ( ) of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain**, **Solid Mechanics**, and **Poroelastic Waves** interfaces are added to the Model Builder.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the **Acoustic–Porous Boundary**, **Porous–Structure Boundary**, and **Acoustic–Structure Boundary** multiphysics couplings.

See The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual.*

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When a predefined multiphysics interface is used, for example **Acoustic–Solid–Poroelastic Waves Interaction**, the **Selection** on the multiphysics couplings is automatically set to **All boundaries**. In this way, the relevant multiphysics couplings are automatically active on all relevant boundaries connecting two physics.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if single **Poroelastic Waves** and **Pressure Acoustics, Frequency Domain** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics couplings, but you need to manually

select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Couplings

The Acoustic–Porous Boundary, Porous–Structure Boundary, and Acoustic–Structure Boundary multiphysics couplings are described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Poroelastic Waves Interface are listed in the section Domain, Boundary, and Pair Nodes for the Poroelastic Waves Interfaces.
- The available physics features for The Solid Mechanics Interface are listed in the *Structural Mechanics Module User's Guide*.
- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface.

The Acoustic–Solid Interaction, Time Explicit Interface

The **Acoustic–Solid Interaction, Time Explicit** multiphysics interface ( ) combines Pressure Acoustics, Time Explicit and Elastic Waves, Time Explicit together with the Acoustic–Structure Boundary, Time Explicit multiphysics coupling. The physics interface is available for 3D and 2D geometries.

Acoustic–structure interaction or ASI refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid domain, and the structural acceleration acts on the fluid domain as a normal acceleration across the fluid-solid boundary. The Acoustic–Solid Interaction, Time Explicit interface can, for example, be used to determine the transmission of sound through an elastic structure or solve for the coupled vibroacoustics phenomena present in an ultrasonic transducer or flowmeter.

The discontinuous Galerkin time explicit method enables efficient computation of acoustically large models (containing many wavelengths) in the time domain. The formulation of the governing equations, effectively requires the acoustics pressure and velocity variations in the fluid domain to be coupled with the structural velocity and strain variations in the solid domain.

A dedicated multiphysics coupling condition is readily defined for the fluid-solid boundary and sets up the continuity in velocity and normal stress.

When a predefined **Acoustic–Solid Interaction, Time Explicit** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Time Explicit** and **Elastic Waves, Time Explicit** interfaces are added to the Model Builder. In addition, the **Multiphysics Couplings** node is added, which automatically includes the **Acoustic–Structure Boundary, Time Explicit** multiphysics coupling.

See The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual.*

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When a predefined multiphysics interface is used, for example **Acoustic–Structure Boundary, Time Explicit**, the **Selection** on the multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all boundaries with **Pressure Acoustics, Time Explicit** on one side and **Elastic Waves, Time Explicit** on the other.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics couplings, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Couplings

The Acoustic–Structure Boundary, Time Explicit multiphysics coupling is described in the Multiphysics Couplings chapter. Note that a pair version of the multiphysics coupling exists that can be used for assemblies: Pair Acoustic–Structure Boundary, Time Explicit.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Pressure Acoustics, Time Explicit Interface are listed in the section Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Time Explicit Interface.
- The available physics features for The Elastic Waves, Time Explicit Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Elastic Waves, Time Explicit Interface.

- Theory Background for the Pressure Acoustics Branch
- Theory for the Elastic Waves, Time Explicit Interface

The Acoustic–Shell Interaction, Frequency Domain Interface

This physics interface requires a Structural Mechanics Module license. For theory and physics interface feature descriptions relating to the Shell interface, see the *Structural Mechanics Module User's Guide*.

The **Acoustic–Shell Interaction, Frequency Domain** multiphysics interface ( ), found under the **Acoustics** > **Acoustic–Structure Interaction** branch ( ) when adding a physics interface, combines features from the Pressure Acoustics, Frequency Domain and Shell interfaces to connect the acoustics pressure variations in the fluid domain with the structural deformation of a shell boundary. It may for example be used for determining the transmission of sound through a thin elastic structure such as a car hood and analyzing the vibroacoustics of a loudspeaker cone.

Acoustic–structure interaction or ASI refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid surface, and the structural acceleration acts on the fluid domain as a normal acceleration across the fluid-structure boundary.

Special physics interface conditions are readily defined at the fluid-shell boundary and set up the fluid loads on the shell boundary and the effect of the structural accelerations on the fluid. The physics interface is available for 3D and 2D axisymmetric geometries, and it is capable of modeling the coupled pressure acoustics and shell vibrations in the frequency domain.

When a predefined **Acoustic–Shell Interaction, Frequency Domain** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Frequency Domain** and **Shell** interfaces are added to the Model Builder.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the multiphysics coupling **Acoustic–Structure Boundary**.

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When a predefined multiphysics interface is used, for example **Acoustic–Shell Interaction, Frequency Domain**, the **Selection** on the multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is

automatically active on all **Shell** boundaries with **Pressure Acoustics, Frequency Domain** on one side or both sides.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if single **Shell** and **Pressure Acoustics, Frequency Domain** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics couplings, but you need to manually select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Coupling

The Acoustic–Structure Boundary multiphysics coupling is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface.

- The available physics features for The Shell and Plate Interfaces are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*.

	• Theory Background for the Pressure Acoustics Branch • Theory for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*

The Acoustic–Shell Interaction, Transient Interface

This physics interface requires a Structural Mechanics license. For theory and physics interface feature descriptions relating to the Shell interface, see the *Structural Mechanics Module User's Guide*.

The **Acoustic–Shell Interaction, Transient** multiphysics interface ( ), found under the **Acoustics** > **Acoustic–Structure Interaction** branch ( ) when adding a physics interface, combines features from the Pressure Acoustics, Transient and Shell interfaces to connect the acoustics pressure variations in the fluid domain with the structural deformation of a shell boundary. It may, for example, be used for determining the transmission of sound through a thin elastic structure such as a car hood and analyzing the vibroacoustics of loudspeaker cone.

Acoustic–structure interaction refers to a multiphysics phenomenon where the acoustic pressure causes a fluid load on the solid surface, and the structural acceleration affects the fluid domain as a normal acceleration across the fluid-structure boundary.

Special physics interface conditions are readily defined at the fluid-shell boundary and set up the fluid loads on the shell boundary and the effect of the structural accelerations on the fluid. The physics interface is only available for 3D geometries, and it is capable of modeling the coupled pressure acoustics and shell vibrations in the time domain.

When a predefined **Acoustic–Shell Interaction, Transient** interface is added from the **Acoustics** > **Acoustic–Structure Interaction** branch of the **Model Wizard** or the **Add Physics** windows, the **Pressure Acoustics, Transient** and **Shell** interfaces are added to the Model Builder.

In addition, the **Multiphysics Couplings** node is added, which automatically includes the **Acoustic–Structure Boundary** multiphysics coupling.

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When a predefined multiphysics interface is used, for example **Acoustic–Shell Interaction, Transient**, the **Selection** on the multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all **Shell** boundaries with **Pressure Acoustics, Transient** on one side or both sides.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if single **Shell** and **Pressure Acoustics, Transient** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available multiphysics couplings, but you need manually to select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics Couplings** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Coupling

The Acoustic–Structure Boundary multiphysics coupling is described in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Pressure Acoustics, Transient Interface are listed in the section Domain, Boundary, Edge, and Point Nodes for the Pressure Acoustics, Transient Interface.
- The available physics features for The Shell and Plate Interfaces are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*.

	• Theory Background for the Pressure Acoustics Branch • Theory for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*

Modeling with the Acoustic–Structure Interaction Branch

In this section:

- Prestressed Acoustic–Structure Interaction
- Solving Large Acoustic–Structure Interaction Models
- Configuration of Perfectly Matched Layers (PMLs) for Acoustic–Structure Interaction Models

Prestressed Acoustic–Structure Interaction

When modeling transducers like microphones or ultrasound horns there are often parts of the structure, like the diaphragm or a bolt, that are under tension or prestressed. In these cases, it is essential to include this effect when modeling the transducer. A prestressed part will shift the resonance frequency of the mechanical system and thus the overall vibroacoustic behavior will change.

- *Piezoelectric Tonpilz Transducer with a Prestressed Bolt*: Application Library path **Acoustics_Module/Piezoelectric_Devices/tonpilz_transducer_prestressed**.
- *The Brüel & Kjær 4134 Condenser Microphone*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/bk_4134_microphone**.

Such prestressed acoustic–structure interaction models can be set up and solved fully coupled including all effects. The procedure is as follows:

1. Set up the model including all relevant physics.
2. Couple the physics using the required multiphysics couplings.
3. All acoustic sources, structural loads, and any forcing that are nonstatic, should be defined using the `linper()` operator. This will ensure that they are used only in the frequency-domain part of the study (that one is set to linear perturbation).
4. Add a **Prestressed Analysis, Frequency Domain** study.

5 Note that in the **Stationary** step the acoustics physics interfaces have an orange warning triangle under the **Physics and Variables Selection**. This simply means that they will not be used in the stationary study since acoustics is not supporting the study type. Therefore the **Solve for this field** will automatically not be marked under the **Dependent Variables** in the **Solver Configurations**.

6 Note also that the **Include geometric nonlinearity** checkbox is selected in the **Frequency-Domain, Perturbation** step. If it is not checked, the prestress effect is lost.

7 Solve the model for the desired frequencies.

A prestressed type of analysis can also be done on pure structural problems in the frequency domain and when searching for eigenfrequencies of structures.

- Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis in the *COMSOL Multiphysics Reference Manual.*
- Prestressed Structures, Modeling Pretensioned Bolts, and Mechanical Damping and Losses in the *Structural Mechanics Module User's Guide*

Solving Large Acoustic–Structure Interaction Models

In models that involve acoustic–structure interaction, the strategy for solving large problems involves solving the system in a segregated way. That is, the system is not solved fully coupled in one step but iterations are used solving one physics interface at a time.

To set up such a solver right-click on the Stationary Solver step and select **Segregated**. In the first **Segregated Step** solve for the structural dependent variables (displacement). Set up a second segregated step where you select the pressure dependent variable. Under each of the steps select the solver of choice for solving the single physics interface problem. For example, in the case where only a small structural domain is included, use an iterative multigrid approach for the acoustics (see Solving Large Acoustics Problems Using Iterative Solvers) and a direct solver for the structure.

This strategy is only readily applicable when the coupling between the solid and the acoustic domain is done via Neumann conditions; this is the case for all models where the acoustic domain uses Pressure Acoustics. In, for example, models with thermoviscous acoustic–structure interaction, the coupling is based on a Dirichlet condition (a pointwise constraint) and required reformulating the continuity condition using weak constraints. Nor does this approach work for models coupling

piezoelectric domains, structures, and acoustics; here a fully coupled approach is necessary.

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

Configuration of Perfectly Matched Layers (PMLs) for Acoustic–Structure Interaction Models

Care should be taken when setting up perfectly matched layers (PMLs) in models that include fluid domains and solid domains or in general, models with several different materials, that being different solids or different fluids.

Two configurations exist for the PMLs in these systems:

- The PML regions of two physics (or two different material models) are adjacent and in contact. In this case, a single PML feature should be used for the two PML domains. Set the **Typical wavelength from** to **User defined** and enter a value based on the longest wavelength. For example, for an air-steel system enter: `5770[m/s]/freq`

  The shorter wavelengths should be captured by increasing the mesh resolution or increasing the **PML scaling curvature parameter**. This will ensure optimal damping of all wavelengths in the system. In any case, control the convergence of the results by increasing the number of mesh layers in the PML.

	• Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual*. • See Perfectly Matched Layers (PMLs) in Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces) in the *Acoustics Module User's Guide*.

- The PML regions of two physics (or two different material models) do not touch. In this case, use one PML feature for each PML domain. Keep the default setting of **Typical wavelength from** to **Physics interface** and select the relevant physics interface under **Physics**.

The reason that two PML features cannot be used when the PML domains are adjacent to each other is that the scaling inside the PMLs will differ (because of different speeds

of sound). This will lead to an unphysical situation at the interface between the two PMLs which results in errors in the solution. Spurious waves may be generated.

See the *Acoustic–Structure Interaction with a Perfectly Matched Layer (PML)* model in the Application Gallery on the COMSOL website. It is located here: https://www.comsol.com/model/acoustic-structure-interaction-with-a-perfectly-matched-layer-pml-23521

5

Aeroacoustics Interfaces

This chapter describes the physics interfaces found under the **Aeroacoustics** branch .

The aeroacoustic branch has physics interfaces to solve the fully linearized acoustic equation for several physical conditions, they are: the *linearized Euler* (LE) equations, the *linearized Navier–Stokes* (LNS) equations, and the *linearized potential flow* (LPF) equations. These equations are also referred to as *convected acoustics*. Flow-induced noise functionality using the Lighthill analogy exists in pressure acoustics when using the Aeroacoustic Flow Source feature, associated multiphysics coupling, and dedicated study.

In this chapter:

- The Linearized Euler, Frequency Domain Interface
- The Linearized Euler, Transient Interface
- The Linearized Euler, Boundary Mode Interface
- The Linearized Navier–Stokes, Frequency Domain Interface
- The Linearized Navier–Stokes, Transient Interface
- The Linearized Navier–Stokes, Boundary Mode Interface
- The Linearized Potential Flow, Frequency Domain Interface

- The Linearized Potential Flow, Transient Interface
- The Linearized Potential Flow, Boundary Mode Interface
- The Compressible Potential Flow Interface
- Modeling with the Aeroacoustics Branch
- Theory Background for the Aeroacoustics Branch
- References for the Aeroacoustics Branch Interfaces

The Linearized Euler, Frequency Domain Interface

The **Linearized Euler, Frequency Domain (lef)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in density, velocity, and pressure in the presence of a stationary background mean-flow that is well approximated by an ideal gas flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized Euler equations.

The equations defined by the Linearized Euler, Frequency Domain interface are the linearized continuity, momentum (Euler), and energy equations. The physics interface solves for the acoustic variations in the density ρ, velocity field $\mathbf{u}$, and pressure p. The equations are formulated in the frequency domain and assume harmonic variation of all sources and fields. The harmonic variation of all fields and sources is given by $e^{i\omega t}$ using the $+i\omega$ convention. The background mean flow can be any stationary gas flow that is well approximated by an ideal gas. The coupling between the acoustic field and the background flow does not include any predefined flow-induced noise. Even though the equations do not include any loss mechanisms, only acoustic modes exist in the frequency domain as the driving frequency is predefined and real valued.

Coupling between a background mean flow, computed from a Fluid Flow model, and the Linearized Euler model is handled by the Background Fluid Flow Coupling multiphysics coupling and the dedicated Mapping study. Details are also found in the Mapping Between Fluid Flow and Acoustics Mesh section.

	For modeling tips and tricks and good practice, see the Modeling with the Aeroacoustics Branch section.

	Flow-induced noise functionality exists in pressure acoustics when using the Aeroacoustic Flow Source feature and associated multiphysics coupling and dedicated study.

The equations are implemented in the so-called scattered field formulation. All equations and boundary conditions are formulated in the total acoustic fields

(ρ_t, $\mathbf{u}_t$, p_t). The total fields are in the presence of the Background Acoustic Fields feature the sum of the background (ρ_b, $\mathbf{u}_b$, p_b) and the scattered field (ρ, $\mathbf{u}$, p):

$$\rho_t = \rho + \rho_b \qquad \mathbf{u}_t = \mathbf{u} + \mathbf{u}_b \qquad p_t = p + p_b$$

The scattered field variables are the variables solved for, that is, the dependent variables. When no background acoustic field is present, the total field is simply equal to the scattered field

$$\rho_t = \rho \qquad \mathbf{u}_t = \mathbf{u} \qquad p_t = p$$

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Euler Model**, **Rigid Wall**, and **Initial Values**. For axisymmetric components an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Euler, Frequency Domain** to select physics features from the context menu.

SETTINGS

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

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

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

LINEARIZED EULER EQUATION SETTINGS

In a 2D Axisymmetric component, select **Out-of-plane mode extension** to add the azimuthal component to the system of equations solved. When selected enter the **Azimuthal mode number** m (sometimes referred to as the circumferential mode number). The latter defines an azimuthal wave number $k_m = m/r$. All out-of-plane gradients of the fields are then defined in terms of the wave number. When selected the Domain Sources, Background Acoustic Fields, and Prescribed Acoustic Fields features will have an additional input for the azimuthal component of the source.

SOUND PRESSURE LEVEL SETTINGS

See Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface. Only **Use reference pressure for air** or **User-defined reference pressure** are available selections.

TYPICAL WAVE SPEED

Enter a value or expression for the **Typical wave speed for perfectly matched layers** c_{ref} (SI unit m/s). The default is `lef.c0` and the value is automatically taken from the material model. If several materials or material models are used, it is best practice to add one PML for each. This will ensure that the typical wavelength is continuous within each PML feature.

For convected acoustic problems, it can be necessary to set the value of the **Typical wavelength from** option in the PML feature manually, to correct for the convected background velocity.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields): the **Density** `rho`, **Velocity field** `u` and its components, and **Pressure** `p`. The name can be changed but the names of fields and dependent variables must be unique within a model.

STABILIZATION

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog.

Select the **Stabilization Method** — **No stabilization applied**, **Galerkin least squares (GLS) stabilization** (the default), **Streamline upwind Petrov-Galerkin (SUPG) stabilization**, or **Streamline diffusion (legacy method)**. When stabilization is selected enter a value for the **Stabilization constant** α_{stab} (dimensionless). The default value is `1e-5` and should typically have a numerical value between `1e-3` and `1e-7`. In cases where there is no background flow, set the value to the lower limit `1e-7`.

The default GLS stabilization is the most efficient stabilization method as it operates on the convective and reactive parts of the governing equations. This is also the default method and the method suggested for most applications. The stabilization constant

α_{stab} can be tuned depending on the problem solved, the nature of the background mean flow, and on the computational mesh.

- Stabilization in the Modeling with the Aeroacoustics Branch section.

If the stabilization is turned off (when **No stabilization applied** is selected), it is recommended to change the discretization (see section below) to ensure a stable numerical scheme; set the order of the pressure one order lower than the velocity and density dependent variables, for example, using a P2-P2-P1 discretization.

DISCRETIZATION

From the list select the element order and type (Lagrange or serendipity) for the density, velocity, and pressure degrees of freedom. The default is **Linear** for all the dependent variables.

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

- Domain, Boundary, and Pair Nodes for the Linearized Euler, Frequency Domain Interface
- Theory Background for the Aeroacoustics Branch

For a tutorial and benchmark model, solving the linearized Euler equations in both the frequency and the time domains, see the Application Gallery model: *Point Source in 2D Jet: Radiation and Refraction of Sound Waves Through a 2D Shear Layer*

https://www.comsol.com/model/point-source-in-2d-jet-radiation-and-refraction-of-sound-waves-through-a-2d-shear-layer-16685

The model requires the Acoustics Module and the CFD Module.

Domain, Boundary, and Pair Nodes for the Linearized Euler, Frequency Domain Interface

The Linearized Euler, Frequency Domain Interface has these domain, boundary, and pair nodes and subnodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- Asymptotic Far-Field Radiation
- Axial Symmetry
- Background Acoustic Fields
- Continuity
- Domain Sources
- Impedance
- Initial Values
- Interior Impedance
- Interior Wall
- Linearized Euler Model
- Moving Wall
- Outflow Boundary
- Periodic Condition
- Prescribed Acoustic Fields
- Pressure (Isentropic)
- Rigid Wall
- Sound Soft Boundary (Isentropic)
- Specific Acoustic Impedance (Isentropic)
- Symmetry

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Linearized Euler Model

Use the **Linearized Euler Model** to set up the governing equations, define the background mean flow, the ideal gas fluid properties, and select gradient term suppression stabilization (advanced physics option), if needed. The governing equations solved are (in the time domain):

$$\frac{\partial \rho_t}{\partial t} + \nabla \cdot (\rho_t \mathbf{u}_0 + \rho_0 \mathbf{u}_t) = S_c$$

$$\frac{\partial \mathbf{u}_t}{\partial t} + \left(\left[\mathbf{u}_t + \frac{\rho_t}{\rho_0}\mathbf{u}_0\right] \cdot \nabla\right)\mathbf{u}_0 + (\mathbf{u}_0 \cdot \nabla)\mathbf{u}_t + \frac{1}{\rho_0}\nabla p_t = \mathbf{S}_m \quad (5\text{-}1)$$

$$\frac{\partial p_t}{\partial t} + \mathbf{u}_t \cdot \nabla p_0 + p_t \gamma(\nabla \cdot \mathbf{u}_0) + \mathbf{u}_0 \cdot \nabla p_t + \gamma p_0(\nabla \cdot \mathbf{u}_t) = S_e$$

where ρ_t, $\mathbf{u}_t$, and p_t are the acoustic perturbations to the density, velocity, and pressure, respectively. The subscript "t" refers to the fact that the acoustic variables are the total fields, that is, the sum of possible Background Acoustic Fields and the scattered fields.

In the frequency domain, the time derivatives of the dependent variables are replaced by multiplication with $i\omega$. The variables with a zero subscript are the background mean flow values, γ is the ratio of specific heats. The right-hand-side source terms S_c, $\mathbf{S}_m$, and S_e are zero per default. They can be defined in the Domain Sources node. Details about the physics solved and references are found in the Theory Background for the Aeroacoustics Branch section.

MODEL INPUTS

In order to model the influence of the background mean flow on the propagation of the acoustic waves in the fluid, the background mean flow temperature T_0, absolute pressure p_0, and velocity field $\mathbf{u}_0$ need to be defined.

Select **User defined** (the default), **Common model input**, or a variable defined by a flow simulation performed using the CFD Module. By default, they are set to the quiescent background conditions of air. All the background flow parameters can also be constants or analytical expressions functions of space.

When modeling aeroacoustics, it is important how the Mapping Between Fluid Flow and Acoustics Mesh is done from a numerical perspective.

Details about the Model Input and the Default Model Inputs are found in the Global and Local Definitions chapter of the *COMSOL Multiphysics Reference Manual*.

Enter **User defined** values for the **Background mean flow temperature** T_0 (SI unit: K), **Background mean flow pressure** p_0 (SI unit: Pa), and **Background mean flow velocity** $\mathbf{u}_0$ (SI unit: m/s). The defaults are 293.15 K, 1 atm, and 0 m/s, respectively.

Note that the **Background mean flow density** also needs to be defined or entered in the **Fluid Properties** section below.

FLUID PROPERTIES

Select an option for the **Background mean flow density** ρ_0 (SI unit: kg/m^3) — **Ideal gas** (the default), **From material**, **User defined** (default value 1.2 kg/m^3), or it can be picked up from a flow interface, for example, from a High Mach Number Flow model as **Density (hmnf/fluid1)**. As the flow is assumed to be an ideal gas, the background density ρ_0 is readily defined as

$$\rho_0 = \frac{p_0}{R_s T_0}$$

where R_s is the specific gas constant.

Define the remaining fluid properties necessary. Select the **Gas constant type**: **Specific gas constant** (the default) or **Mean molar mass**. The defaults take values **From material** or for **User defined** enter another value or expression:

- **Specific gas constant** R_s (SI unit: J/(kg·K)). The default is 287.058 J/(kg·K)).
- **Mean molar mass** M_n (SI unit: g/mol). The default is 28.97 g/mol), which calculates $R_s = R/M_n$, where R is the gas constant.

Select an option from the **Specify Cp or** γ list: **Ratio of specific heats** (the default) or **Heat capacity at constant pressure**. The defaults take values **From material** or for **User defined** enter another value or expression:

- **Ratio of specific heats** γ (dimensionless). The default is 1.4.
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)). The default is 1005.4 J/(kg·K)), which calculates $\gamma = C_p/(C_p - R_s)$.

GRADIENT TERM SUPPRESSION STABILIZATION

When the linearized Euler (LE) equations are solved in the time domain (or in the frequency domain with an iterative solver), linear physical instability waves can develop, the so-called Kelvin-Helmholtz instabilities. They are instabilities that grow exponentially because no losses exist in the LE equations (no viscous dissipation and no heat conduction). They are also naturally limited by nonlinearities in the full Navier–Stokes flow equations. The terms responsible for the instabilities are typically

the reactive terms in the governing equations. It has been shown that in some problems the growth of these instabilities can be limited, while the acoustic solution is retained, by canceling terms involving gradients of the mean flow quantities. This is known as gradient terms suppression (GTS) stabilization.

More details are available in Ref. 9, Ref. 10, Ref. 11, and in the Theory Background for the Aeroacoustics Branch section.

All the aeroacoustic specific terms can be disabled. They are grouped into *Reactive terms* and *Convective terms*. Select the following checkboxes to activate the applicable gradient term suppression (GTS):

Reactive terms

- **Suppression of mean flow density gradients**

  This option sets the following reactive term in the continuity equation to zero:

$$\mathbf{u}_t \cdot \nabla \rho_0 = 0$$

- **Suppression of mean flow velocity gradients**

  This option sets the following reactive terms in the three governing equations to zero:

$$\rho_t(\nabla \cdot \mathbf{u}_0) = 0 \qquad \left(\left[\mathbf{u}_t + \frac{\rho_t}{\rho_0}\mathbf{u}_0\right] \cdot \nabla\right)\mathbf{u}_0 = \mathbf{0} \qquad \gamma p_t(\nabla \cdot \mathbf{u}_0) = 0$$

- **Suppression of mean flow pressure gradients**

  This option sets the following term in the energy equation to zero:

$$\mathbf{u}_t \cdot \nabla p_0 = 0$$

Convective terms

- **Suppression of all convective terms**

  This option removes all the convective terms in the governing equations. This is a drastic measure as it removes the convective influence of the flow. It should be considered carefully before doing this. This option removes all terms of the type:

$$(\mathbf{u}_0 \cdot \nabla)(\ldots) = 0$$

Rigid Wall

The **Rigid Wall** condition is used to model a rigid wall, corresponding to the sound hard wall condition in Pressure Acoustics. In the case of lossless flows, this reduces to the

slip condition $\mathbf{u}_t \cdot \mathbf{n} = 0$, where $\mathbf{n}$ is the surface normal. Add this node from the **Wall Conditions** submenu.

CONSTRAINT SETTINGS

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

On highly curved surfaces, it may be necessary to switch the **Constraint method** from the default **Elemental** to **Nodal** in order to avoid locking effects.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

For systems with a trailing edge on an airfoil or with an edge singularity between a liner and a rigid wall, the exclude edges and points functionality can be used to set up an appropriate Kutta condition. See Suppressing Constraints on Lower Dimensions.

Initial Values

The **Initial Values** node adds initial values for the density, the velocity field, and pressure dependent variables. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial values of the **Density** `rho` (SI unit: kg/m^3), the **Velocity field** **u** (SI unit: m/s), and the **Pressure** `p` (SI unit: Pa).

Axial Symmetry

In a 2D axisymmetric geometry, the **Axial Symmetry** condition is automatically added on the axis of symmetry at $r = 0$. On this boundary, the radial component of the total velocity is set equal to zero

$$u_t = 0 \qquad \mathbf{u}_t = (u_t, v_t, w_t)$$

Domain Sources

Add a **Domain Sources** node to define the mass source, and momentum and energy source types. This domain feature adds the right-hand side S_c, $\mathbf{S}_m$, and S_e to the governing Equation 5-1 defined through different options

$$S_c = M$$

$$\mathbf{S}_m = \mathbf{A} - \frac{\mathbf{u}_0 M}{\rho_0} \quad \text{or} \quad \mathbf{S}_m = \frac{1}{\rho_0}(\mathbf{F} - \mathbf{u}_0 M)$$

$$S_e = p_{rc} + \frac{\gamma p_0 M}{\rho_0} \quad \text{or} \quad S_e = (\gamma - 1)Q + \frac{\gamma p_0 M}{\rho_0}$$

where M is a mass source, $\mathbf{A}$ is an acceleration source, $\mathbf{F}$ is a volume force source, p_{rc} is a pressure rate of change source, and Q is a volumetric heat source. Notice that the mass source term enters both the momentum and the energy equations in a consistent way. The **Domain Sources** can be used to create any user defined source by entering expressions into the fields. The sources can for example be point-like Gaussian sources to model an idealized single vortex or two interacting vortices.

DOMAIN SOURCES

Enter a **Mass source** M (SI unit: kg/(m^3·s)).

Select a **Momentum source type**: **Acceleration** (the default) or **Volume force**. For **Acceleration** enter vector expressions or values for the **Acceleration source** $\mathbf{A}$ (SI unit: m/s^2). For **Volume force** enter vector expressions or values for the **Volume force source** $\mathbf{F}$ (SI unit: N/m^3).

Select an **Energy source type**: **Pressure rate of change** (the default) or **Heat source**. For **Pressure rate of change** enter an expression or value for the **Pressure rate of change source** p_{rc} (SI unit: Pa/s). For **Heat source** enter vector expressions or values for the **Heat source** Q (SI unit: W/m^3).

Background Acoustic Fields

The **Background Acoustic Fields** makes it possible to define a background field in a domain. This condition can be used to model scattering problems or as an inlet-like condition. In the latter case, when a perfectly matched layer is also present, you can set up a model with an incident field that also lets any reflected waves leave the computational domain. The feature is similar to the Background Pressure Field feature in Pressure Acoustics and the Background Acoustic Fields in Thermoviscous Acoustics.

This condition defines the background fields at the domain level (ρ_b, $\mathbf{u}_b$, p_b). The total acoustic field is now the sum of the scattered field (the dependent variables solved for) and the background field, such that

$$\rho_t = \rho + \rho_b \qquad \mathbf{u}_t = \mathbf{u} + \mathbf{u}_b \qquad p_t = p + p_b$$

On interior boundaries, continuity in the total field is automatically applied. All boundary conditions are expressed in terms if the total fields.

BACKGROUND ACOUSTIC FIELDS

Enter values or expressions for:

- **Background acoustic density** ρ_b (SI unit: kg/m^3).
- **Background acoustic velocity** $\mathbf{u}_b$ (SI unit: m/s).
- **Background acoustic pressure** p_b (SI unit: Pa).

Several predefined variables exist for the total, background, and scattered field: the temperature, the entropy, the intensity, and the sound pressure level. They are located in the plot menu group **Background and scattered fields**, when postprocessing.

Pressure (Isentropic)

The **Pressure (Isentropic)** node is used for prescribing a pressure p_p at a boundary assuming the isentropic relation between pressure and density (adiabatic relation). This means defining

$$p_t = p_p \qquad \rho_t = \frac{p_p}{c_0^2}.$$

Add this node from the **Prescribed Conditions** submenu.

PRESSURE

Enter a value or expression for the prescribed pressure p_p (SI unit: Pa).

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Prescribed Acoustic Fields

The **Prescribed Acoustic Fields** condition makes it possible to prescribe one or more of the dependent variables at a boundary. When specifying (constraining) the dependent variables for the linearized Euler equations, it is necessary to not underconstrain the system. Typically, this requires defining both the density and pressure, defining the velocity, or defining all three at the same time. Add this node from the **Prescribed Conditions** submenu.

For systems that can be assumed isentropic, where the acoustic changes in the entropy are zero, enter an expression for the pressure `p_user` and the density `p_user/1ef.c0^2`. This corresponds to the classical relation $p = c_0^2\rho$. This is also achieved using the Pressure (Isentropic) condition.

PRESCRIBED VALUES FOR ACOUSTIC VARIABLES

By default no checkboxes are selected. Click to select the following checkboxes as needed.

- **Prescribed density** ρ_p (SI unit: kg/m^3).
- **Prescribed velocity** (SI unit: m/s). If this checkbox is selected, choose one or more of these additional checkboxes: **Prescribed in x direction** u_{px}, **Prescribed in y direction** u_{py}, or **Prescribed in z direction** u_{pz} (for 3D components); or **Prescribed in r direction** u_{pr} and **Prescribed in z direction** u_{pz} (for 2D axisymmetric components).
- **Prescribed pressure** p_p (SI unit: Pa).

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Sound Soft Boundary (Isentropic)

Use the **Sound Soft Boundary (Isentropic)** condition to prescribe a zero acoustic pressure and zero acoustic density condition. The condition represents an boundary with a zero impedance, the opposite limit of a rigid wall. The condition prescribes:

$$p_t = 0 \qquad \rho_t = 0\,.$$

Add this node from the **Prescribed Conditions** submenu.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Impedance

Use the **Impedance** condition to specify a specific (input) impedance on an exterior boundary. The impedance can be any expression and can, for example, be a function of the frequency (`freq`). The condition can be used to model a porous lining in a duct or specifying an outlet impedance. The condition is based on the Ingard-Myers condition, see Ref. 1 and About the Impedance Boundary Condition located in the Theory Background for the Aeroacoustics Branch section. The condition is a so-called low frequency approximation; the viscous boundary layer of the background flow is assumed infinitely thin at the impedance wall. Add this node from the **Impedance Conditions** submenu.

IMPEDANCE (INGARD-MYERS)

Enter the value of the **Specific impedance** Z_n (SI unit: Pa·s/m). This can be an analytical expression or data provided through an interpolation function. Click to select the **Grazing flow conditions** checkbox, if the impedance is at a boundary with no normal velocity component for the background flow, $\mathbf{u}_0 \cdot \mathbf{n} = 0$. This simplifies the equations that are solved for in the boundary condition.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

For systems with a trailing edge on an airfoil or with an edge singularity between a liner and a rigid wall, the exclude edges and points functionality can be used to set up an appropriate Kutta condition. See Suppressing Constraints on Lower Dimensions.

Symmetry

The **Symmetry** node adds a symmetry condition at a boundary; it acts the same as the Rigid Wall condition.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and physics interfaces being modeled. This feature works well for cases like opposing parallel boundaries. In other cases, use the **Destination Selection** section to control the destination. By default, it contains the source and destination selection that COMSOL Multiphysics identifies. The periodic condition automatically defines a mapping between the source $\mathbf{x}_s$ and the destination $\mathbf{x}_d$ points on the boundaries.

BOUNDARY SELECTION

The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODIC CONDITION

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

- For **Continuity,** the values of the field variables at destination are set equal to the source: $\rho(\mathbf{x}_d) = \rho(\mathbf{x}_s)$, $\mathbf{u}(\mathbf{x}_d) = \mathbf{u}(\mathbf{x}_s)$, and $p(\mathbf{x}_d) = p(\mathbf{x}_s)$. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that corresponding velocity components are connected.
- For **Antiperiodicity**, the values of the field variables on the destination are set equal to the values on the source with the sign reversed: $\rho(\mathbf{x}_d) = -\rho(\mathbf{x}_s)$, $\mathbf{u}(\mathbf{x}_d) = -\mathbf{u}(\mathbf{x}_s)$, and $p(\mathbf{x}_d) = -p(\mathbf{x}_s)$. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that corresponding velocity components are connected.
- For **Floquet periodicity**, also known as Bloch periodicity, enter a **k-vector for Floquet periodicity** $\mathbf{k}_F$ (SI unit: rad/m) for the **x**, **y**, and **z** coordinates (3D components), the **r** and **z** coordinates (2D axisymmetric components), or **x** and **y** coordinates (2D components). This is the wave number of the excitation.

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

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

  This condition is used to model any geometry that has a cyclic periodic structure such as a microphone or a loudspeaker driver. Setting the azimuthal mode number determines what mode is analyzed. The response of the full system to an external excitation is in general a linear combination of many different modes. The option exists in 3D and 2D.
- For **User defined** select the checkbox for any of the field variables as needed. Then for each selection, choose the **Type of periodicity** — **Continuity** or **Antiperiodicity**. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that the corresponding velocity components are connected.

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

!	To get correct behavior of the **Periodic Condition**, in particular the **Floquet periodicity** and the **Cyclic symmetry** conditions, it is recommended that the source and destination meshes are identical. This can be achieved using the **Identical Mesh** mesh feature. When the **Periodic Condition** stretches across regions with a mix of default material models, PMLs, background pressure fields, or background acoustic fields, it is recommended to add one **Periodic Condition** for each set of such boundaries.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF DESTINATION

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog. For information about the **Orientation of Destination** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Mapping Between Source and Destination** section, see Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Moving Wall

The **Moving Wall** condition is used to model a vibrating wall with harmonic variations. The condition is a variant of the Ingard–Myers condition used for the **Impedance**. Add this node from the **Wall Conditions** submenu. See Ref. 1 and About the Moving Wall Boundary Condition located in the Theory Background for the Aeroacoustics Branch section for further details.

MOVING WALL

Select a **Displacement**: **Inward normal displacement** v_n (SI unit: m) (the default), **Displacement field** $\mathbf{v}_w$ (SI unit: m), **Inward normal velocity** u_n (SI unit: m/s), or **Velocity field** $\mathbf{u}_w$(SI unit: m/s). Then enter values or expressions based on the selection.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Specific Acoustic Impedance (Isentropic)

The **Specific Acoustic Impedance (Isentropic)** adds a simple acoustic impedance assuming isentropic (adiabatic) behavior for the fields. This condition should be used with care as it is not necessary that the solution is isentropic near a boundary. The Ingard–Myers type impedance condition is defined by the Impedance conditions. The isentropic specific acoustic impedance conditions reads:

$$\frac{p_t}{\mathbf{n} \cdot \mathbf{u}_t} = Z_n \qquad p_t = c_0^2 \rho_t$$

In transient models, use this boundary condition in combination with the Absorbing Layers for the Linearized Euler, Transient Interface to create efficient nonreflection conditions. Add this node from the **Impedance Conditions** submenu.

IMPEDANCE

Enter an expression or value for the **Specific impedance** Z_n (SI unit: Pa·s/m). The default value is set to `let.rho0*let.c0`, which is the specific characteristic impedance of a plane propagating wave.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Interior Wall

The **Interior Wall** boundary condition is used to model a wall condition on an interior boundary. It is similar to the Rigid Wall boundary condition available on exterior boundaries, except that it applies on both sides (up and down) of an interior boundary.

It allows discontinuities (in density, velocity, and pressure) across the boundary. Use the Interior Wall boundary condition to avoid meshing thin structures by applying this slip-like condition on interior curves and surfaces instead. Add this node from the **Interior Conditions** submenu.

Interior Impedance

Use the **Interior Impedance** condition to specify a specific transfer impedance on an interior boundary. The impedance can be any expression and can, for example, be a function of the frequency (`freq`). The condition is based on the Ingard-Myers condition, see Ref. 1 and About the Impedance Boundary Condition located in the Theory Background for the Aeroacoustics Branch section. The condition is a so-called low frequency approximation; the viscous boundary layer of the background flow is assumed infinitely thin at the impedance wall. Add this node from the **Interior Conditions** submenu.

INTERIOR IMPEDANCE (INGARD–MYERS)

Enter the value of the **Specific transfer impedance** Z_n (SI unit: Pa·s/m). This can be an analytical expression or data provided through an interpolation function. Click to select the **Grazing flow conditions** checkbox, if the impedance is at a boundary with no normal velocity component for the background flow, $\mathbf{u}_0 \cdot \mathbf{n} = 0$. This simplifies the equations that are solved for in the boundary condition.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

For systems with a trailing edge on an airfoil or with an edge singularity between a liner and a rigid wall, the exclude edges and points functionality can be used to set up an appropriate Kutta condition. See Suppressing Constraints on Lower Dimensions.

Asymptotic Far-Field Radiation

Use the **Asymptotic Far-Field Radiation** node to prescribe open nonreflecting radiation conditions. The boundary condition is based on the asymptotic behavior of the acoustic waves in the far field. The condition is also known as the condition of Tam and Dong (see Ref. 17). The condition applies at boundaries far from a compact acoustic source. The waves must be assumed spherical in 3D (and 2D axisymmetric) or cylindrical in 2D. This means that the condition is valid in the limit where

$$kr \gg 1 \qquad r = |\mathbf{r} - \mathbf{r}_0| \qquad k = \frac{\omega}{c_0}$$

where r is the distance from the source located at $\mathbf{r}_0$ and k is the wave number. The condition is formulated for the scattered field only, in problems involving an Background Acoustic Fields feature. The condition of Tam and Dong reads:

$$\left[\frac{1}{V(\theta)}\frac{\partial}{\partial t} + \frac{\partial}{\partial r} + \frac{1}{\eta r}\right]\mathbf{U} = 0 \qquad \mathbf{U} = \begin{bmatrix}\rho \\ \mathbf{u} \\ p\end{bmatrix}$$

$$V(\theta) = \mathbf{u}_0 \cdot \mathbf{e}_r + \sqrt{c_0^2 - (\mathbf{u}_0 \cdot \mathbf{e}_\theta + \mathbf{u}_0 \cdot \mathbf{e}_\phi)}$$

where $\eta = 1$ in the 3D (and 2D axisymmetric) spherical case and $\eta = 2$ in the 2D cylindrical case. In the frequency domain, the time derivative is replaced by a multiplication with $i\omega$.

The acoustic perturbations behave differently than the entropy and vorticity waves. The acoustic waves propagate in all directions at the speed of sound (corrected by the background flow) while the entropy and vorticity waves are only convected by the background flow. The linearized Euler equations support both these waves so both have to be taken care off. At boundaries where the background flow leaves the computational domain, add the Outflow Boundary subfeature. A default Outflow Boundary node is added with a cleared selections.

When using the **Asymptotic Far-Field Radiation** condition careful meshing near the boundary needs to be considered as gradients of the degrees of freedom enter the equation. Because of this challenge, using the absorbing layers (ALs) in the time

domain or the perfectly matched layers (PMLs) in the frequency domain is often more adequate and results in more accurate solution.

Details about Absorbing Layers for the Linearized Euler, Transient Interface are found in the Modeling with the Aeroacoustics Branch section.

ASYMPTOTIC FAR-FIELD RADIATION

Enter a **Source location** $\mathbf{r}_0$ (SI unit: m).

Outflow Boundary

A default **Outflow Boundary** subnode is added to the Asymptotic Far-Field Radiation. The outflow condition has to be added where the background mean flow leaves the computational domain to ensure that vorticity and entropy waves are not created here.

When solving, the interface can return an error if the condition is added where there is no background mean flow, $\mathbf{u}_0 = \mathbf{0}$. Here, certain terms evaluate to zero. Adding the condition at boundaries where the background mean flow enters the computational domain can result in unphysical results.

Continuity

Continuity is available as an option at interfaces between parts in a pair. This condition gives continuity in all the dependent variables: density, velocity, and pressure. It corresponds to a situation where the boundary has no direct effect on the acoustic fields (the subscripts "src" and "dest" in the equation refer to the source and destination, respectively — the two sides of the pair):

$$\mathbf{U}_{\text{src}} - \mathbf{U}_{\text{dest}} = 0, \qquad \mathbf{U} = (\rho_t, \mathbf{u}_t, p_t)^T$$

PAIR SELECTION

Select an identity pair. A pair is automatically created in the geometry when **Create pairs** is selected in the **Form Union/Assembly** step.

CONSTRAINT SETTINGS

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

The Linearized Euler, Transient Interface

The **Linearized Euler, Transient (let)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in density, velocity, and pressure in the presence of a stationary background mean-flow that is well approximated by an ideal gas flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized Euler equations.

The equations defined by the Linearized Euler, Transient interface are the linearized continuity, momentum (Euler), and energy equations. The physics interface solves for the acoustic variations in the density ρ, velocity field $\mathbf{u}$, and pressure p. The equations are formulated in the time domain. The background mean flow can be any stationary gas flow that is well approximated by an ideal gas. The coupling between the acoustic field and the background flow does not include any predefined flow induced noise. As the equations do not include any loss mechanisms, nonacoustic modes and instabilities can exist in the time domain.

The equations are defined using a scattered-field formulation just as in The Linearized Euler, Frequency Domain Interface and allows the use of Background Acoustic Fields. Open nonreflecting conditions are set up with the Absorbing Layers for the Linearized Euler, Transient Interface.

Coupling between a background mean flow, computed from a Fluid Flow model, and the Linearized Euler model is handled by the Background Fluid Flow Coupling multiphysics coupling and the dedicated Mapping study. Details are also found in the Mapping Between Fluid Flow and Acoustics Mesh section.

	For modeling tips and tricks and good practice, see the Modeling with the Aeroacoustics Branch section.

	Flow-induced noise functionality exists in pressure acoustics when using the Aeroacoustic Flow Source feature and associated multiphysics coupling and dedicated study.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Euler Model**, **Rigid Wall**, and **Initial Values**. For axisymmetric components an **Axial Symmetry** node is also added.

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

SETTINGS

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

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

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

STABILIZATION

See Stabilization in the frequency domain interface for details.

DAMPING PARAMETERS FOR ABSORBING LAYERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. In the **Damping Parameters for Absorbing Layers** section you can change and control the values of the artificial damping added in the Absorbing Layers for the Linearized Euler, Transient Interface. Enter a value for the **Numerical viscosity parameter** μ_{num} (default value is 100 Pa·s) and the **Numerical viscosity curvature** n (default value is 2).

TRANSIENT SOLVER SETTINGS

Enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. Select the **Time stepping** (method) as **Fixed (preferred)** the default and recommended or **Free**. The **Free** option is in general not recommended for wave problems. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be

reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

Details about Time Stepping in Transient Models are found in the Modeling with the Aeroacoustics Branch section.

DISCRETIZATION

See Discretization in the frequency domain interface for details.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Density** rho, **Velocity field** u and its components, and **Pressure** p. The name can be changed but the names of fields and dependent variables must be unique within a model.

- Domain, Boundary, and Pair Nodes for the Linearized Euler, Transient Interface
- Theory Background for the Aeroacoustics Branch

Domain, Boundary, and Pair Nodes for the Linearized Euler, Transient Interface

The Linearized Euler, Transient Interface has these nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

This physics interface shares some of its node settings with The Linearized Euler, Frequency Domain Interface.

- Asymptotic Far-Field Radiation[1]
- Axial Symmetry[1]
- Background Acoustic Fields[1]
- Continuity[1]
- Domain Sources[1]
- Initial Values
- Interior Wall[1]
- Linearized Euler Model[1]
- Moving Wall
- Rigid Wall[1]
- Prescribed Acoustic Fields[1]
- Pressure (Isentropic)[1]
- Specific Acoustic Impedance (Isentropic)[1]
- Symmetry[1]

[1]This feature is described in the Linearized Euler, Frequency Domain interface.

Initial Values

The **Initial Values** node adds initial values for the density, the velocity field, and pressure dependent variables. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial values of the **Density** `rho` (SI unit: kg/m^3), the **Velocity field** **u** (SI unit: m/s), and the **Pressure** `p` (SI unit: Pa).

Moving Wall

The **Moving Wall** condition can in the time domain only be defined in terms of the normal displacement. It is used to define a vibrating or moving wall. The displacement can be any time-dependent expression. The condition can, for example, be used to model an actuator creating sound in a gas flowmeter.

MOVING WALL

Enter a **Normal displacement** v_n (SI unit: m).

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

The Linearized Euler, Boundary Mode Interface

The **Linearized Euler, Boundary Mode** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute and identify propagating and nonpropagating modes in waveguides and ducts in the presence of a stationary background mean-flow that is well approximated by an ideal gas flow. The interface performs a boundary mode analysis on a boundary, inlet, or cross section of the waveguide. The interface exists in 3D and 2D axisymmetric geometries and applies to boundaries.

The interface solves for the acoustic variations of density, velocity, and pressure, as well as the out-of-plane wave number of the modes. The Linearized Euler, Boundary Mode interface is, for example, used when setting up sources in waveguide systems in the presence of a background mean-flow. The background flow is assumed to be uniform in the propagation direction.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Euler Model**, **Rigid Wall**, and **Initial Values**. For axisymmetric components, an **Axial Symmetry** node is also added. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Euler, Boundary Mode** to select physics features from the context menu.

SETTINGS

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

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

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

- Most boundary conditions and domain conditions are described in the The Linearized Euler, Frequency Domain Interface; except for Linearized Euler Model, Rigid Wall, and Initial Values.
- For theory details see Theory Background for the Aeroacoustics Branch.

Stabilization is not implemented for the Linearized Euler, Boundary Mode interface, which instead uses a default P1-P2-P1 discretization that ensures stability.

Linearized Euler Model

The **Linearized Euler Model** node adds the governing equations, and define the background mean flow and the ideal gas fluid properties. Gradient term suppression stabilization (advanced physics option) can be selected, if needed. See the Linearized Euler Model for The Linearized Euler, Frequency Domain Interface for details.

BOUNDARY MODE SETTINGS

Define the **Propagation direction** of the computed modes. The default is to use the surface physics normal (components `lebm.nx`, `lebm.ny`, and `lebm.nz`) that points out of domains.

Rigid Wall

The **Rigid Wall** condition is used to model a rigid wall, corresponding to the sound hard wall condition in Pressure Acoustics. In the case of lossless flows, this reduces to the slip condition $\mathbf{u}_t \cdot \mathbf{n} = 0$ where $\mathbf{n}$ is the normal to the edge, that is, the waveguide wall. See the Rigid Wall condition for The Linearized Euler, Frequency Domain Interface for further details.

Initial Values

The **Initial Values** node adds initial values for the density, velocity, and pressure variables.

INITIAL VALUES

Enter a value or expression for the initial values **Density** `rho` (SI unit: kg/m^3), **Velocity field** $\mathbf{u}$ (SI unit: m/s), and **Pressure** p (SI unit: Pa).

The Linearized Navier–Stokes, Frequency Domain Interface

The **Linearized Navier–Stokes, Frequency Domain (lnsf)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in pressure, velocity, and temperature in the presence of any stationary isothermal or nonisothermal background mean flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized Navier–Stokes equations.

The equations are formulated in the frequency domain and assume harmonic variation of all sources and fields. The equations include viscous losses and thermal conduction as well as the heat generated by viscous dissipation, if relevant. The coupling between the acoustic field and the background flow does not include any predefined flow induced noise.

Flow-induced noise functionality exists in pressure acoustics when using the Aeroacoustic Flow Source feature and associated multiphysics coupling and dedicated study.

The equations defined by the Linearized Navier–Stokes, Frequency Domain interface are the linearized continuity, momentum (Navier–Stokes), and energy equations. The physics interface solves for the acoustic variations in the pressure p, velocity field $\mathbf{u}$, and temperature T. The harmonic variation of all fields and sources is given by $e^{i\omega t}$ using the $+i\omega$ convention. The equations are formulated in the frequency domain for any fluid including losses due to viscosity and thermal conduction. The background mean flow can be any stationary flow.

The Linearized Navier–Stokes, Frequency Domain interface is formulated in the so-called scattered-field formulation where the total acoustic field (subscript t) is the sum of the scattered field (the field solved for p, $\mathbf{u}$, and T) and a possible background acoustic field (subscript "b"), such that

$$p_\mathrm{t} = p + p_\mathrm{b} \qquad \mathbf{u}_\mathrm{t} = \mathbf{u} + \mathbf{u}_\mathrm{b} \qquad T_\mathrm{t} = T + T_\mathrm{b}$$

All governing equations and boundary conditions are formulated in the total field variables. When no Background Acoustic Fields feature is present (the background field values are zero per default), the total field is simply the field solved for

$$p_t = p \qquad \mathbf{u}_t = \mathbf{u} \qquad T_t = T$$

Coupling the interfaces to structures enables detailed vibration analysis of structures in the presence of flow, such as FSI, in the frequency domain. The coupling in the frequency domain and time domain is readily performed using the predefined Aeroacoustic–Structure Boundary multiphysics coupling feature.

Coupling between a background mean flow, computed from a Fluid Flow model, and the Linearized Euler model is handled by the Background Fluid Flow Coupling multiphysics coupling and the dedicated Mapping study. Details are also found in the Mapping Between Fluid Flow and Acoustics Mesh section

	For modeling tips and tricks and good practice see the Modeling with the Aeroacoustics Branch section.

	The *Helmholtz Resonator with Flow: Interaction of Flow and Acoustics* tutorial model gives an example of how to model the detailed interaction between flow and acoustics. The model requires both the Acoustics Module and the CFD Module. Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/helmholtz_resonator_with_flow**

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Navier–Stokes Model**, **Wall**, and **Initial Values**. For axisymmetric components, an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Navier–Stokes, Frequency Domain** to select physics features from the context menu.

SETTINGS

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

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to

different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

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

EQUATION

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

- For **Study controlled**, the scaling of the equations is optimized for the numerical performance of the different solvers and study types.
- For **Frequency domain** you can manually enter the scaling parameter Δ under **Linearized Navier–Stokes Equation Settings** section.

LINEARIZED NAVIER–STOKES EQUATION SETTINGS

Click to select **Adiabatic formulation** to use an adiabatic equation of state and disable the temperature degree of freedom for the linearized Navier–Stokes equations. This formulation is applicable when the thermal losses can be disregarded; this is often the case in liquids like water. In gases, like air, on the other hand, the full formulation is necessary. When **Adiabatic formulation** is selected, all temperature conditions and options are disabled in the user interface.

In a 2D Axisymmetric component, select **Out-of-plane mode extension** to add the azimuthal component to the system of equations solved. When selected enter the **Azimuthal mode number** m (sometimes referred to as the circumferential mode number). The latter defines an azimuthal wave number $k_{\mathrm{m}} = m/r$. All out-of-plane gradients of the fields are then defined in terms of the wave number. When selected the Domain Sources, Background Acoustic Fields, and Prescribed Velocity features will have an additional input for the azimuthal component of the source.

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

The default **Scaling factor** Δ is $1/(i\omega)$. This value correspond to the equations for a Frequency Domain study when the equations are study controlled. To get the equations corresponding to an Eigenfrequency study, change the **Scaling factor** Δ to 1. Changing the scaling factor influences the coupling to other physics.

SOUND PRESSURE LEVEL SETTINGS

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

TYPICAL WAVE SPEED

Enter a value or expression for the **Typical wave speed for perfectly matched layers** c_{ref} (SI unit m/s). The default is `lnsf.c0` and the value is automatically taken from the material model. If several materials or material models are used, it is best practice to add one PML for each. This will ensure that the typical wavelength is continuous within each PML feature.

For convected acoustic problems, it can be necessary to set the value of the **Typical wavelength from** option in the PML feature manually, to correct for the convected background velocity.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Pressure** p, **Velocity field** u and its components, and **Temperature variation** T. The name can be changed but the names of fields and dependent variables must be unique within a model.

STABILIZATION

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog.

Select the **Stabilization Method** — **No stabilization applied**, **Galerkin least squares (GLS) stabilization** (the default), **Streamline upwind Petrov–Galerkin (SUPG) stabilization**, or **Streamline diffusion (legacy method)**. When stabilization is selected, enter a value for the **Stabilization constant** α_{stab} (dimensionless). The default value is `1` and is well suited for most modeling situations.

The default GLS stabilization is the most efficient stabilization method as it operates on the convective, reactive, and diffusive parts of the governing equations. This is also the default method and the method suggested for most applications. The stabilization constant α_{stab} can be tuned depending on the problem solved, the nature of the background mean flow, and on the computational mesh.

Note that if a problem is solved where no background mean flow is present, it is recommended to either use one of the Thermoviscous Acoustics Interfaces or turn off stabilization and switch to a P1-P2-P2 discretization.

- Stabilization in the Modeling with the Aeroacoustics Branch section.

DISCRETIZATION

From the list select the element order and type (Lagrange or serendipity) the default is **Linear** for all the dependent variables.

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

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Domain, Boundary, and Pair Nodes for the Linearized Navier–Stokes, Frequency Domain and Transient Interfaces
- Theory Background for the Aeroacoustics Branch

As a tutorial model showcasing acoustic–structure interaction in the presence of flow, that is FSI in the frequency domain, see the Application Gallery model: *Vibrating Plate in a 2D Viscous Parallel Plate Flow*.

https://www.comsol.com/model/vibrating-plate-in-a-2d-viscous-parallel-plate-flow-18871

An advanced example of a Coriolis flowmeter is found in the Application Gallery: *Coriolis Flowmeter: FSI Simulation in the Frequency Domain*.

https://www.comsol.com/model/coriolis-flowmeter-fsi-simulation-in-the-frequency-domain-51831

Both the models solve the linearized Navier–Stokes equations coupled to solid mechanics in the presence of flow. The models requires both the Acoustics Module and the CFD Module.

Domain, Boundary, and Pair Nodes for the Linearized Navier–Stokes, Frequency Domain and Transient Interfaces

The Linearized Navier–Stokes, Frequency Domain Interface and the The Linearized Navier–Stokes, Transient Interface has these domain, boundary, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- Adiabatic
- Axial Symmetry
- Background Acoustic Fields
- Boundary Stress
- Continuity
- Domain Sources
- First-Order Material Parameters
- Heat Flux
- Initial Values
- Interior Impedance
- Interior Wall
- Isothermal
- Linearized Navier–Stokes Model
- Periodic Condition
- Prescribed Pressure
- Prescribed Temperature
- Prescribed Velocity
- Pressure (Adiabatic)
- No Stress
- Impedance
- No Slip
- Slip (perfect)
- Symmetry
- Wall

Linearized Navier–Stokes Model

The **Linearized Navier–Stokes Model** sets up the governing equations, defines the background mean flow, fluid properties, and the compressibility and thermal expansion properties of the fluid. The governing equations solved are the continuity, momentum, and energy equations:

$$\begin{aligned}
&\frac{\partial \rho_t}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u}_t + \rho_t \mathbf{u}_0) = M \\
&\rho_0\left(\frac{\partial \mathbf{u}_t}{\partial t} + (\mathbf{u}_t \cdot \nabla)\mathbf{u}_0 + (\mathbf{u}_0 \cdot \nabla)\mathbf{u}_t\right) + \rho_t(\mathbf{u}_0 \cdot \nabla)\mathbf{u}_0 = \nabla \cdot \sigma + \mathbf{F} - \mathbf{u}_0 M \\
&\rho_0 C_p\left(\frac{\partial T_t}{\partial t} + (\mathbf{u}_t \cdot \nabla)T_0 + (\mathbf{u}_0 \cdot \nabla)T_t\right) + \rho C_p(\mathbf{u}_0 \cdot \nabla)T_0 \\
&-\alpha_p T_0\left(\frac{\partial p_t}{\partial t} + (\mathbf{u}_t \cdot \nabla)p_0 + (\mathbf{u}_0 \cdot \nabla)p_t\right) - \alpha_p T_t(\mathbf{u}_0 \cdot \nabla)p_0 = \nabla \cdot (k\nabla T_t) + \Phi + Q
\end{aligned} \tag{5-2}$$

where p_t, $\mathbf{u}_t$, and T_t are the acoustic perturbations to the pressure, velocity, and temperature, respectively. The subscript "t" refers to the fact that the acoustic variables are the total fields, that is, the sum of possible Background Acoustic Fields and the scattered fields.

In the frequency domain, the time derivatives of the dependent variables are replaced by multiplication with $i\omega$. The stress tensor is σ and Φ is the viscous dissipation

function. The right-hand-side source terms M, $\mathbf{F}$, and Q are initially zero; they can be defined using the Domain Sources feature. The variables with a zero subscript are the background mean flow values. The material parameters are defined below. Details about the physics and references are found in the Theory Background for the Aeroacoustics Branch section.

The constitutive equations are the stress tensor and the linearized equation of state, while the Fourier heat conduction law is readily included in the above energy equation,

$$\begin{aligned}\sigma &= -p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) + \left(\mu_B - \frac{2}{3}\mu\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\\ \rho_t &= \rho_0(\beta_T p_t - \alpha_p T_t)\end{aligned} \tag{5-3}$$

when **Adiabatic formulation** is selected in the Linearized Navier–Stokes Equation Settings section the equation of state reduces to

$$\rho_t = \rho_0\beta_s p_t = \rho_0\frac{\beta_T}{\gamma}p_t$$

The linearized viscous dissipation function is defined as

$$\begin{aligned}\Phi &= \nabla\mathbf{u}_t{:}\tau(\mathbf{u}_0) + \nabla\mathbf{u}_0{:}\tau(\mathbf{u}_t)\\ \tau(\mathbf{u}_t) &= \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) + \left(\mu_B - \frac{2}{3}\mu\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\\ \tau(\mathbf{u}_0) &= \mu(\nabla\mathbf{u}_0 + (\nabla\mathbf{u}_0)^T) + \left(\mu_B - \frac{2}{3}\mu\right)(\nabla\cdot\mathbf{u}_0)\mathbf{I}\end{aligned} \tag{5-4}$$

MODEL INPUTS

In order to model the influence of the background mean flow on the propagation of the acoustic waves in the fluid, the background mean flow temperature T_0, absolute pressure p_0, and velocity field $\mathbf{u}_0$ need to be defined. The density is defined in the **Fluid Properties** section below, and is per default taken from the material. It is thus a function of the model inputs, that is, the background pressure and temperature. Select **User defined** (the default) or **Common model input**. For **User defined** enter values for:

- **Background mean flow temperature** T_0 (SI unit K). The default is 293.15 K.

- **Background mean flow pressure** p_0 (SI unit: Pa). The default is 1 atm.
- **Background mean flow velocity** $\mathbf{u}_0$ (SI unit: m/s). The defaults are 0 m/s.

When modeling aeroacoustics, it is important how the Mapping Between Fluid Flow and Acoustics Mesh is done from a numerical perspective. The Background Fluid Flow Coupling multiphysics coupling handles this in an automated manner.

Physically the Coupling to Turbulent Flows (Eddy Viscosity) is also important to model the attenuation of acoustics waves due to turbulence.

Details about the Model Input and the Default Model Inputs are found in the Global and Local Definitions chapter of the *COMSOL Multiphysics Reference Manual*.

FLUID PROPERTIES

The defaults for the following are taken **From material**. For **User defined** edit the default values:

- **Background mean flow density** $\rho_0(p_0, T_0)$ (SI unit: kg/m^3). The **Ideal gas** option can also be selected as the density formulation.
- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Bulk viscosity** μ_B (SI unit: Pa·s).
- **Thermal conductivity** k (SI unit: W/(m·K)).
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)).

For a discussion about the air and water materials as commonly used in acoustics and other material properties, such as the bulk viscosity, see the Acoustic Properties of Fluids chapter of this manual.

THERMAL EXPANSION AND COMPRESSIBILITY

The **Thermal Expansion and Compressibility** section is displayed if **From material** or **User defined** is selected as the **Background mean flow density** under the **Fluid Properties** section. For the **Ideal gas** option the parameters are readily defined.

Select an option from the **Coefficient of thermal expansion** α_p list — **From material**, **From background mean flow density**, **From speed of sound** (the default), or **User defined**.

Select an option from the **Isothermal compressibility** β_T list — **From background mean flow density, From isentropic compressibility**, **From speed of sound** (the default), or **User defined**.

The different options for defining the (isobaric) coefficient of thermal expansion and the isothermal compressibility stem from their respective thermodynamic definitions:

$$\alpha_p = -\frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial T_0}\right]_p = \frac{1}{c_0}\sqrt{\frac{C_p(\gamma-1)}{T_0}}$$

$$\beta_T = \frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial p_0}\right]_T = \frac{1}{\rho_0}\frac{\gamma}{c_0^2} = \gamma\beta_s$$

- For the **From speed of sound** (the default for both) options the values for the **Speed of sound** c_0 (SI unit: m/s) and **Ratio of specific heats** γ (dimensionless) are taken **From material**. For **User defined** enter a different value or expression. The **From speed of sound** option is typically preferred (and the default) as the speed of sound and the ratio of specific heats are material properties often more readily available (see definition above).
- For the **From background mean flow density** the values are computed from the density expression (see definition above). This option is only valid if the density material property has the built in dependency on pressure p_0 and temperature T_0. This is not always the case.
- For **From material**, the coefficient of thermal expansion is taken from the materials node. Not all materials have this property defined. If this is the case, a small warning cross will appear on the materials node.
- For the **From isentropic compressibility** option, the values for the **Isentropic compressibility** β_s (SI unit: 1/Pa) and **Ratio of specific heats** γ (dimensionless) are taken **From material** (the default). For **User defined** enter a different values or expressions.
- For **User defined** enter a value or expression for the isobaric coefficient of thermal expansion α_p (SI unit: 1/K) and/or the isothermal compressibility β_T (SI unit: 1/Pa).

VISCOUS DISSIPATION FUNCTION

Select the **Include viscous dissipation function** checkbox if you want to include the heat source generated by the viscous losses. The viscous dissipation function Φ is defined in Equation 5-4.

GRADIENT TERM SUPPRESSION STABILIZATION

When the linearized Navier–Stokes (LNS) equations are solved in the time domain (or in the frequency domain with an iterative solver), linear physical instability waves can develop, the so-called Kelvin–Helmholtz instabilities. They are instabilities that can grow, but to less degree than in the linearized Euler equations because losses are present in the LNS equations. They are further naturally limited by nonlinearities in the full Navier–Stokes flow equations. The terms responsible for the instabilities are typically the reactive terms in the governing equations. It has been shown that in some problems the growth of these instabilities can be limited, while the acoustic solution is retained, by canceling terms involving gradients of the mean-flow quantities. This is known as gradient terms suppression (GTS) stabilization.

The GTS option can also be used to alter the governing equations to fit a desired formulation of the linearized Navier–Stokes equations. This can be relevant in certain applications.

All the aeroacoustic specific terms can be disabled. They are grouped into *Reactive terms* and *Convective terms*. Select the following checkboxes to activate the applicable gradient term suppression (GTS):

Reactive terms

- **Suppression of mean flow pressure gradients**

  This option sets the following term to zero:

$$\mathbf{u}_t \cdot \nabla p_0 = 0 \qquad \mathbf{u}_0 \cdot \nabla p_0 = 0$$

- **Suppression of mean flow velocity gradients**

  This option sets the following term to zero:

$$\nabla \cdot \mathbf{u}_0 = 0 \qquad \mathbf{u}_t \cdot \nabla \mathbf{u}_0 = 0 \qquad \mathbf{u}_0 \cdot \nabla \mathbf{u}_0 = 0$$

- **Suppression of mean flow temperature gradients**

  This option sets the following term to zero:

$$\mathbf{u}_t \cdot \nabla T_0 = 0 \qquad \mathbf{u}_0 \cdot \nabla T_0 = 0$$

- **Suppression of mean flow density gradients**

  This option sets the following term to zero:

$$\mathbf{u}_t \cdot \nabla \rho_0 = 0$$

Convective terms

- **Suppression of all convective terms**

  This option removes all the convective terms in the governing equations. This is a drastic measure as it removes the convective influence of the flow. It should be considered carefully before doing this. This option removes all terms of the type

$$(\mathbf{u}_0 \cdot \nabla)(\ldots) = 0$$

Domain Sources

Add a **Domain Sources** node to define the mass source, volume force source, and heat source, the right-hand-side of Equation 5-2. These can, for example, model varying thermal sources in a combustion chamber or a pulsating laser. Notice that the mass source term enters both the continuity and the momentum equation; this results from a consistent derivation of the governing equations. A mass source term will, when convected, also result in a momentum source.

DOMAIN SOURCES

Enter values or expressions for the following:

- **Mass source** M (SI unit: $\mathrm{kg/(m^3 \cdot s)}$).
- **Volume force source** $\mathbf{F}$ (SI unit: $\mathrm{N/m^3}$).
- **Heat source** Q (SI unit: $\mathrm{W/m^3}$).

First-Order Material Parameters

To display this node in the context menu, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

Use the **First Order Material Parameters** node to include subtle (acoustic) variations in the material parameters due to the acoustic variations of the dependent variables. The selected material parameters vary according to a linearization about their background values (at pressure p_0 and temperature T_0). The viscosity μ is, for example, replaced by:

$$\mu \rightarrow \mu + p\frac{\partial \mu}{\partial p}\bigg|_{T_0} + T\frac{\partial \mu}{\partial T}\bigg|_{p_0}$$

FIRST-ORDER MATERIAL PARAMETERS

By default, no checkboxes are selected. Click to select the following checkboxes as needed.

- **Derivatives of dynamic viscosity** $\partial\mu/\partial p$ (SI unit: s) and $\partial\mu/\partial T$ (SI unit: kg/(m·s·K)).
- **Derivatives of bulk viscosity** $\partial\mu_B/\partial p$ (SI unit: s) and $\partial\mu_B/\partial T$ (SI unit: kg/(m·s·K)).
- **Derivatives of heat capacity at constant pressure** $\partial C_p/\partial p$ (SI unit: $m^3/(kg\cdot K)$) and $\partial C_p/\partial T$ (SI unit: $m^2/(s^2\cdot K^2)$).
- **Derivatives of thermal conduction** $\partial k/\partial p$ (SI unit: $m^2/(s\cdot K)$) and $\partial k/\partial T$ (SI unit: $m\cdot kg/(s^2\cdot K^2)$).
- **Derivatives of coefficient of thermal expansion** $\partial\alpha_p/\partial p$ (SI unit: $m\cdot s^2/(kg\cdot K)$) and $\partial\alpha_p/\partial T$ (SI unit: $1/K^2$).

VISCOUS DISSIPATION FUNCTION

Select the **Include viscous dissipation function** checkbox to include the effects of the redefined material parameters on the viscous dissipation function.

Background Acoustic Fields

When the **Background Acoustic Fields** feature is added to a domain, it is possible to define the value of the background acoustic field variables p_b, $\mathbf{u}_b$, and T_b. Using this feature, it is possible to set up scattering problems as well as defining acoustic fields at an inlet of a waveguide (using a small domain at the inlet).

BACKGROUND ACOUSTIC FIELDS

Enter expressions for the **Background acoustic pressure** p_b, the **Background acoustic velocity** $\mathbf{u}_b$, and the **Background temperature variation** T_b. These can be analytical expressions or values of dependent variables solved in a previous study that defines the background field.

Initial Values

The **Initial Values** node adds initial values for the pressure, the velocity field, and temperature dependent variables. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial values of the **Pressure** p (SI unit: Pa), the **Velocity field** $\mathbf{u}$ (SI unit: m/s), and the **Temperature** T (SI unit: K).

Axial Symmetry

In a 2D axisymmetric geometry, the **Axial Symmetry** condition is automatically added on the axis of symmetry at $r = 0$. On this boundary, the radial component of the total velocity is set equal to zero

$$u_t = 0 \qquad \mathbf{u}_t = (u_t, v_t, w_t)$$

Wall

The **Wall** condition is the most common condition at solid surfaces. This condition contains both a mechanical and a thermal selection. The default is a no-slip and isothermal condition applicable in most cases. More advanced conditions at boundaries can be set up by combining any of the **Mechanical** and **Thermal** conditions available.

The **Wall** condition is also the default condition added when the physics interface is set up. Note that the selections in this default condition can be changed (they are not locked to all boundaries). When used together with a perfectly matched layer (PML), it is recommended to use the no-slip option at the outer boundary of the layer. Specifically, using the slip condition in combination with a rational scaling in the PML will lead to a singularity and an error.

The no-slip condition is the origin of the viscous boundary layer and the isothermal condition is the origin of the thermal boundary layer. It is within these acoustic boundary layers that the main dissipation happens. In some applications, for example, large muffler systems, it is often not necessary to model these loss effects and thus switching to perfect slip and adiabatic can be a good approximation. This also means that the mesh does not need to resolve the acoustic boundary layer which can save DOFs.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip (perfect)**. See the No Slip and Slip (perfect) conditions for further details.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

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

If **Slip (perfect)** is selected for the **Mechanical condition** and **Weak constraints** is selected as the **Constraint** option, then the default Nitsche (penalty like) formulation (**Automatic**) is switched to a Lagrange multiplier formulation instead. See the Slip (perfect) condition for details. For the **No-slip** and **Isothermal** selections, a weak formulation is used instead of a pointwise constraint.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Pressure (Adiabatic)

Add a **Pressure (Adiabatic)** node to give a pressure boundary condition with adiabatic conditions for the temperature. The condition is given by

$$p_t = p_p \qquad \sigma\mathbf{n} = p_p\mathbf{n} \qquad -\mathbf{n}\cdot(-k\nabla T_t) = 0$$

where p_p is the desired pressure at the boundary, σ is the stress tensor, and $\mathbf{n}$ the surface normal. This is a good approximation when prescribing a pressure at an inlet or outlet.

PRESSURE (ADIABATIC)

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

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Symmetry

The **Symmetry** node adds a boundary condition that represents symmetry. It corresponds to the Slip (perfect) condition for the mechanical degrees of freedom and the Adiabatic condition for the temperature variation. The symmetry condition should only be applied to flat surfaces.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and physics interfaces being modeled. This feature works well for cases like opposing parallel boundaries. In other cases, use the **Destination Selection** section to control the destination. By default, it contains the source and destination selection that COMSOL Multiphysics identifies. The periodic condition automatically defines a mapping between the source $\mathbf{x}_s$ and the destination $\mathbf{x}_d$ points on the boundaries.

BOUNDARY SELECTION

The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default, it contains the selection that COMSOL Multiphysics identifies.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODICITY SETTINGS

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

- For **Continuity**, the values of the field variables at destination are set equal to the source: $p(\mathbf{x}_d) = p(\mathbf{x}_s)$, $\mathbf{u}(\mathbf{x}_d) = \mathbf{u}(\mathbf{x}_s)$, and $T(\mathbf{x}_d) = T(\mathbf{x}_s)$. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that corresponding velocity components are connected.
- For **Antiperiodicity**, the values of the field variables on the destination are set equal to the values on the source with the sign reversed: $p(\mathbf{x}_d) = -p(\mathbf{x}_s)$, $\mathbf{u}(\mathbf{x}_d) = -\mathbf{u}(\mathbf{x}_s)$, and $T(\mathbf{x}_d) = -T(\mathbf{x}_s)$. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that corresponding velocity components are connected.
- For **Floquet periodicity**, also known as Bloch periodicity, enter a **k-vector for Floquet periodicity** $\mathbf{k}_F$ (SI unit: rad/m) for the **x**, **y**, and **z** coordinates (3D components), the **r** and **z** coordinates (2D axisymmetric components), or **x** and **y** coordinates (2D components). This is the wave number of the excitation.

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

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

  This condition is used to model any geometry that has a cyclic periodic structure such as a microphone or a loudspeaker driver. Setting the azimuthal mode number determines what mode is analyzed. The response of the full system to an external excitation is in general a linear combination of many different modes. The option exists in 3D and 2D.
- For **User defined** select the checkbox for any of the field variables as needed. Then for each selection, choose the **Type of periodicity** — **Continuity** or **Antiperiodicity**. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that corresponding velocity components are connected.

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

To get correct behavior of the **Periodic Condition**, in particular the **Floquet periodicity** and the **Cyclic symmetry** conditions, it is recommended that the source and destination meshes are identical. This can be achieved using the **Identical Mesh** mesh feature. When the **Periodic Condition** stretches across regions with a mix of default material models, PMLs, background pressure fields, or background acoustic fields, it is recommended to add one **Periodic Condition** for each set of such boundaries.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF DESTINATION

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog. For information about the **Orientation of Destination** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Mapping Between Source and Destination** section, see Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Interior Wall

The **Interior Wall** boundary condition is used to model a wall condition on an interior boundary. It is similar to the **Wall** boundary condition available on exterior boundaries except that it applies on both sides (up and down) of an interior boundary. It allows discontinuities (in pressure, velocity, and temperature) across the boundary. Use the **Interior Wall** boundary condition to avoid meshing thin structures by applying this slit-like condition on interior curves and surfaces instead.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip (perfect)**. See the No Slip and Slip (perfect) conditions for further details.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

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

If **Slip (perfect)** is selected for the **Mechanical condition** and **Weak constraints** is selected as **Constraint**, then the default Nitsche (penalty like) formulation is switched to a Lagrange multiplier formulation instead. For the **No-slip** and **Isothermal** selections a weak formulation is used instead of a pointwise constraint.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Interior Impedance

Use the **Interior Impedance** node to apply a transfer impedance condition on an interior boundary. The condition relates the normal stress at the two sides of the boundary (up and down) to the normal velocity via the normal specific transfer impedance Z_n. The condition allows discontinuities (a slit) in pressure and temperature depending on the condition chosen for the temperature.

$$(\sigma_{up} - \sigma_{down})\mathbf{n} = Z_n(\mathbf{u}_t \cdot \mathbf{n})\mathbf{n}$$

$$\sigma = \left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla \cdot \mathbf{u}_t)\mathbf{I}\right]$$

$$Z_n = \frac{p_{t,down} - p_{t,up}}{\mathbf{u}_t \cdot \mathbf{n}}$$

MECHANICAL

Enter a value or expression for the **Specific transfer impedance** Z_n (SI unit: Pa·s/m). This is the transfer impedance of the interior boundary.

Select a condition for the **Tangential velocity** — **Slip** (the default) or **No slip**. The **No Slip** option will apply an additional constraint for the velocity in the tangential direction on the two sides of the interior boundary.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

No Slip

The **No Slip** sets up a no-slip condition for the acoustic particle velocity on a hard wall:

$$\mathbf{u}_t = \mathbf{0}$$

Add this node from the **Mechanical** submenu.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Mechanical and thermal boundary conditions contribute with each other such that a condition can be set on the velocity and temperature simultaneously. Thermal type conditions override each other and the same is true for mechanical type conditions.

Slip (perfect)

Add a **Slip (perfect)** node to define perfect slip (no-penetration) for the acoustic particle velocity defined by

$$\mathbf{n} \cdot \mathbf{u}_t = 0$$

$$\sigma_n - (\sigma_n \cdot \mathbf{n})\mathbf{n} = 0, \qquad \sigma_n = \sigma\mathbf{n}$$

where σ is the stress tensor. The last equation expresses that no tangential stress exists at that boundary. Add this node from the **Mechanical** submenu.

This results in a so-called no-penetration condition where no viscous boundary layer is created. This condition can be used at boundaries where it is not necessary to model

the losses in the viscous boundary layer. Used together with the Adiabatic condition, no acoustic boundary layer is modeled.

When running a model with a perfectly matched layer (PML), it is recommended not to combine a **Slip (perfect)** condition with a rational scaling in the PML, as this will lead to a singularity and an error.

Per default, the **Slip (perfect)** condition uses a Nitsche formulation (also often referred to as discontinuous Galerkin or penalty formulation) to prevent locking problems on curved surfaces.

CONSTRAINT SETTINGS

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

Select **Weak constraints** as the **Constraint** if you want to switch from the default Nitsche formulation (**Automatic**) to a Lagrange multiplier formulation for the perfect slip condition (this formulation is not suited for an iterative solver). The Lagrange multiplier formulation also prevents locking problems.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Prescribed Velocity

Add a **Prescribed Velocity** node to define the velocity at a boundary. Add this node from the **Mechanical** submenu.

PRESCRIBED VELOCITY

Select the components to prescribe and enter a value for the **Prescribed in the direction** (SI unit: m/s).

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Prescribed Pressure

Add a **Prescribed Pressure** node to prescribe the pressure at a boundary (using a constraint). Add this node from the **Mechanical** submenu.

PRESCRIBED PRESSURE

Enter a **Pressure** p_p (SI unit: Pa).

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

No Stress

Use the **No Stress** node to set the total surface stress equal to zero:

$$\left[-p_t\mathbf{I}+\mu(\nabla\mathbf{u}_t+\nabla\mathbf{u}_t^{\mathrm{T}})-\left(\frac{2\mu}{3}-\mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n}=\mathbf{0}$$

Add this node from the **Mechanical** submenu.

Boundary Stress

Use the **Boundary Stress** node to define the total surface (boundary) stress through a stress tensor, a traction force, or a pressure. Add this node from the **Mechanical** submenu. The stress condition reads

$$\left[-p_t\mathbf{I}+\mu(\nabla\mathbf{u}_t+(\nabla\mathbf{u}_t)^{\mathrm{T}})-\left(\frac{2\mu}{3}-\mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n}=\sigma_p\mathbf{n}$$

where σ_p is a symmetric stress tensor, the product $\sigma_p\mathbf{n}$ defines a traction force. The traction condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = \mathbf{F}_{tr}$$

where $\mathbf{F}_{tr}$ is the surface traction force. The pressure condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -p_{bnd}\mathbf{n}$$

where p_{bnd} is the pressure applied to the boundary (acting in the normal direction).

STRESS

Select a **Stress type** — **Stress tensor, Traction** (the default), or **Pressure**. For **Stress tensor** enter the (symmetric) **Stress tensor** σ_p (SI unit: N/m^2) components, for **Traction** enter the traction force $\mathbf{F}_{tr}$ (SI unit: N/m^2) components, and for **Pressure** enter the boundary pressure p_{bnd} (SI unit: Pa).

Impedance

Use the **Impedance** node to specify a (normal) specific impedance Z_n on a boundary. Add this node from the **Mechanical** submenu. The condition has a user defined option as well as the option to define a simple RCL impedance model. The latter is useful for modeling the interaction with, for example, a microphone represented by a simple mass-spring-damper system.

The boundary condition reads:

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + \nabla\mathbf{u}_t^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -Z_n(\mathbf{u}_t\cdot\mathbf{n})\mathbf{n}\,.$$

The condition relates the normal stress at the boundary to the normal velocity, through the value if the specific impedance. The tangential velocity can optionally be constrained to zero.

MECHANICAL

Select the Impedance model as **User defined** (the default) or **RCL**. For a transient model only the **User defined** option exists.

- For **User defined** enter a value or expression for the **Specific impedance** Z_n (SI unit: Pa·s/m).

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

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

  Notice the matching diagram and **Equation** section information for each choice. Then enter the **Equivalent acoustic resistance** R_{ac} (SI unit: $kg/(m^4 \cdot s)$), the **Equivalent acoustic compliance** C_{ac} (SI unit: $m^4 \cdot s^2/kg$) and the **Equivalent acoustic inertance** L_{ac} (SI unit: kg/m^4).

Select a condition for the **Tangential velocity** — **Slip (perfect)** (the default) or **No slip**. With the **Slip (perfect)** option the feature is applied outside the viscous boundary layer, as this condition mimics the behavior of a corresponding **Pressure Acoustics Model** with an impedance condition. The **No Slip** option will apply an additional constraint for the velocity in the tangential direction on the boundary. In this case, the impedance condition could represent a lumped mechanical property of a wall, including the boundary layer effects.

Isothermal

The **Isothermal** node sets up an isothermal condition:

$$T_t = 0$$

This is a good approximation on solid walls, as heat conduction is typically much higher in solids than in fluids. Add this node from the **Thermal** submenu.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Adiabatic

Add an **Adiabatic** node to set up an adiabatic condition for the temperature, that is, the natural condition for the temperature

$$-\mathbf{n} \cdot (-k\nabla T_t) = 0$$

Add this node from the **Thermal** submenu.

Prescribed Temperature

Add a **Prescribed Temperature** node to define the temperature variation T_p at a boundary. Add this node from the **Thermal** submenu.

PRESCRIBED TEMPERATURE

Enter a value or expression for the **Temperature** T_p (SI unit: K).

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Heat Flux

Add a **Heat Flux** node to define the inward normal heat flux q_n at a boundary:

$$-\mathbf{n} \cdot (-k\nabla T_t) = q_n$$

Add this node from the **Thermal** submenu.

HEAT FLUX

Enter a value or expression for the **Inward normal heat flux** q_n (SI unit: W/m^2).

Continuity

Continuity is available as an option at interfaces between parts in a pair. This condition gives continuity in all the dependent variables: pressure, velocity, and temperature. It corresponds to a situation where the boundary has no direct effect on the acoustic fields (subscripts "src" and "dest" in the equation refers to the source and destination, the two sides of the pair):

$$\mathbf{U}_{src} - \mathbf{U}_{dest} = 0, \qquad \mathbf{U} = (p_t, \mathbf{u}_t, T_t)^T$$

The Linearized Navier–Stokes, Transient Interface

The **Linearized Navier–Stokes, Transient (lnst)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in pressure, velocity, and temperature in the presence of any stationary isothermal or nonisothermal background mean-flow. The physics interface is used for aeroacoustic simulations that can be described by the linearized Navier–Stokes equations.

The equations are formulated in the time domain and include viscous losses and thermal conduction as well as the heat generated by viscous dissipation, if relevant. The coupling between the acoustic field and the background flow does not include any predefined flow induced noise.

	Flow-induced noise functionality exists in pressure acoustics when using the Aeroacoustic Flow Source feature and associated multiphysics coupling and dedicated study.

The equations defined by the Linearized Navier–Stokes, Transient interface are the linearized continuity, momentum (Navier–Stokes), and energy equations. The physics interface solves for the acoustic variations in the pressure, velocity, and temperature. The equations are formulated in the time domain for any fluid including losses due to viscosity and thermal conduction. The background mean flow can be any stationary flow.

The governing equations are defined using a scattered field formulation just as in The Linearized Navier–Stokes, Frequency Domain Interface and can use Background Acoustic Fields feature.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Navier–Stokes Model**, **Wall**, and **Initial Values**. For axisymmetric components, an **Axial Symmetry** node is also added. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Navier–Stokes, Transient** to select physics features from the context menu.

Coupling between a background mean flow, computed from a Fluid Flow model, and the Linearized Euler model is handled by the Background Fluid Flow Coupling multiphysics coupling and the dedicated Mapping study. Details are also found in the Mapping Between Fluid Flow and Acoustics Mesh section

For modeling tips and tricks and good practice see the Modeling with the Aeroacoustics Branch section.

SETTINGS

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

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

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

TRANSIENT SOLVER SETTINGS

Enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. Select the **Time stepping** (method) as **Fixed (preferred)** the default and recommended or **Free**. The **Free** option is in general not recommended for wave problems. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

Details about Time Stepping in Transient Models are found in the Modeling with the Aeroacoustics Branch section.

The Stabilization, Dependent Variables, and Discretization settings are the same as for The Linearized Navier–Stokes, Frequency Domain Interface.

- For boundary conditions and domain conditions see Domain, Boundary, and Pair Nodes for the Linearized Navier–Stokes, Frequency Domain and Transient Interfaces except for the Impedance condition.
- For theory details, see Theory Background for the Aeroacoustics Branch.

Impedance

Use the **Impedance** condition to specify a normal specific impedance Z_n on a boundary. Add this node from the **Mechanical** submenu. The boundary condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -Z_n(\mathbf{u}_t\cdot\mathbf{n})\mathbf{n}$$

The condition relates the normal stress at the boundary to the normal velocity, through the value if the specific impedance. The tangential velocity can optionally be constrained to zero.

MECHANICAL

Enter a value or expression for the **Specific impedance** Z_n (SI unit: Pa·s/m).

Select a condition for the **Tangential velocity** — **Slip (perfect)** (the default) or **No slip**. The **No Slip** option will apply an additional constraint for the velocity in the tangential direction on the boundary. When **Slip (perfect)** is selected the impedance mimics the behavior of a corresponding pressure acoustics model with a normal impedance condition.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

The Linearized Navier–Stokes, Boundary Mode Interface

The **Linearized Navier–Stokes, Boundary Mode (lnsbm)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute and identify propagating and nonpropagating modes in waveguides and ducts in the presence of any stationary isothermal or nonisothermal background mean-flow. The interface performs a boundary mode analysis on a boundary, inlet, or cross section of the waveguide. The interface considers all thermal and viscous loss effects and the interaction with the background flow. This includes the acoustic boundary layer losses, if necessary. The interface exists in 3D and 2D axisymmetric geometries and applies to boundaries.

The interface solves for the acoustic variations of pressure, velocity, and temperature, as well as the out-of-plane wave number of the modes. The Linearized Navier–Stokes, Boundary Mode interface is, for example, used when setting up sources in waveguide systems in the presence of a background mean-flow. The background flow is assumed to be uniform in the propagation direction.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Navier–Stokes Model**, **Wall**, and **Initial Values**. For axisymmetric components, an **Axial Symmetry** node is also added. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Navier–Stokes, Boundary Mode** to select physics features from the context menu.

SETTINGS

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

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

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

For boundary conditions and domain conditions see:

- Domain, Boundary, and Pair Nodes for the Linearized Navier–Stokes, Frequency Domain and Transient Interfaces
- Theory Background for the Aeroacoustics Branch

Stabilization is not implemented for the Linearized Navier–Stokes, Boundary Mode interface, which instead uses a default P1-P2-P2 discretization that ensures stability.

Linearized Navier–Stokes Model

The **Linearized Navier–Stokes Model** adds the governing equations, and defines the background mean flow and the fluid properties. Gradient term suppression stabilization (advanced physics option) can be selected, if needed. See the Linearized Navier–Stokes Model for The Linearized Navier–Stokes, Frequency Domain Interface for details.

BOUNDARY MODE SETTINGS

Define the **Propagation direction** of the computed modes. The default is to use the surface physics normal (components `lnsbm.nx`, `lnsbm.ny`, and `lnsbm.nz`) that points out of domains.

Wall

The **Wall** condition is the most common condition at solid surfaces. This condition contains both a mechanical and a thermal selection. The default is a no-slip and isothermal condition applicable in most cases. More advanced conditions at boundaries can be set up by combining any of the **Mechanical** and **Thermal** conditions available. See the Wall condition for The Linearized Navier–Stokes, Frequency Domain Interface for details.

Initial Values

The **Initial Values** node adds initial values for the density, velocity, and pressure variables.

INITIAL VALUES

Enter a value or expression for the initial values **Pressure** p (SI unit: Pa), **Velocity field** $\mathbf{u}$ (SI unit: m/s), and **Temperature variation** T (SI unit: K).

Impedance

For the **Impedance** condition see Impedance (without the **RCL** option) available for The Linearized Navier–Stokes, Frequency Domain Interface.

The Linearized Potential Flow, Frequency Domain Interface

The **Linearized Potential Flow, Frequency Domain (lpff)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in the velocity potential in the presence of an inviscid and irrotational background mean flow — that is, a potential flow. The background flow can be modeled using The Compressible Potential Flow Interface. The physics interface is used for convected acoustic simulations that can be described by the linearized compressible potential flow equations.

The equations are formulated in the frequency domain and assume harmonic variation of all sources and fields. The physics interface is limited to flows with a Mach number $Ma < 1$, partly due to limitations in potential flow and partly due to the acoustic boundary settings needed for supersonic flow. The coupling between the acoustic field and the background flow does not include any predefined flow-induced noise.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Potential Flow Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**. For axisymmetric components, an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Potential Flow, Frequency Domain** to select physics features from the context menu.

SETTINGS

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

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

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

LINEARIZED POTENTIAL FLOW EQUATION SETTINGS

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

	For 2D components, the **Out-of-plane wave number** k_z (SI unit: rad/m) is 0 rad/m by default.
	For 2D axisymmetric components the **Azimuthal wave number** m (dimensionless) is 0 by default.

SOUND PRESSURE LEVEL SETTINGS

The settings are the same as Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface.

GLOBAL PORT SETTINGS

Select the **Port sweep setting** as **No port sweep** (the default) or **Activate port sweep** to enable the port sweep functionality. The **Activate port sweep** option is used to compute the full scattering matrix when Port conditions are used. When selected, also specify a name for the sweep parameter in the **Sweep parameter name** field.

Select the **Mode shape normalization** as **Amplitude normalization** (the default), **Power normalization**, or **Intensity normalization**. This setting controls if the pressure mode shapes p_n are normalized to have a unit maximum amplitude or carry unit power or intensity. The intensity and power normalization are usually used for convective acoustic simulations in jet turbofan noise modeling, Ref. 24 and Ref. 25. The selection determines how the scattering matrix is to be interpreted.

The **Amplitude normalization** ensures that $\max(|p_\mathrm{n}|) = 1$, the **Power normalization** ensures that the pressure modes fulfill

$$\frac{1}{A}\int p_\mathrm{n} p^*{}_\mathrm{n} dA = 1$$

where A is the port areas. Finally, the **Intensity normalization** gives

$$\int p_\mathrm{n} p^*{}_\mathrm{n} dA = 1\,.$$

TYPICAL WAVE SPEED FOR PERFECTLY MATCHED LAYERS

Enter a value or expression for the typical wave speed for perfectly matched layers c_{ref} (SI unit m/s). The default is `lpff.c0` and the value is automatically taken from the material model. If several materials or material models are used, it is best practice to add one PML for each. This will ensure that the typical wavelength is continuous within each PML feature.

For convected acoustic problems, it can be necessary to set the value of the **Typical wavelength from** option in the PML feature manually, to correct for the convected background velocity.

DISCRETIZATION

From the list select the element order and type (Lagrange or serendipity) for the velocity potential, the default is **Quadratic Lagrange**.

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

DEPENDENT VARIABLES

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

- Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Frequency Domain Interface
- Theory Background for the Aeroacoustics Branch

- *Flow Duct*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct**
- *Flow Duct — With Boundary Mode Analysis*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct_boundary_mode**
- *Flow Duct — Modes with Impedance Condition*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct_impedance**
- *Jet Pipe*: Application Library Path **Acoustics_Module/Aeroacoustics_and_Noise/jet_pipe**
- *Doppler Shift*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/doppler_shift**

Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Frequency Domain Interface

The Linearized Potential Flow, Frequency Domain Interface has these domain, boundary, edge, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- Axial Symmetry
- Continuity
- Initial Values
- Impedance
- Interior Impedance and Pair Impedance
- Interior Sound Hard Boundary (Wall)
- Linearized Potential Flow Model
- Mass Flow Circular Source
- Mass Flow Edge Source
- Mass Flow Point Source
- Normal Mass Flow
- Normal Velocity
- Periodic Condition
- Plane Wave Radiation
- Port
- Port Reference Axis
- Sound Hard Boundary (Wall)
- Sound Soft Boundary
- Symmetry
- Velocity Potential
- Vortex Sheet

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

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Linearized Potential Flow Model

The **Linearized Potential Flow Model** node adds the equations for frequency domain aeroacoustics modeling. You here need to enter the material properties as well as the background mean flow information.

MODEL INPUTS

In order to model the influence of the background mean flow on the propagation of the acoustic waves in the fluid, the background mean flow temperature T_0, absolute pressure p_0, and velocity field $\mathbf{u}_0$ need to be defined.

Select **User defined** (the default), **Common model input**, or a variable defined by a flow simulation performed using the CFD Module. By default, they are set to the quiescent

background conditions of air. All the background flow parameters can also be constants or analytical expressions functions of space. When the Background Potential Flow Coupling multiphysics is used to couple The Compressible Potential Flow Interface to the acoustics domain, the required input field will appear locked (grayed out).

Details about the Model Input and the Default Model Inputs are found in the Global and Local Definitions chapter of the *COMSOL Multiphysics Reference Manual.*

It is important that the velocity field is a solution to a compressible potential flow simulation. It has to be an irrotational and inviscid flow, for example, a constant flow field $\mathbf{u}_0$ or a solution modeled with The Compressible Potential Flow Interface. Any other type of flow yields unphysical solutions for this formulation of the governing equations.

Enter **User defined** values for the **Background mean flow temperature** T_0 (SI unit: K), **Background mean flow pressure** p_0 (SI unit: Pa), and **Background mean flow velocity** $\mathbf{u}_0$ (SI unit: m/s). The defaults are 293.15 K, 1 atm, and 0 m/s, respectively.

Note that the **Background mean flow density** and **Speed of Sound** also need to be defined or entered in the **Fluid Properties** section below.

FLUID PROPERTIES

The default values for the **Density** ρ_0 (SI unit: kg/m^3) and the **Mean flow speed of sound** c_0 (SI unit: m/s) are taken **From material**. For **User defined**, enter other values or expressions.

Coupling the flow from The Compressible Potential Flow Interface interface to the linearized potential flow model can be set up using the Background Potential Flow Coupling found under the **Multiphysics Couplings** node. The feature appears when both physics interfaces are present in a model.

!	Material discontinuities, that is jumps in the material properties, are not handled by the interface. A jump due to the presence of a shear layer that separates a stream from the free velocity field is handled using the Vortex Sheet condition.

Initial Values

The **Initial Values** node adds initial values for the velocity potential. For The Linearized Potential Flow, Transient Interface it also adds initial values for the velocity potential, first time derivative. If needed, add more **Initial Values** nodes from the **Physics** toolbar or the main physics interface node's context menu.

INITIAL VALUES

Enter a value or expression for the initial value of the **Velocity potential** phi (SI unit: m^2/s).

For The Linearized Potential Flow, Transient Interface also enter a **Velocity potential, first time derivative**, $\partial\phi/\partial t$ (SI unit: m^2/s^2).

Sound Hard Boundary (Wall)

Use the **Sound Hard Boundary (Wall)** condition to model rigid boundary surfaces or walls. It prescribes a vanishing normal component of the particle velocity at the boundary. Multiplied by the density, it can equivalently be expressed as a *no-flow* condition:

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{u}_0\cdot\nabla\phi\right)\right)=0$$

The sound-hard boundary condition is available for all study types. The equation above applies to the time-domain calculations in The Linearized Potential Flow, Transient Interface; to obtain the corresponding condition for frequency domain, simply replace $\partial/\partial t$ by $i\omega$.

In The Linearized Potential Flow, Boundary Mode Interface, the no-flow or wall condition, known as *sound hard*, sets the normal acceleration — and thus also the normal velocity — to zero at the edge.

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi+(\nabla\phi\cdot\mathbf{u}_0))\right)=0$$

Velocity Potential

Use the **Velocity Potential** node when coupling two Linearized Potential Flow, Frequency Domain interfaces together because it can sometimes be necessary to set the velocity potential: $\phi=\phi_0$.

VELOCITY POTENTIAL

Enter a **Velocity potential** ϕ_0 (SI unit: m^2/s).

CONSTRAINT SETTINGS

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

Normal Mass Flow

Use the **Normal Mass Flow** node to set the inward mass flow boundary condition.

For The Linearized Potential Flow, Frequency Domain Interface, the natural boundary condition for the total wave has the meaning of a mass flow through the boundary surface:

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{u}_0\cdot\nabla\phi\right)\right)=m_\mathrm{n}$$

For The Linearized Potential Flow, Boundary Mode Interface, the natural edge condition for the total wave has the meaning of normal mass flow.

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi+(\nabla\phi\cdot\mathbf{u}_0)+\lambda\phi u_\mathrm{n})\right)=m_\mathrm{n}$$

NORMAL MASS FLOW

Enter an **Inward mass flow** m_n (SI unit: $\mathrm{kg}/(\mathrm{m}^2\cdot\mathrm{s})$).

Plane Wave Radiation

The **Plane Wave Radiation** is a class of nonreflecting boundary conditions, which assume that there is an outgoing plane wave, and optionally also an incoming exciting wave.

For transient analysis the boundary condition is

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}+(\nabla\phi\cdot\mathbf{u}_0)\right)\right)-\rho_0 k_n\frac{\partial\phi}{\partial t}-\mathbf{n}\cdot\mathbf{u}_0\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}-k_n\frac{\partial\phi}{\partial t}\mathbf{n}\cdot\mathbf{u}_0\right)=$$

$$\rho_0\frac{\partial}{\partial t}\phi_0 k_k(\mathbf{n}\cdot\mathbf{n}_\mathbf{k})-\mathbf{n}\cdot\mathbf{u}_0\frac{\rho_0}{c_0^2}k_k\frac{\partial\phi_0}{\partial t}\mathbf{n}_\mathbf{k}\cdot\mathbf{u}_0-\rho_0 k_n\frac{\partial\phi_0}{\partial t}+\mathbf{n}\cdot\mathbf{u}_0\frac{\rho_0}{c_0^2}\left(k_n\frac{\partial\phi_0}{\partial t}\mathbf{n}\cdot\mathbf{u}_0\right)$$

$$k_k=\frac{1}{c_0+\mathbf{u}_0\cdot\mathbf{n}_\mathbf{k}}\qquad k_n=\frac{1}{c_0+\mathbf{u}_0\cdot\mathbf{n}}$$

while the corresponding time-harmonic equation reads

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi+\nabla\phi\cdot\mathbf{u}_0)\right)-\rho_0 ik_n\phi-\mathbf{n}\cdot\mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega-ik_n\mathbf{n}\cdot\mathbf{u}_0)\phi=$$

$$\rho_0\left(ik_k\mathbf{n}\cdot\mathbf{n}_\mathbf{k}-\mathbf{n}\cdot\frac{\mathbf{u}_0}{c_0^2}(ik_k\mathbf{n}_\mathbf{k}\cdot\mathbf{u}_0)-ik_n+\mathbf{n}\cdot\frac{\mathbf{u}_0}{c_0^2}(ik_n\mathbf{n}\cdot\mathbf{u}_0)\right)\phi_0 e^{-i\mathbf{k}\cdot\mathbf{r}}$$

$$k_k=\frac{\omega}{c_0+\mathbf{u}_0\cdot\mathbf{n}_\mathbf{k}}\qquad k_n=\frac{\omega}{c_0+\mathbf{u}_0\cdot\mathbf{n}}\qquad\mathbf{k}=k\,\mathbf{n}_\mathrm{k}\qquad\mathbf{n}_\mathrm{k}=\frac{\mathbf{e}_\mathrm{k}}{|\mathbf{e}_\mathrm{k}|}$$

Specify an Incident Velocity Potential (incoming plane wave)

$$\phi_0 e^{-i\mathbf{k}\cdot\mathbf{r}}$$

by supplying its amplitude, ϕ_0, and propagation wave direction vector, $\mathbf{e}_\mathrm{k}$. The vector $\mathbf{n}_\mathrm{k}$ is the normalized wave direction vector of unit length.

This boundary condition can be used for a port, if only plane waves are excited. In general, it is recommended to use the Port condition for waveguides.

Incident Velocity Potential

The **Incident Velocity Potential** subnode is available from the context menu (right-click the Plane Wave Radiation parent node) or from the **Physics** toolbar, **Attributes** menu. Use this to add a velocity potential and wave direction.

INCIDENT VELOCITY POTENTIAL

Enter a **Velocity potential** ϕ_0 (SI unit: m^2/s) and **Wave direction** $\mathbf{e}_k$ (SI unit: m). The default for the wave direction is the inward normal direction of the boundary.

Sound Soft Boundary

The **Sound Soft Boundary** creates a boundary condition for a sound soft boundary, where the acoustic pressure vanishes and $p = 0$.

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

CONSTRAINT SETTINGS

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

Periodic Condition

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

BOUNDARY SELECTION

The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODIC CONDITION

Select a **Type of periodicity**: **Continuity** (the default) or **Antiperiodicity**.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF DESTINATION

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog. For information about the **Orientation of Destination** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Mapping Between Source and Destination** section, see Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Normal Velocity

Use the **Normal Velocity** node in time-harmonic analysis to specify the velocity component normal to the boundary:

$$\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi+\mathbf{u}_0\cdot\nabla\phi)\right)=\rho_0\left(v_\mathrm{n}+\frac{1}{i\omega}\mathbf{u}_0\cdot\nabla v_\mathrm{n}\right)$$

Here, v_n denotes the outward normal velocity at the boundary surface, which is specified in the v_n text field. The formulation is a variant of the Ingard–Myers type impedance condition as $v_\mathrm{n} = p/Z_\mathrm{i}$ (see the Impedance condition).

NORMAL VELOCITY

Enter a **Normal velocity** v_n (SI unit: m/s).

Impedance

Use the **Impedance** node in frequency domain analysis to define the specific (input) impedance of an external domain as the ratio of pressure to normal velocity $Z_\mathrm{n} = p/(\mathbf{n}\cdot\mathbf{v})$, at the boundary. The formulation is following the so-called Ingard–Myers condition. The associated impedance boundary condition is

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\frac{\rho_0}{c_0^2}(i\omega\phi+\mathbf{u}_0\cdot\nabla\phi)\mathbf{u}_0\right)=\rho_0\left(\frac{p}{Z_\mathrm{n}}+\frac{1}{i\omega}(\mathbf{u}_0\cdot\nabla)\frac{p}{Z_\mathrm{n}}\right)$$

IMPEDANCE (INGARD-MYERS)

Enter a **Specific impedance** Z_n (SI unit: Pa·s/m).

Symmetry

Use the **Symmetry** condition on symmetry boundaries. Using symmetries in a model is important in order to reduce the model complexity. Assigning symmetries is also important when other features analyze a model setup. Symmetries are, for example, important in the correct behavior of the Port condition.

Port

The **Port** boundary condition is used to excite and absorb specific acoustic modes that enter or leave waveguide structures, like a duct or channel, in a linearized potential flow model. A given port condition supports one specific propagating mode. To provide the full acoustic description, combine several port conditions on the same boundary. Make sure that all relevant propagating modes in the studied frequency range are included (among the modes that have a cutoff frequency in the frequency

range). By doing this, the combined port conditions provide a superior nonreflecting or radiation condition for waveguides to, for example, the Plane Wave Radiation condition or a perfectly matched layer (PML) configuration. Modal decomposition is also used in jet turbofan noise modeling, where the noise source from the rotating turbofan is decomposed into modes; see Ref. 24 and Ref. 25. The **Annular** and **Circular** port type options provide the standardized modes to define the sources for modal source decomposition.

The same port boundary condition feature should not be applied to several waveguide inlets or outlets. The port condition supports S-parameter (scattering parameter) calculation, but it can also be used as a source to just excite a system. The **Port** boundary condition exists for 3D and 2D axisymmetric models.

On a given boundary, a combination of ports will define the total acoustic field (the sum of incident and outgoing waves). For a linearized potential flow model the total velocity potential ϕ is then

$$\phi = \sum_{i \in \text{bnd}} A^{\text{in}} e^{i\varphi} \phi_j^{\text{in}} \delta_{ij} + S_{ij} \phi_i^{\text{out}}$$

where the summation "i" is over all ports on the given boundary "bnd", S_{ij} is the scattering parameter, A^{in} is the amplitude of the incident field (at port "j"), φ is the phase of the incident field, ϕ^{in} is the mode shape of the incident mode, and ϕ^{out} is the mode shape of the outgoing mode. In the presence of a background flow, incident and outgoing modes shapes are not identical for the same mode. The mode wavenumber will also be shifted due to the convective effects. The mode shapes are normalized to have either a unit maximum amplitude, a unit power, or a unit intensity (see the normalization option in the Global Port Settings section). The scattering parameter S_{ij} defines the amplitude of mode i when a system is exited at port j (with mode j). This corresponds to a multimode expansion of the solution on the given port. The

scattering parameters are automatically calculated when an acoustic model is set up with just one port exciting the system.

When working with ports in the Linearized Potential Flow interfaces, it is important to remember that computed duct modes (for example, using The Linearized Potential Flow, Boundary Mode Interface) do not represent an orthogonal basis set when a nonuniform background flow or an impedance boundary condition is introduced (see Ref. 26). This fact is taken into account in the formulation of the port equations, by including so-called cross-terms between the modes.

The modes associated with the **Annular** and **Circular** port type options are orthogonal by nature, as they assume uniform flow and sound hard wall conditions.

In practice, this means that energy-based postprocessing variables like the incident or outgoing power of a given mode cannot be used directly. The energy-based expressions are given by the product of acoustic variables, so when modes are no longer orthogonal, energy is also carried by the cross-terms that exist in the model. Analyzing the total incident or outgoing power can instead be done by integrating the intensity variables defined by the linearized potential flow model: `lpff.Ix`, `lpff.Iy`, and so on.

PORT PROPERTIES

Enter a unique **Port name**. Only nonnegative integer numbers can be used as **Port name** as it is used to define the elements of the S-parameter matrix. The numeric port names are also required for port sweep functionality. The port name is automatically incremented by one every time a port condition is added. Select the **Type of port** as **User defined** (the default), **Annular**, or **Circular**.

- For the **User defined** option, **Port Outgoing Mode Settings** and **Port Incident Mode Settings** sections will appear, as the incident and outgoing modes in general are different in the presence of flow.
- For the **Annular** and **Circular** options, the incident and outgoing modes are automatically computed. For these options, it is assumed that the background mean flow is uniform and that the waveguide walls are sound hard. Select the **Uniform outward normal mean flow** as **Average outward normal mean flow** (the default) or **User defined**. The first option will compute the uniform flow value as the average of the

Background mean flow velocity (on the boundary) defined either in the Linearized Potential Flow Model or by the Background Potential Flow Coupling multiphysics feature. For **User defined** enter the velocity u_{0n}, which has to be a constant and cannot be space dependent. Note that it is the outward velocity, relative to outward normal.

When using either the **Annular** or **Circular** option, remember to add the Port Reference Axis subfeature to define the reference direction for the azimuthal angle dependency of the port modes.

For the **Annular** option, if performing a parametric sweep over the port mode numbers m or n, only the **Parametric Sweep** can be used. The **Auxiliary sweep** option cannot be used, for this specific case.

PORT OUTGOING MODE SETTINGS

This section is present if **Type of port** is selected as **User defined**.

Select if the **Outgoing mode** is **On** (the default) or **Off**, letting you define if an outgoing mode is handled by the port and a scattering parameter solved for. For certain combinations of background flow, boundary conditions, and frequency a situation can occur when no outgoing propagating mode exists.

- Enter an expression for the **Outgoing mode shape** ϕ_n^{out} (SI unit: m^2/s).
- Enter an expression for the **Outgoing mode wave number** k_n^{out} (SI unit: rad/m).

PORT MODE SETTINGS

This section is present if **Type of port** is selected as **Annular** or **Circular**. For these two options the mode shapes are computed from semi-analytical expressions (see Ref. 24 and Ref. 25). The modes are computed in terms of the pressure mode shapes, such that they correspond to the standard inputs necessary for model source decomposition.

Enter the **Mode number, azimuthal** m and the **Mode number, radial** n, both integer values. In a 2D axisymmetric model the azimuthal mode number m is entered at the physics interface level, in the Linearized Potential Flow Equation Settings. In 3D models select the **Azimuthal angle dependency** as **Sine** or **Cosine**. The setting determines which of the two orthogonal mode representations that is used. In general, add two **Port** conditions, one for each angle dependency setting, in order to describe the acoustic field fully at a waveguide inlet or outlet.

For the **Annular** option, enter the **Tolerance** (default value `1e-8`) used when computing the shape of the analytical modes (used for solving the associated Bessel-like equation); the default value is in general a good choice.

PORT INCIDENT MODE SETTINGS

Select if the **Incident wave excitation at this port** is **On** or **Off** (the default), letting you define if an incident mode is exciting the system and handled by the port. In the same manner as for the outgoing mode, a situation can occur when no incident modes exist.

For the **User defined** option:

- Enter an expression for the **Incident mode shape** ϕ_n^{in} (SI unit: m^2/s).
- Enter an expression for the **Incident mode wave number** k_n^{in} (SI unit: rad/m).

The mode shape and the mode wave number can be computed using The Linearized Potential Flow, Boundary Mode Interface. The boundary mode study will typically produce a solution with many modes and mode wave numbers. The simplest way to use a specific computed mode in the port is to use the `withsol()` operator.

For example, to select the third eigenvalue mode shape and out-of-plane wave number, type:

- `withsol('sol1',phi,setind(lambda,3))`
- `withsol('sol1',lpfbm.kz,setind(lambda,3))`

where `'sol1'` refers to the solution of the boundary mode study. This tag is easily found looking at the datasets. It is also assumed that the dependent variable is here called `phi`, and that the tag to the boundary mode interface is `lpfbm`.

Optionally, if, for example, a parametric sweep is run over a given parameter, say a Mach number `M`, the `withsol()` operator can be called with the additional argument `setval(M,0.5)`, to retrieve the solution for the parameter `M` equal to `0.5`, or any other value solved for.

See, for example, the *Flow Duct — With Boundary Mode Analysis* application library model.

For all **Type of port** options, if **Incident wave excitation at this port** is set to **On**, then select the **Define incident wave** option as **Power** (the default), **Amplitude (pressure)**, **Amplitude (velocity potential)**, or **Mode scale**. These options decide how the factor A^{in}

that is multiplied with the incident mode shape ϕ_n^{in} is computed. Remember that the scaling of the incident mode shape is controlled from the Global Port Settings section.

- For **Power**, enter the power P^{in} (SI unit: W) associated with the incident field at the port, has the given value.
- For **Amplitude (pressure)**, enter the amplitude A_p^{in} (SI unit: Pa) associated with the incident pressure at the port, has the given value.
- For **Amplitude (velocity potential)**, enter the amplitude A_ϕ^{in} (SI unit: m^2/s) associated with the incident velocity potential at the port, has the given value.
- For **Mode scale**, enter the value S^{in} that is directly multiplied with the incident mode shape. This option is often most relevant when doing a modal decomposition of sources. Here each mode is multiplied with a given factor.

The **Phase** φ can also be entered. The phase contribution $\exp(i\varphi)$ is then multiplied with the amplitude factor A^{in}.

PORT GEOMETRY

The settings in this section ensure that the correct port area (including symmetries) is used for computing important quantities like power, volume flow, and transfer matrix elements, associated with the port. This section is only visible in 3D.

Select the **Port area** as **Use symmetries** (the default) or **Selected boundaries**.

- When **Use symmetries** is selected, symmetry conditions adjacent to the port will automatically be taken into account if the **Port area multiplication factor** is set to **Automatic** (the default); if set to **User defined**, enter the area multiplication factor A_{scale} manually.
- When **Selected boundaries** is selected, the port will have the area of the selected boundaries, without taking any symmetry conditions into account.

CONSTRAINT SETTINGS

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

- *Flow Duct*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct**
- *Flow Duct — With Boundary Mode Analysis*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct_boundary_mode**
- *Flow Duct — Modes with Impedance Condition*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct_impedance**

Port Reference Axis

Adding the **Port Reference Axis** feature is necessary when the port type is set to **Circular** or **Annular** in the Port feature. Select two points in the geometry to define the azimuthal reference axis for the port mode orientation.

Interior Impedance and Pair Impedance

Use the **Interior Impedance** or **Pair Impedance** node in a frequency domain analysis to define the specific transfer impedance at the boundary between two domains or parts in an assembly as the ratio of pressure drop to normal velocity $Z_n = \Delta p/(\mathbf{n}\cdot\mathbf{u})$. The formulation is following the so-called Ingard–Myers condition. The associated boundary condition is

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0\cdot\nabla\phi)\mathbf{u}_0\right)_{\text{up}} = \rho_0\left(\frac{p}{Z_n} + \frac{1}{i\omega}(\mathbf{u}_0\cdot\nabla)\frac{p}{Z_n}\right)_{\text{up}} - \rho_0\left(\frac{p}{Z_n} + \frac{1}{i\omega}(\mathbf{u}_0\cdot\nabla)\frac{p}{Z_n}\right)_{\text{down}}$$

$$-\mathbf{n}\cdot\left(\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0\cdot\nabla\phi)\mathbf{u}_0\right)_{\text{down}} = \rho_0\left(\frac{p}{Z_n} + \frac{1}{i\omega}(\mathbf{u}_0\cdot\nabla)\frac{p}{Z_n}\right)_{\text{up}} - \rho_0\left(\frac{p}{Z_n} + \frac{1}{i\omega}(\mathbf{u}_0\cdot\nabla)\frac{p}{Z_n}\right)_{\text{down}}$$

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

INTERIOR/PAIR IMPEDANCE (INGARD–MYERS)

Enter a **Specific transfer impedance** Z_n (SI unit: Pa·s/m).

Vortex Sheet

Use the **Vortex Sheet** boundary condition to model a shear layer that separates a stream from the free velocity field. The velocity potential is discontinuous over this boundary. Vortex sheets are only applicable on interior boundaries or between assemblies.

The equations defining the vortex sheet boundary condition are

$$\left[\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi+\mathbf{u}_0\cdot\nabla\phi)\right)\right]_i=[\rho_0(i\omega+\mathbf{u}_0\cdot\nabla)w]_i \qquad i = \text{up, down}$$

$$p_{up}=p_{down} \qquad w_{up}=-w_{down}$$

where w denotes the outward normal displacement (SI unit: m) of the vortex sheet surface. The condition ensures continuity in the normal stress (the pressure) and the normal displacement, while allowing for a slit (jump) in the tangential component of the total velocity.

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

This boundary condition adds an additional degree of freedom `lpff.vs1.w` (here using the default name for the physics interface and boundary condition) that is solved for.

Interior Sound Hard Boundary (Wall)

For The Linearized Potential Flow, Frequency Domain Interface and The Linearized Potential Flow, Transient Interface, use the **Interior Sound Hard Boundary (Wall)** condition to model interior rigid boundary surfaces, or walls. It prescribes a vanishing normal component of the particle velocity at the boundary. Multiplied by the density, it can equivalently be expressed as a *no-flow* condition:

$$\left[-\mathbf{n}\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t}+\mathbf{u}_0\cdot\nabla\phi\right)\right)\right]_i = 0 \qquad i = \text{up, down}$$

The up/down refers to the up and down side of an interior boundary. The sound-hard boundary condition is available for all analysis types. The equation above applies to the time-domain calculations in the Linearized Potential Flow, Transient interface; to obtain the corresponding condition for the frequency domain, replace $\partial/\partial t$ with $i\omega$.

Continuity

Continuity is available as an option at interfaces between parts in a pair. This condition gives continuity in the velocity potential as well as continuity in the mass flow. It corresponds to a situation where the boundary has no direct effect on the acoustic velocity potential field (the subscripts 1 and 2 in the equation refer to the two sides of the pair):

$$\mathbf{n}\cdot\left[\left(\rho_0\left(\nabla\phi-\frac{\mathbf{u}_0}{c_0^2}(i\omega+\mathbf{u}_0\cdot\nabla)\phi\right)\right)_1-\left(\rho_0\left(\nabla\phi-\frac{\mathbf{u}_0}{c_0^2}(i\omega+\mathbf{u}_0\cdot\nabla)\phi\right)\right)_2\right] = 0$$

CONSTRAINT SETTINGS

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

Mass Flow Edge Source

For 3D components, use a **Mass Flow Edge Source** to specify the mass flow rate on an edge:

$$-\frac{\rho_0}{c_0^2}i\omega(i\omega\phi+\mathbf{u}_0\cdot\nabla\phi)+\nabla\cdot\left(\rho_0\nabla\phi-\mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi+\mathbf{u}_0\cdot\nabla\phi)\right) = m'e^{i\theta}\delta(\mathbf{x}-\mathbf{x}_0)\mathrm{d}l$$

In the time domain, the $i\omega$ factors are replaced by partial time derivatives, $\partial/\partial t$.

MASS FLOW EDGE SOURCE

Enter a **Mass flow rate** m' (SI unit: kg/(m^2·s)) and enter a **Phase** θ (dimensionless).

Mass Flow Point Source

Add a **Mass Flow Point Source** node to specify the mass flow rate on a point:

$$-\frac{\rho_0}{c_0^2} i\omega(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi) + \nabla \cdot \left(\rho_0 \nabla\phi - \mathbf{u}_0 \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi)\right) = m' e^{i\theta} \delta(\mathbf{x} - \mathbf{x}_0)$$

In the time domain, the $i\omega$ factors are replaced by partial time derivatives, $\partial/\partial t$.

MASS FLOW POINT SOURCE

Enter a **Mass flow rate** m' (SI unit: kg/s for 3D and 2D axisymmetric components; kg/(m·s) for 2D components).

Enter a **Phase** θ (dimensionless).

Doppler Shift: Application Library path
Acoustics_Module/Aeroacoustics_and_Noise/doppler_shift

Mass Flow Circular Source

For 2D axisymmetric components, use a **Mass Flow Circular Source** node to add a circular source located at $\mathbf{x} = \mathbf{x}_0$:

$$-\frac{\rho_0}{c_0^2} i\omega(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi) + \nabla \cdot \left(\rho_0 \nabla\phi - \mathbf{u}_0 \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi)\right) = m' e^{i\theta} \delta(\mathbf{x} - \mathbf{x}_0) r \, d\varphi$$

In the time domain, factors $i\omega$ are replaced by partial time derivatives, $\partial/\partial t$.

MASS FLOW CIRCULAR SOURCE

Enter a **Mass flow rate** m' (SI unit: kg/(m·s) and enter a **Phase** θ (dimensionless).

Mass Flow Line Source on Axis

For 2D axisymmetric components, use a **Mass Flow Line Source on Axis** node to add a line source along the symmetry axis:

$$-\frac{\rho_0}{c_0^2} i\omega(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi) + \nabla \cdot \left(\rho_0 \nabla\phi - \mathbf{u}_0 \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi) \right) =$$

$$m' e^{i\theta} \delta(\mathbf{x} - \mathbf{x}_0) \mathrm{d}z$$

In the time domain, the $i\omega$ factors are replaced by partial time derivatives, $\partial/\partial t$.

MASS FLOW LINE SOURCE ON AXIS

Enter a **Mass flow rate** m' (SI unit: kg/(m·s)) and enter a **Phase** θ (dimensionless).

Axial Symmetry

The **Axial symmetry** feature is a default node added for all axisymmetric components. The boundary condition is active on all boundaries on the symmetry axis.

The Linearized Potential Flow, Transient Interface

The **Linearized Potential Flow, Transient (lpft)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations in the velocity potential in the presence of an inviscid and irrotational background mean flow, that is, a potential flow. The background flow can be modeled using The Compressible Potential Flow Interface. The physics interface is used for convected acoustic simulations that can be described by the linearized compressible potential flow equations.

The equations are formulated in the time domain. The physics interface is limited to flows with a Mach number Ma < 1, partly due to limitations in potential flow and partly due to the acoustic boundary settings needed for supersonic flow. The coupling between the acoustic field and the background flow does not include any predefined flow-induced noise.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Potential Flow Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**. For axisymmetric components, an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Potential Flow, Transient** to select physics features from the context menu.

SETTINGS

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

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

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

TRANSIENT SOLVER SETTINGS

Enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used

in the model. Select the **Time stepping** (method) as **Fixed (preferred)** the default and recommended or **Free**. The **Free** option is in general not recommended for wave problems. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

Details about Time Stepping in Transient Models are found in the Modeling with the Aeroacoustics Branch section.

The remainder of the settings are shared with The Linearized Potential Flow, Frequency Domain Interface.

- Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Transient Interface
- Theory Background for the Aeroacoustics Branch

Domain, Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Transient Interface

The Linearized Potential Flow, Transient Interface has these domain, boundary, edge, point, and pair nodes, listed in alphabetical order and described for The Linearized Potential Flow, Frequency Domain Interface. The nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- Continuity
- Initial Values
- Interior Sound Hard Boundary (Wall)
- Linearized Potential Flow Model
- Mass Flow Point Source
- Mass Flow Circular Source
- Mass Flow Edge Source
- Normal Mass Flow
- Periodic Condition
- Plane Wave Radiation
- Sound Hard Boundary (Wall)
- Velocity Potential

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Symmetry

Use the **Symmetry** condition on symmetry boundaries. Using symmetries in a model is important in order to reduce the model complexity.

The Linearized Potential Flow, Boundary Mode Interface

The **Linearized Potential Flow, Boundary Mode (lpfbm)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ) when adding a physics interface, is used to compute and identify propagating and nonpropagating modes in waveguides and ducts in the presence of a stationary background compressible potential mean-flow. The interface performs a boundary mode analysis on a boundary, inlet, or cross section of the waveguide. The interface exists in 3D and 2D axisymmetric geometries and applies to boundaries. This study is used, for example, when computing and specifying the modes at a Port condition in The Linearized Potential Flow, Frequency Domain Interface. The physics interface solves an eigenvalue equation on boundaries, searching for the out-of-plane wave numbers at a given frequency.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Linearized Potential Flow Model**, **Sound Hard Boundary (Wall)**, and **Initial Values**. For 2D axisymmetric models an **Axial Symmetry** node is also added.

Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Linearized Potential Flow, Boundary Mode** to select physics features from the context menu.

This physics interface is limited to flows with a Mach number $\mathrm{Ma} < 1$, partly due to limitations in the potential flow formulation and partly due to the acoustic boundary settings needed for supersonic flow.

SETTINGS

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

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

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

LINEARIZED POTENTIAL FLOW EQUATION SETTINGS

For 2D axisymmetric components, the **Azimuthal mode number** m (dimensionless) is 0 by default. It is an integer entering the axisymmetric expression for the velocity potential:

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

SOUND PRESSURE LEVEL SETTINGS

The settings are the same as Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface.

DEPENDENT VARIABLES

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

DISCRETIZATION

From the list, select the element order and type (Lagrange or serendipity). The default is **Quadratic Lagrange**.

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

	• Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Boundary Mode Interface • Theory Background for the Aeroacoustics Branch

	• *Flow Duct*: Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct** • *Jet Pipe*: Application Library Path **Acoustics_Module/Aeroacoustics_and_Noise/jet_pipe**

Boundary, Edge, Point, and Pair Nodes for the Linearized Potential Flow, Boundary Mode Interface

The Linearized Potential Flow, Boundary Mode Interface has these boundary, edge, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

The features marked with (*) in the list below are described for The Linearized Potential Flow, Frequency Domain Interface. The only difference for the Linearized Potential Flow, Boundary Mode interface is that you apply the features to boundaries instead of domains for 3D components, and to edges in 2D axisymmetry. In the Linearized Potential Flow Model you can enter a **Unit normal** that defines the positive propagation direction.

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using.

- Continuity*
- Impedance
- Initial Values*
- Linearized Potential Flow Model*
- Normal Mass Flow*
- Sound Hard Boundary (Wall)*
- Sound Soft Boundary*
- Symmetry
- Velocity Potential*

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Symmetry

Use the **Symmetry** condition to apply a symmetry for the boundary mode analysis. Remember that the presence of a symmetry condition restrict the solution space for the modes solved for.

Impedance

Use the **Impedance** condition to prescribe an impedance at a solid boundary according to the Ingard-Myer formulation. For the boundary mode interface, gradients in the out-of-plane direction(s) are implemented in a consistent manner using the solved for or prescribed wave number(s).

IMPEDANCE (INGARD-MYERS)

Enter a value or expression for the **Specific impedance** Z_n (SI unit: Pa·s/m). The default value is `lpfbm.rho0*lpfbm.c0` which corresponds to the specific characteristic impedance of the medium at rest.

The Compressible Potential Flow Interface

The **Compressible Potential Flow (cpf)** interface ( ), found under the **Acoustics** > **Aeroacoustics** branch ( ), is used to compute the velocity potential and density in a subsonic compressible potential flow model. Derived values include the associated pressure, velocity, and temperature in the flow. In a compressible potential flow model, the flow is assumed to be represented by an ideal barotropic, irrotational fluid at constant entropy, that is, the fluid is also inviscid. The flow can only have one unique origin or inlet as it uses a global total energy definition.

The physics interface is used for modeling the background mean flow used as input to The Linearized Potential Flow, Frequency Domain Interface or The Linearized Potential Flow, Transient Interface.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Compressible Potential Flow**, **Slip Velocity**, and **Initial Values**. For axisymmetric models an **Axial Symmetry** node is also added.

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

The potential flow formulation describes compressible and incompressible inviscid and irrotational flows (with no vorticity). However, for transonic and supersonic flows, shocks can occur, and then the flow often becomes rotational. For supersonic flow with shocks, the potential flow formulation is therefore often inaccurate.

SETTINGS

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

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

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

REFERENCE VALUES

Edit or enter the values as needed:

- **Reference pressure** p_{ref} (SI unit: Pa). The default is 1 atm.
- **Reference density** ρ_{ref} (SI unit: $\mathrm{kg/m^3}$). The default is 1.2 $\mathrm{kg/m^3}$.
- **Reference velocity** v_{ref} (SI unit: m/s).
- **Reference force potential** Ψ_{ref} (SI unit: J/kg).

DEPENDENT VARIABLES

This physics interface defines two dependent variables (field), the **Mean flow velocity potential** `Phi` and the **Density** `rho`. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

Select the discretization order for the **Mean flow velocity potential** and the **Density** dependent variables. The default is **Quadratic** for both.

- Domain, Boundary, and Pair Nodes for the Compressible Potential Flow Interface
- Theory Background for the Aeroacoustics Branch

Flow Duct: Application Library path
Acoustics_Module/Aeroacoustics_and_Noise/flow_duct

Domain, Boundary, and Pair Nodes for the Compressible Potential Flow Interface

The Compressible Potential Flow Interface has these domain, boundary, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows

users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

- Compressible Potential Flow Model
- Initial Values
- Interior Wall (Slip Velocity)
- Mass Flow
- Mean Flow Velocity Potential
- Normal Flow
- Periodic Condition
- Slip Velocity
- Symmetry

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

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Compressible Potential Flow Model

The **Compressible Potential Flow Model** node adds equations for time-dependent or stationary modeling of compressible potential flow. The model will solve for the mean flow velocity potential and the density. The pressure and the velocity field are defined from these dependent variables.

COMPRESSIBLE POTENTIAL FLOW MODEL

For the **Ratio of specific heats** γ (dimensionless) select **From material** (the default) or **User defined**. For the **User defined** option, the default value is 1.4.

Enter a **Force potential** ψ (SI unit: J/kg). To model a domain force or domain source acting on the fluid.

Select the **Calculate temperature** option it you need to also calculate the temperature field in the fluid. If this is selected, you also need to specify the **Specific gas constant** R_s (SI unit: J/(kg·K)). Select **From material** (the default) or **User defined**. Enabling this calculates the temperature according to

$$T = \frac{p_A}{\rho R_s}$$

where p_A is the pressure and ρ the density in the fluid.

Depending on the material selected in the **Materials** node for the fluid (typically Air) the **Model Inputs** may be active. Per default the **Absolute pressure** is taken from the model itself (unavailable). If the **Calculate temperature** option is enabled the **Temperature** is also taken from the model (unavailable). This will give a physically consistent solution. In both cases, you can click the **Make All Model Inputs Editable** icon to enter a user defined expression.

Note that the interface can be solved for in domains that have a PML defined. The PML does not have any influence in the solution, as it does not represent an acoustic perturbation. This means that the background flow solution can be used directly as input, also in PML domains, when solving a model with The Linearized Potential Flow, Frequency Domain Interface.

Initial Values

The **Initial Values** node adds initial values for the mean flow velocity potential and density variables. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter a value or expression for the initial values **Mean flow velocity potential** `Phi` (SI unit: m^2/s). Enter a **Density** `rho` (SI unit: kg/m^3); the default is the reference density `cpf.rhoref`.

Slip Velocity

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

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

Symmetry

The **Symmetry** condition is applied at planes where the model has symmetries. Using the intrinsic symmetries of a problem is a simple way to reduce the model size.

Normal Flow

The **Normal Flow** node implies that the flow is normal to the boundary and thus that the tangential velocity is zero. This corresponds to a constant velocity potential along the boundary. Because the velocity potential is determined only up to a constant, imposing this condition fixes the arbitrary constant to zero where $\phi = 0$.

Setting the **Normal Flow** condition on two or more disjoint boundaries can result in the wrong features unless symmetry implies that the velocity potential is equal on the boundaries in question. In general, only one unique flow origin or inlet can be set up.

Mass Flow

The **Mass Flow** node specifies the mass flow through the boundary. The mass flow is given by the product of the normal velocity v_n and the density at the boundary ρ_{bnd}. This results in a flux condition given by

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

MASS FLOW

Enter the **Normal velocity** v_n (SI unit: m/s) and **Fluid density at the boundary** ρ_{bnd} (SI unit: kg/m^3). The defaults are `cpf.vref` and `cpf.rhoref`, respectively. The values given in the Reference Values.

Mean Flow Velocity Potential

Use the **Mean Flow Velocity Potential** condition to prescribe the flow potential Φ at a boundary, $\Phi = \Phi_0$. This condition can be used to set up theoretical flow conditions at a boundary or couple to other physics interfaces.

MEAN FLOW VELOCITY POTENTIAL

Enter the Mean flow velocity potential Φ_0 (SI unit: m^2/s).

Periodic Condition

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

BOUNDARY SELECTION

The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODIC CONDITION

From the **Type of periodicity** list, select **Continuity** (default) or **Antiperiodicity**.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF DESTINATION

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog. For information about the **Orientation of Destination** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Mapping Between Source and Destination** section, see Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Interior Wall (Slip Velocity)

The **Interior Wall (Slip Velocity)** is used to set up interior walls on interior boundaries. This simplifies modeling and meshing steps as the wall does not need to have a thickness. The condition is simply applied to an interior boundary in the model. The condition enforces a vanishing mass flow on the up and down sides of the wall.

$$(-\mathbf{n} \cdot \rho \nabla \Phi)_{\text{up}} = 0 \qquad (-\mathbf{n} \cdot \rho \nabla \Phi)_{\text{down}} = 0$$

The dependent variables are defined as being discontinuous (so called slits) at this boundary.

Modeling with the Aeroacoustics Branch

In this section:

- Selecting an Aeroacoustics Interface
- Meshing
- Stabilization
- Solver Suggestions for Large Aeroacoustic Models
- Absorbing Layers for the Linearized Euler, Transient Interface
- Lagrange and Serendipity Shape Functions
- Time Stepping in Transient Models
- Mapping Between Fluid Flow and Acoustics Mesh
- Coupling to Turbulent Flows (Eddy Viscosity)
- Eigenfrequency Studies
- Suppressing Constraints on Lower Dimensions
- Postprocessing Variables

Selecting an Aeroacoustics Interface

When modeling a muffler with an internal nonisothermal flow, a jet engine, or a flow sensor you should consider which physics interface to use. The influence the background mean flow has on the acoustic behavior in an aeroacoustic model can be modeled in several ways. The effects that need to be included typically depend on the Mach number (Ma). A rule of thumb says that for a Mach number below 0.1 ($\mathrm{Ma} < 0.1$) the convective effects of the background flow need not to be included, above they do. Other considerations are of course also important, for example, whether viscous and thermal losses are important, if the background flow has large gradients, or if the flow is turbulent or has vorticity.

- Mach number less than 0.1 ($\mathrm{Ma} < 0.1$)

  In this situation the convective flow effects can normally be neglected and Pressure Acoustics can be used. Only the background temperature distribution $T = T(\mathbf{x})$ and background pressure distribution $p_A = p_A(\mathbf{x})$ need to be included. This can be done

directly in the Pressure Acoustics interface as a **Model Inputs**. The effects are included by making sure that the material properties depend on the local pressure and temperature; the spatial variations in the speed of sound is $c = c(p_A,T) = c(\mathbf{x})$ and in the density $\rho = \rho(p_A,T) = \rho(\mathbf{x})$, respectively.

- Mach number greater than 0.1 ($\mathrm{Ma} > 0.1$):

  In this case the convective effects of the flow can probably not be neglected. Modeling this type of system requires the use of one of the Aeroacoustics interfaces: linearized potential flow, linearized Euler, or linearized Navier–Stokes. The choice depends on the assumptions about the flow that can be made.

Note that when the background mean flow velocity $\mathbf{u}_0$ is set to zero, the linearized Navier–Stokes equations reduce to the thermoviscous acoustic equations. However, in thermoviscous acoustics the default discretization is P1, P2, P2 and no stabilization is applied.

Meshing

When solving a model using one of the Aeroacoustic interfaces, it is important to remember that this is a wave problem. This means that the wavelength should be resolved by an appropriate number of mesh elements. Using the same guidelines as for Pressure Acoustics is reasonable, that is, a first good mesh should use at least 5 elements per wavelength for second order shape functions. In the linearized Euler interface, the default shape functions are first order and here using at least 15 to 20 elements per wavelength is a first good choice.

Meshing guidelines for the Pressure Acoustics interface is given in Meshing: Resolving the Waves in Space under the Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces) section.

The Background Flow

Another important parameter to consider is resolving details in the background mean flow field. The acoustic mesh should capture gradients that exist in the background flow as these have a large influence on the acoustic propagation. Waves may, for example, be reflected and refracted in shear layers.

Meshing the Acoustic Boundary Layers in LNS

The Linearized Navier–Stokes interfaces, just as the Thermoviscous Acoustics interface, captures the physics of the acoustic boundary layer. When no-slip and isothermal conditions are used on walls, a viscous and a thermal boundary layer will exist. In order for the solution to be well behaved and capture losses correctly, it is important to mesh this layer, for example, by using a boundary layer mesh. If the model is large, using a single boundary layer mesh, with the approximate extend of the acoustic boundary layer, is the minimum requirement for a good solution.

Resolving Vorticity and Physics in LE and LNS

In the LE and LNS interfaces, the GLS stabilization is very efficient and can ensure smooth and converged solution even without resolving details like vorticity generation (the propagation of vorticity waves). If these are important processes in the model, the mesh should of course be able to resolve these details. Either refine the mesh or switch to (P2, P2, P2) discretization (keep the stabilization turned on). For LNS models, these phenomena are typically generated at walls where the no-slip condition generates vorticity when an acoustic wave is interacting with the background flow.

Stabilization

When solving the linearized Euler (LE) and linearized Navier–Stokes (LNS) equations using the finite element method (the standard Galerkin formulation), it can be shown that the method looses its good approximation characteristics when convective terms are present and when these terms locally dominate (Ref. 19). This can lead to spurious numerical oscillations. To remedy these oscillations stabilization is used.

It can be shown that if the cell Péclet number has a value larger than one $Pe^c > 1$ the solution will oscillate. In the linearized Navier–Stokes interface the variables `lnsf.CellPe_th` and `lnsf.CellPe_v` can be plotted to assess the characteristic number when comparing convection to viscous and thermal diffusion, respectively.

THE STABILIZATION METHODS

In the linearized Navier–Stokes (LNS) and linearized Euler (LE) interfaces two stabilization methods are implemented. The Galerkin least squares (GLS) and the streamline upwind Petrov–Galerkin (SUPG) stabilization. It is in general recommended to use the GLS method. The SUPG method is implemented for completeness and can be used by experienced users. There is also the option to not use

stabilization or to use the legacy method from version 5.2a and older. The legacy method for the LE interface is based on the implementation discussed in Ref. 10.

The implementation of the GLS stabilization methods follows the one discussed in Ref. 14, 19, 21, and 23. A general overview about stabilization methods can be found in Ref. 22. The GLS method combines stability and accuracy and the method order of accuracy is $O(h^{2p+1})$ where p is the shape function order and h is the local mesh size. The SUPG method has an accuracy order of $O(h^{p+1/2})$ for convection dominated problems and $O(h^{p+1})$ for diffusion dominated problems.

The default discretization for the LNS and LE interface is to use linear elements for all the dependent variables (P1-P1-P1). This effectively removes the stabilization on the diffusive parts of the equation. The GLS method is still superior to the SUPG method as it also stabilizes the reactive terms. These are the terms where gradients of the background mean flow fields enter. If no stabilization is used, then set a P1-P2-P2 discretization for the dependent variables as it ensures a more stable numerical scheme.

The equation residuals can be visualized in the LE and the LNS interfaces by plotting, for example, `lnsf.res_e` (equation residual for the energy equation), `lnsf.res_mx` (equation residual for the momentum equation x-component), `lnsf.res_my`, `lnsf.res_mz`, and `lnsf.res_e` (equation residual for the energy equation).

See for example the model *Point Source in 2D Jet: Radiation and refraction of sound waves through a 2D shear layer* found in the Application Gallery:
www.comsol.com/model/point-source-in-2d-jet-radiation-and-refraction-of-sound-waves-through-a-2d-shear-layer-16685

Solver Suggestions for Large Aeroacoustic Models

Studying aeroacoustic applications often lead to very large models that can contain many degrees of freedom (DOFs). Different steps can be taken to reduce the size of the models, for example, consider the use of slip instead of no-slip conditions in the LNS interface (avoiding the details in the acoustic boundary layer) or consider the choice between Lagrange and Serendipity Shape Functions.

When a model reaches a size where it can no longer fit in memory (when solved using the default direct solver) or if the solution procedure is slow with a direct solver, then it can be advantageous to switch to an iterative solver approach.

For the Linearized Navier–Stokes interfaces, iterative solver suggestions are automatically generated when the default solver is generated. These can be seen by expanding the **Solver Configuration** tree under **Stationary Solver** or **Time-Dependent Solver**. Per default a direct solver is used and two iterative solvers are suggested and disabled. To turn on one of these approached right-click the solver and select **Enable** (or press F4). The first suggestion *(GMRES with Direct Precon.)* uses an iterative solver with a direct preconditioner. This method is typically faster than the direct solver and uses 20% less memory. The second suggestion *(GMRES with DD)* uses an iterative solver with the domain decomposition method. This method is very robust (also for multiphysics applications) and very memory efficient, but it can be slow.

For the Linearized Euler interfaces the default solver is PARDISO, which is also the most robust option. Iterative solver will often struggle for the linearized Euler equations because there is not dissipation involved and instabilities are inherent.

Further details on solving large models can be found in the Solver Suggestions for Large Thermoviscous Acoustics Models section. These suggestions also apply to the linearized Navier–Stokes physics.

SOLVING IN THE ADIABATIC CASE

In certain cases, it is a good simplification to not include thermal conduction in the model and treat all processes as adiabatic (isentropic). This is, for example, relevant for fluids where the thermal boundary layer is much thinner than the viscous, like in water. Not solving for the temperature field T also saves some degrees of freedom (DOFs).

This is achieved by selecting the **Adiabatic formulation** option under the Linearized Navier–Stokes Equation Settings section. When **Adiabatic formulation** is selected, all temperature options and conditions are disabled in the user interface.

Absorbing Layers for the Linearized Euler, Transient Interface

The Linearized Euler, Transient Interface includes so-called *absorbing layers,* also often referred to as *sponge layers*. The layers work by combining three techniques: a scaling system, damping, and a simple nonreflecting condition.

The layers are set up by adding the **Absorbing Layer** under the **Definitions** node. This adds a special scaled system. The scaling effectively slows down the propagating waves and ensures that they "align up" (normal) toward the outer boundary. This means that the waves will hit the outer boundary in a closer to normal direction. Damping in the form of artificial numerical viscosity, is automatically applied in the layer in a matched manner (increasing from zero at the boundary to the physical domain). The artificial viscosity attenuates or damps high-frequency components of the wave in the layer. The amount of damping is controlled under the Damping Parameters for Absorbing Layers section at the main physics level. Finally, at the outer boundary of the layer add a simple Specific Acoustic Impedance (Isentropic) boundary condition, which will work well to remove all remaining waves, as normal incidence has been ensured.

The physical thickness of the layer should be such that it contains a minimum of 10 structured mesh elements. In practice, this means that it should roughly have a thickness equal to the largest wavelength resolved in the model.

For the **Absorbing Layers** select the **Type** (Cartesian, cylindrical, spherical, or user defined) under the **Geometry** section. Enter values for the **Physical Width** and **Pole Distance** under the **Scaling** section.

For more details on the **Geometry** and **Scaling** see the Infinite Elements, Perfectly Matched Layers, and Absorbing Layers section in the *COMSOL Multiphysics Reference Manual.*

Lagrange and Serendipity Shape Functions

In most of the physics interfaces in the Acoustics Module and specifically in the Aeroacoustics interfaces, you can choose between two families of shape functions in the **Discretization** section: *Lagrange* and *serendipity*. The current default is to use Lagrange shape functions.

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

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

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

In the *COMSOL Multiphysics Reference Manual*:

- The Lagrange Element (shlag)
- The Nodal Serendipity Element (shnserp)

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

Time Stepping in Transient Models

When solving transient wave problems, it is recommended to use a fixed time stepping in the solver (this is the manual option in the solver settings). In general, there is no point in using an automatic time-step control which can be provided by the time-dependent solver. The tolerances in the automatic error control are difficult to tune in wave problems when there is weak but important high-frequency content. It is recommended to use the suggestion generated when the **Fixed (preferred)** method is chosen (the default) in the **Transient Solver Settings** section. The internal time step of the solver that is generated when the **Maximal frequency to resolve** is set, will in most cases produce a solver that is adequate. It is assuming that the user has generated a mesh that properly resolves the same maximal frequency (minimal wavelength).

For further details see the Time Stepping: Resolving the Waves in Time section in Modeling with the Thermoviscous Acoustics Branch.

Mapping Between Fluid Flow and Acoustics Mesh

When the aeroacoustic and the Fluid Flow (CFD) model are solved on different computational mesh, careful mapping of the CFD solution from the CFD mesh onto the acoustics mesh should be done. This step is crucial in order not to introduce unphysical numerical noise into the acoustic solution (Ref. 20). If the solution is not

properly mapped, terms containing gradients of the background mean flow variables (especially the reactive terms) become very noisy and are typically the source of the numerical error. This mapping is handled by the multiphysics coupling Background Fluid Flow Coupling and the dedicated Mapping study.

If the acoustic problem is solved on the same mesh as the CFD, the mapping problem is less of an issue, especially if both physics use the same shape function order (they are both linear per default). Using the Background Fluid Flow Coupling is also necessary if perfectly matched layers (PMLs) are present in the model, in order to get the flow field inside the PML regions. In any case, the coupling can still be useful as the CFD solution will be smoothed by the mapping (the degree of smoothing can be controlled).

The multiphysics coupling of turbulent flow and acoustics is described in the application library model *Helmholtz Resonator with Flow: Interaction of Flow and Acoustics*. Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/helmholtz_resonator_with_flow**

MANUAL MAPPING

The multiphysics coupling Background Fluid Flow Coupling can couple all *Single-Phase Flow* and *Nonisothermal Flow* interfaces including the *High Mach Number Flow* interfaces. If such a coupling and mapping is necessary, manual mapping can be set up using the procedure described here.

The manual mapping is defined by a study that solves an additional set of equations which maps and smooths the background flow variables onto the acoustics mesh. The mapping equations can be set up using the **Weak Form PDE** interface from the **Mathematics** branch. In the most general case the mean background flow pressure p_0, velocity field $\mathbf{u}_0$, density ρ_0, temperature T_0, and turbulent viscosity μ_T variables should be mapped onto corresponding variables on the acoustics mesh, for example, $p_{0,aco}$, $\mathbf{u}_{0,aco}$, $\rho_{0,aco}$, $T_{0.aco}$, and $\mu_{T,aco}$. These new variables should then be used as the model inputs in the aeroacoustics model.

A simple but efficient mapping and smoothing is achieved by solving

$$p_{0,\,\mathrm{aco}} - p_0 = \delta h^2 \nabla \cdot (\nabla p_{0,\,\mathrm{aco}})$$
$$u_{i,\,0,\,\mathrm{aco}} - u_{i,\,0} = \delta h^2 \nabla \cdot (\nabla u_{i,\,0,\,\mathrm{aco}})$$
$$\rho_{0,\,\mathrm{aco}} - \rho_0 = \delta h^2 \nabla \cdot (\nabla \rho_{0,\,\mathrm{aco}})$$
$$T_{0,\,\mathrm{aco}} - T_0 = \delta h^2 \nabla \cdot (\nabla T_{0,\,\mathrm{aco}})$$
$$\mu_{T,\,\mathrm{aco}} - \mu_T = \delta h^2 \nabla \cdot (\nabla \mu_{T,\,\mathrm{aco}})$$

where the term on the right-hand side adds smoothing using isotropic diffusion. The amount of diffusion is controlled by the parameter δ (a constant that can be tuned with a typical value of 0.01) and the mesh size squared h^2. The term corresponds to so-called source term stabilization as known from CFD.

In the **Weak Form PDE** interface, define as many dependent variables as necessary and give them the same shape order as the order used for the acoustics (typically all linear). Assume that the source CFD variable for the x-velocity component is `u` and the new destination variable is `U0`. Then the above mapping is achieved with the following **Weak Expression**, entered in the user interface:

```
(U0-
withsol('sol1',u,setval(Ma,0.1)))*test(U0)+delta*h^2*(U0x*test(U0
x)+U0y*test(U0y)+U0z*test(U0z)
```

For the pressure, map the total pressure `spf.pA` to `P0` using:

```
(P0-
withsol('sol1',spf.pA,setval(Ma,0.1)))*test(P0)+delta*h^2*(P0x*te
st(P0x)+P0y*test(P0y)+P0z*test(P0z))
```

Notice the use of the `withsol()` operator, which is an extrusion-like coupling operator that can refer directly to a solution object and parameter value. In this example, it is used to fetch data from the CFD mesh, in the solution generated by solver `sol1` and for the parameter value `Ma` = `0.1`. The `setval()` statement is optional. It is important to use such an operator such that the CFD solution is mapped and interpolated correctly to the integration (Gauss) points on the acoustics mesh.

In the *COMSOL Multiphysics Reference Manual*:

- The Weak Form PDE
- Operators, Functions, and Constants and Built-In Operators for details about the `withsol()` operator
- Numerical Stabilization (general introduction)

In some cases it can be necessary to extend the mapping equation with boundary conditions for the mapped variables. This should be done at symmetry planes for the mapped velocity field. This is also the cases when flow details near walls are important, for example, to resolve the acoustic boundary layer. In this case, add a no-slip condition on $\mathbf{u}_{0,\mathrm{aco}}$. In general, constraining the value of the mapped variables to the value in the CFD model values at boundaries can be necessary.

Coupling to Turbulent Flows (Eddy Viscosity)

When acoustic waves propagate through a turbulent flow, they will experience attenuation due to the turbulence. It can be shown theoretically (see Ref. 18 and Ref. 19) that the experienced attenuation corresponds to extending the value of the dynamic viscosity μ to include the eddy viscosity μ_τ, such that the total effective viscosity is:

$$\mu_t = \mu + \mu_\tau$$

This effect can be included in the linearized Navier–Stokes model by using the effective dynamic viscosity as taken from a CFD model `spf.mu_eff`. This option is available in the Background Fluid Flow Coupling. This variable includes the dynamic viscosity and the turbulent viscosity (`spf.muT`). Notice, however, that the value of the turbulent viscosity can be much larger than the true eddy viscosity and that it depends on the turbulence model selected. This means that the experienced attenuation can be too large. Typically, more advanced turbulence models like the SST model give less numerical diffusion and thus a better estimate of the eddy viscosity.

Eigenfrequency Studies

Both the linearized Euler and the linearized Navier–Stokes interfaces have the Eigenfrequency study as a predefined study type. It should be noted that this analysis type is challenging when a nonzero background mean flow is present. Mathematically it can be very difficult to identify the pure acoustic eigenvalues.

In order to identify acoustic modes in the eigenvalue spectrum, it can be a good idea to use the **Region** option in the **Eigenfrequency search method** when setting up an eigenfrequency study step. In this way, the nonacoustic modes that consist of entropy and vorticity waves (eigenfrequencies with large imaginary components) can be filtered out. The pure acoustic modes typically have eigenfrequencies with a small complex component can then more easily be identified. The final identification needs to be done with a visual inspection of the mode shape in postprocessing. Set the search

region for the real part as desired and then set a small interval for the imaginary part, for example, from –0.02 to 0.02.

For more details see the Eigenfrequency section in the *COMSOL Multiphysics Reference Manual*

Suppressing Constraints on Lower Dimensions

Sometimes, boundary conditions on two adjacent objects can come into conflict on a shared object (point or edge). For the all the boundary conditions in the aeroacoustics physics that are based on constraints, you have the possibility to select that certain objects of lower dimensions should be excluded from the main selection. To do this, you must first click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. In the settings a new sections named **Excluded Edges** and/or **Excluded Points** will then appear (when applicable). In these sections, you can select geometrical objects which should be excluded from the main selection when the constraint is applied.

Kutta Like Conditions for Linearized Euler

A Kutta condition can be set up to handle edge singularities that appear, for example, at the trailing edge of an airfoil or at the transition from a liner (modeled with an Ingard-Myers type impedance) to a hard wall. By using **Exclude Edges** and **Exclude Points** the appropriate Kutta condition can be set up. The problematic edges or points are excluded from the adjacent conditions and extra conditions can be set up here using the user defined **Pointwise Constraint**.

Velocity Condition for Linearized Navier–Stokes

When using the mechanical Prescribed Velocity condition, in a linearized Navier–Stokes model, you may get conflicts if it is located next to a no-slip condition (a wall). To remedy this, you can exclude the common edge or point.

Periodic Condition

When using the **Periodic Condition**, you may get conflicts or duplicates which makes the model either overconstrained, or problematic for the automatic constraint elimination algorithm. If you are aware of such situations, it is good practice to remove one of the potentially conflicting constraints. This could be in multiphysics models where you combine solid mechanics with linearized Navier–Stokes acoustics and have edges where you both apply a **Periodic Condition** and the **Aeroacoustic–Structure Boundary** multiphysics coupling.

Particularly, if the geometry is curved, there is a risk that these constraints are not identical from a numerical point of view. In this case, excluding the conflicting edge from the selected boundary will make the behavior unique and fully predictable.

Another example where constraints will come in conflict is if you want to, for example, use periodic condition on parts of the geometry using weak constraints, while keeping the default pointwise constraints on other parts. If the same mesh node has both types of constraints, the solution will fail, so you must exclude any common geometrical objects from the selection in one of the constraints.

See also Excluded Surfaces, Excluded Edges, and Excluded Points in the *COMSOL Multiphysics Reference Manual*.

Postprocessing Variables

Intensity

The flow of energy is expressed by the acoustic intensity $\mathbf{I}$, which is formally defined by the time-averaged sound power per unit area (unit: $\mathrm{W/m^2}$). This quantity is available as a built-in postprocessing variable.

For a general fluid, including thermal and viscous losses (see Ref. 15 and Ref. 16 for details), the time-averaged intensity is given by

$$\mathbf{I} = \frac{1}{T}\int_0^T p\mathbf{u}\,\mathrm{d}t =$$

$$\frac{1}{2}\mathrm{Re}\left[(\rho_0\mathbf{u}+\rho\mathbf{u}_0)\left(\frac{p}{\rho_0}+\mathbf{u}_0\cdot\mathbf{u}\right)^* - \left[(\rho_0\mathbf{u}+\rho\mathbf{u}_0)^{\mathrm{T}}\cdot\left(\frac{1}{\rho_0}\tau-\frac{\rho}{\rho_0^2}\tau_0\right)^*\right]^{\mathrm{T}}\right.$$

$$\left. + \rho_0 T(s\mathbf{u}_0)^* + Tk\left[-\frac{\nabla T}{T_0}+\frac{T}{T_0^2}\nabla T_0\right]^*\right]$$

where the viscous stress tensors for the acoustic and background fields, τ and τ_0 respectively, are given by (in index notation)

$$\tau_{ij} = \mu\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) - \left(\mu_B - \frac{2}{3}\mu\right)\left(\frac{\partial u_k}{\partial x_k}\right)\delta_{ij}$$

$$\tau_{0ij} = \mu\left(\frac{\partial u_{0j}}{\partial x_i} + \frac{\partial u_{0i}}{\partial x_j}\right) - \frac{2}{3}\mu\left(\frac{\partial u_{0k}}{\partial x_k}\right)\delta_{ij}$$

It is understood that all dependent variables in these equations are the complex amplitudes.

In the time domain, the equivalent quantity is the instantaneous intensity $\mathbf{i}$ given by

$$\mathbf{i} = p\mathbf{v} = (\rho_0\mathbf{u} + \rho\mathbf{u}_0)\left(\frac{p}{\rho_0} + \mathbf{u}_0 \cdot \mathbf{u}\right) - \left[(\rho_0\mathbf{u} + \rho\mathbf{u}_0)^T \cdot \left(\frac{1}{\rho_0}\tau - \frac{\rho}{\rho_0^2}\tau_0\right)\right]^T + \rho_0 T s \mathbf{u}_0 + Tk\left[-\frac{\nabla T}{T_0} + \frac{T}{T_0^2}\nabla T_0\right]$$

where the dependent variables now include the explicit time dependence. The instantaneous expression is not defined in the frequency domain since it would represent effects happening at the double frequency.

See also the theory section The Energy Corollary for the Linearized Euler interface.

The intensity variables are defined for most Aeroacoustics physics where it makes sense. That is, the instantaneous quantities are defined in the time domain and the (time averaged) intensity variables are defined in the frequency domain. It is understood that the intensity vector is in general the time averaged quantity. For the Linearized Potential Flow interface, the variables are not defined in the time domain since it would require solving for an extra variable.

TABLE 5-1: INTENSITY VARIABLES IN 3D.

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ix	x-component of the intensity vector (frequency domain only)
phys_id.Iy	y-component of the intensity vector (frequency domain only)
phys_id.Iz	z-component of the intensity vector (frequency domain only)
phys_id.Ii_mag	Magnitude of the instantaneous intensity vector (time domain only)

TABLE 5-1: INTENSITY VARIABLES IN 3D.

VARIABLE	DESCRIPTION
phys_id.Iix	x-component of the instantaneous intensity vector (time domain only)
phys_id.Iiy	y-component of the instantaneous intensity vector (time domain only)
phys_id.Iiz	z-component of the instantaneous intensity vector (time domain only)

TABLE 5-2: INTENSITY VARIABLES IN 2D AXISYMMETRIC.

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ir	r-component of the intensity vector (frequency domain only)
phys_id.Iz	z-component of the intensity vector (frequency domain only)
phys_id.Ii_mag	Magnitude of the instantaneous intensity vector (time domain only)
phys_id.Iir	r-component of the instantaneous intensity vector (time domain only)
phys_id.Iiz	z-component of the instantaneous intensity vector (time domain only)

TABLE 5-3: INTENSITY VARIABLES IN 2D.

VARIABLE	DESCRIPTION
phys_id.I_mag	Magnitude of the intensity vector (frequency domain only)
phys_id.Ix	x-component of the intensity vector (frequency domain only)
phys_id.Iy	y-component of the intensity vector (frequency domain only)
phys_id.Ii_mag	Magnitude of the instantaneous intensity vector (time domain only)
phys_id.Iix	x-component of the instantaneous intensity vector (time domain only)
phys_id.Iiy	y-component of the instantaneous intensity vector (time domain only)

Theory Background for the Aeroacoustics Branch

The scientific field of *aeroacoustics* deals with the interaction between a background mean flow and an acoustic field propagating in this flow. In general, this concerns both the very complex description of the creation of sound by turbulence in the background flow, that is, flow induced noise and the influence the background mean flow has on the propagation of an externally created sound field, that is, *flow borne noise* or *convected acoustics*. The *computational aeroacoustics* (CAA) capabilities of the aeroacoustics interfaces in COMSOL Multiphysics only cover the convected acoustics situation.

Aeroacoustic simulations would ideally involve solving the fully compressible continuity, momentum (Navier–Stokes equations), and energy equations in the time domain. The acoustic pressure waves would then form a subset of the fluid solution. This approach is often impractical for real-world CAA applications due to the required computational time and memory resources. Instead, for solving many practical engineering problems, a decoupled two-step approach is used: first solve for the fluid flow, then the acoustic perturbations of the flow.

For solving the acoustic problem, the governing equations are linearized around the background mean flow and only solved for the acoustic perturbation. Acoustic variables are assumed to be small and perturbation theory can be used, for example, the total pressure

$$p_{\text{tot}} = p_0 + p$$

is the sum of the background mean pressure p_0 and the acoustic pressure variations p (sometimes labeled p' or p_1).

This section presents the basic mathematical framework for the aeroacoustic equations solved in the aeroacoustic interfaces, starting with the general governing equations for fluid flow, that is, conservation equations, constitutive equations, and equations of state. Then the linearized Navier–Stokes equations, the linearized Euler equations, the linearized potential flow equations, and finally the equations for the compressional potential flow are presented.

In this section the theory background for:

- General Governing Equations
- Linearized Navier–Stokes
- Linearized Euler
- Scattered Field Formulation for LE and LNS
- Linearized Potential Flow
- Compressible Potential Flow

General Governing Equations

The equations governing the physics in any fluid are the foundation for deriving the linearized aeroacoustic equations and in general any acoustics equations. The governing equations for the motion of a compressible fluid are the continuity equation (mass conservation), the Navier–Stokes equation (momentum conservation), and the general heat transfer equation (energy conservation). In order to close the system of equations, constitutive equations are needed, along with the equation of state, and thermodynamic relations. See, for example, Ref. 3, Ref. 4, Ref. 5, Ref. 6, or Ref. 7 for details and further reading.

CONSERVATION EQUATIONS

The conservation equations are for mass, momentum, and energy:

$$\begin{aligned} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= M \\ \rho \frac{D\mathbf{u}}{Dt} = \rho\left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u}\right) &= \nabla \cdot \sigma + \mathbf{F} \\ \rho T \frac{Ds}{Dt} = \rho T\left(\frac{\partial s}{\partial t} + (\mathbf{u} \cdot \nabla)s\right) &= -\nabla \cdot \mathbf{q} + \Phi + Q \end{aligned} \tag{5-5}$$

where $\mathbf{u}$ is the velocity field, ρ is the density, T is the temperature, s is the specific entropy, σ is the stress tensor, $\mathbf{q}$ is the local heat flux, Φ is the viscous dissipation function, and M, $\mathbf{F}$, and Q are source terms. The operator D/Dt is the material derivative (or advection operator) defined as

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + (\mathbf{u} \cdot \nabla)$$

THERMODYNAMIC RELATIONS

Some thermodynamic relations are necessary when reformulating the energy equations in terms of other sets of thermodynamic variables, like (p, T) or (ρ, p). They are the density differential, the specific energy relation, a relation due to Helmholtz, and the fundamental entropy relation:

$$\frac{d\rho}{\rho} = -\alpha_p dT + \beta_T dp$$

$$du = \left(\frac{\partial u}{\partial T}\right)_\rho + \left(\frac{\partial u}{\partial \rho}\right)_T$$

$$\left(\frac{\partial u}{\partial \rho}\right)_T = \frac{1}{\rho^2}\left(p - \frac{\alpha_p}{\beta_T} T\right)$$

$$ds = \frac{1}{T} du + \frac{p}{T} d(\rho^{-1})$$

where u is the specific internal energy, α_p is the coefficient of thermal expansion (isobaric), and β_T is the isothermal compressibility. See, for example, Ref. 5 and Ref. 6 for details. They are defined together with the specific heat at constant pressure C_p and specific heat at constant volume C_v as

$$\alpha_p = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p \qquad \beta_T = \frac{1}{\rho}\left(\frac{\partial \rho}{\partial p}\right)_T$$

$$C_v = \left(\frac{\partial u}{\partial T}\right)_\rho \qquad C_p = \left(\frac{\partial u}{\partial T}\right)_p$$

Using the above thermodynamic relations, the entropy differential can be expressed as (used for the linearized Navier–Stokes equations)

$$\rho T ds = \rho C_p dT - \alpha_p T dp$$

while for an ideal gas it can be given as (used for the linearized Euler equations)

$$ds = C_v \frac{dp}{\rho} - C_p \frac{d\rho}{p}$$

CONSTITUTIVE EQUATIONS

The constitutive equations are the equations of state (density expressed in terms of any set of thermodynamic variables), the Stokes expression for the stress tensor, and the Fourier heat conduction law

$$\sigma = -p\mathbf{I} + \tau = -p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) + \left(\mu_{\mathrm{B}} - \frac{2}{3}\mu\right)(\nabla \cdot \mathbf{u})\mathbf{I}$$

$$\mathbf{q} = -\mathrm{k}\nabla T$$

where k is the thermal conduction, μ is the dynamic viscosity, and μ_{B} is the bulk viscosity. This then also defines the viscous dissipation function

$$\Phi = \nabla\mathbf{u}{:}\tau$$

PERTURBATION THEORY

In the following, the governing equations are linearized and expanded to first order in the small parameters around the average stationary background solution. For details about perturbation theory see Ref. 3, Ref. 4, and Ref. 8. The small parameter variables (1'st order) represent the acoustic variations on top of the stationary background mean (or average) flow (0'th order solution). Note that when solving the equations, the value of the acoustic field variables can also represent nonacoustic waves like thermal waves (entropy waves) and vorticity waves. In the time domain, these can be linear instabilities and can actually represent the onset of turbulence.

The dependent variables and sources are expanded according to

$$A = A_0(\mathbf{x}) + A_1(\mathbf{x}, t)$$

where A is any of the dependent variables or sources. In the frequency domain, the first order variables are assumed to be harmonic and expanded into Fourier components, such that

$$A = A_0(\mathbf{x}) + A_1(\mathbf{x})e^{i\omega t}$$

The first order variation to material parameters that are not treated as dependent variables like the density ρ in the Linearized Navier–Stokes interface, is expressed using the density differential and related to perturbations in the temperature and the pressure.

The above perturbation schemes are inserted into the governing equations and the linearized acoustic equations are derived retaining only first order linear terms.

Linearized Navier–Stokes

The linearized Navier–Stokes equations are derived by linearizing the full set of fluid flow equations given in General Governing Equations. After some manipulation, the continuity, momentum, and energy equations and are:

$$\begin{aligned}
&\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u} + \rho \mathbf{u}_0) = M \\
&\rho_0\left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u}_0 + (\mathbf{u}_0 \cdot \nabla)\mathbf{u}\right) + \rho(\mathbf{u}_0 \cdot \nabla)\mathbf{u}_0 = \nabla \cdot \sigma + \mathbf{F} - \mathbf{u}_0 M \\
&\rho_0 C_\mathrm{p}\left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T_0 + \mathbf{u}_0 \cdot \nabla T\right) + \rho C_\mathrm{p}(\mathbf{u}_0 \cdot \nabla T_0) \\
&-\alpha_\mathrm{p} T_0\left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p_0 + \mathbf{u}_0 \cdot \nabla p\right) - \alpha_\mathrm{p} T(\mathbf{u}_0 \cdot \nabla p_0) = \nabla \cdot (\mathrm{k}\nabla T) + \Phi + Q
\end{aligned} \tag{5-6}$$

where p, $\mathbf{u}$, and T are the acoustic perturbations to the pressure, velocity, and temperature, respectively. In the frequency domain, the time derivatives of the dependent variables are replaced by multiplication with $i\omega$. The stress tensor is σ and Φ is the viscous dissipation function. The variables with a zero subscript are the background mean flow values and the subscript "1" is dropped on the acoustic variables.

The constitutive equations are the stress tensor and the linearized equation of state, while the Fourier heat conduction law is readily included in the above energy equation,

$$\sigma = -p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^\mathrm{T}) + \left(\mu_\mathrm{B} - \frac{2}{3}\mu\right)(\nabla \cdot \mathbf{u})\mathbf{I}$$

$$\rho = \rho_0(\beta_\mathrm{T} p - \alpha_\mathrm{p} T)$$

The linearized viscous dissipation function is defined as

$$\Phi = \nabla \mathbf{u} : \tau(\mathbf{u}_0) + \nabla \mathbf{u}_0 : \tau(\mathbf{u})$$

$$\tau(\mathbf{u}) = \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^\mathrm{T}) + \left(\mu_\mathrm{B} - \frac{2}{3}\mu\right)(\nabla \cdot \mathbf{u})\mathbf{I}$$

$$\tau(\mathbf{u}_0) = \mu(\nabla \mathbf{u}_0 + (\nabla \mathbf{u}_0)^\mathrm{T}) + \left(\mu_\mathrm{B} - \frac{2}{3}\mu\right)(\nabla \cdot \mathbf{u}_0)\mathbf{I}$$

The terms in the governing equations presented in Equation 5-6 can be divided into four categories. The time derivative (or frequency dependent) term, convective terms like $\mathbf{u}_0 \cdot \nabla p$, reactive terms like $\mathbf{u} \cdot \nabla p_0$, diffusive terms, and source terms. In many aeroacoustic formulations, the reactive terms are removed (or simplified) from the

governing equations in order to avoid the Kelvin-Helmholtz instabilities. This is sometimes referred to as gradient term stabilization (GTS).

Linearized Euler

The linearized Euler equations are derived from Euler's equations, that is Equation 5-5 with no thermal conduction and no viscous losses. The fluid in the Linearized Euler interface is assumed to be an ideal gas. The energy equation is also often written in terms of the pressure. This manipulation is possible using the thermodynamic differential for the entropy valid for an ideal gas. This is the common approach in the literature. A review of the linearized Euler equations is found in, for example, Ref. 12 and Ref. 13.

GOVERNING EQUATIONS

A linearization of the governing equations yields after some manipulation

$$\begin{aligned}
&\frac{\partial \rho_t}{\partial t} + \nabla \cdot (\rho_t \mathbf{u}_0 + \rho_0 \mathbf{u}_t) = M \\
&\frac{\partial \mathbf{u}_t}{\partial t} + \left(\left[\mathbf{u}_t + \frac{\rho_t}{\rho_0}\mathbf{u}_0\right] \cdot \nabla\right)\mathbf{u}_0 + (\mathbf{u}_0 \cdot \nabla)\mathbf{u}_t + \frac{1}{\rho_0}\nabla p_t = \frac{1}{\rho_0}(\mathbf{F} - \mathbf{u}_0 M) \\
&\frac{\partial p_t}{\partial t} + \mathbf{u}_t \cdot \nabla p_0 + p_t \gamma(\nabla \cdot \mathbf{u}_0) + \mathbf{u}_0 \cdot \nabla p_t + \gamma p_0(\nabla \cdot \mathbf{u}_t) = (\gamma - 1)Q + \frac{\gamma p_0}{\rho_0} M
\end{aligned} \tag{5-7}$$

Here, the subscript "1" has been dropped on the acoustic perturbation variables. The time derivatives are replaced with multiplication by $i\omega$ in the frequency domain. And source terms include a mass source M, a volume force $\mathbf{F}$, and a heat source Q.

Some ideal gas relations of interest are the equation of state, specific heat capacity and specific gas constant relations, and the isothermal compressibility and isobaric coefficients of thermal expansion:

$$c_0^2 = \frac{\gamma p_0}{\rho_0} = R_s \gamma T_0$$

$$\gamma = \frac{C_p}{C_v} \qquad C_p = C_v + R_s$$

$$\alpha_p = \frac{1}{T_0} \qquad \beta_T = \frac{1}{p_0}$$

It also follows from the governing equations and the thermodynamic relations that the acoustic variations in the specific entropy s and in the temperature T are given by

$$s = \frac{C_v}{p_0} p - \frac{C_p}{\rho_0} \rho$$

$$T = T_0 \left(\frac{p}{p_0} - \frac{\rho}{\rho_0} \right)$$

The specific heat capacity at constant volume and constant pressure are often labeled with a lowercase c. Here we use uppercase C's but they are the specific quantities.

INSTABILITIES

When the linearized Euler (LE) equations are solved in the time domain (or in the frequency domain with an iterative solver), linear physical instability waves can develop, the so-called Kelvin-Helmholtz instabilities. They are instabilities that grow exponentially because no losses exist in the LE equations (no viscous dissipation and no heat conduction). Furthermore, they are limited by nonlinearities in the full Navier–Stokes flow equations. It has been shown that in certain cases, the growth of these instabilities can be limited, while the acoustic solution is retained, by canceling terms involving gradients of the mean flow quantities. This is known as gradient terms suppression (GTS) stabilization. See more details in Ref. 9, Ref. 10, and Ref. 11.

Gradient terms suppression (GTS) stabilization is available for the linearized Euler equations. The reactive terms involving gradients of the background flow can be disabled individually. These terms are responsible for the generation of the instabilities. It is also possible to disable all convective terms if desired.

THE ENERGY COROLLARY

Expressions for the energy flux, that is, the acoustic intensity vector, are often referred to as Myers' energy corollary, see Ref. 15 and Ref. 16. The instantaneous intensity vector $\mathbf{I}_i$ is defined for both transient and frequency domain models as

$$\mathbf{I}_i = (\rho_0 \mathbf{u} + \rho \mathbf{u}_0) \left(\frac{p}{\rho_0} + \mathbf{u}_0 \cdot \mathbf{u} \right) + \rho_0 \mathbf{u}_0 T s$$

The (time averaged) intensity vector $\mathbf{I}$ is given in the frequency domain by

$$\mathbf{I} = \frac{1}{2}\mathrm{Re}\left((\rho_0\mathbf{u} + \rho\mathbf{u}_0)\left(\frac{p}{\rho_0} + \mathbf{u}_0 \cdot \mathbf{u}\right)^* + \rho_0\mathbf{u}_0 Ts^*\right)$$

ABOUT THE IMPEDANCE BOUNDARY CONDITION

The Linearized Euler, Frequency Domain Interface and The Linearized Euler, Transient Interface have an Impedance physics feature and its theory is included here.

In the frequency domain the Ingard-Myers equation (Ref. 1) gives an expression for the normal velocity at a boundary with a normal impedance condition. It is a so-called low-frequency approximation condition in the limit of very thin flow boundary layers (compared to the wavelength). Such conditions are used, for example, for porous lining conditions in ducts (Ref. 2). The condition is given by:

$$\mathbf{u} \cdot \mathbf{n} = \frac{p}{Z_\mathrm{n}} + \frac{1}{i\omega}\mathbf{u}_0 \cdot \nabla\left(\frac{p}{Z_\mathrm{n}}\right) - \frac{p}{i\omega Z_\mathrm{n}}\mathbf{n} \cdot (\mathbf{n} \cdot \nabla\mathbf{u}_0) \tag{5-8}$$

where the surface normal $\mathbf{n}$ here points out of the domain and is the surface normal impedance.

If the flow is parallel to the impedance boundary condition $\mathbf{u}_0 \cdot \mathbf{n} = 0$, for example, slip flow over a mechanical impedance boundary condition (the same is true for the moving wall boundary condition described below), one can use a formulation with the tangential derivative ($\nabla_{||}$) for the second term on the right-hand side:

$$\mathbf{u}_0 \cdot \nabla A = \mathbf{u}_0 \cdot \nabla_{||} A + \mathbf{u}_0 \cdot \mathbf{n}(\mathbf{n} \cdot \nabla A) = \mathbf{u}_0 \cdot \nabla_{||} A$$

where A is an arbitrary scalar.

The last term on the right-hand side of Equation 5-8 can be reformulated as follows:

$$\mathbf{n} \cdot (\mathbf{n} \cdot \nabla\mathbf{u}_0) = (\mathbf{n} \cdot \nabla)(\mathbf{n} \cdot \mathbf{u}_0) - \mathbf{u}_0 \cdot ((\mathbf{n} \cdot \nabla)\mathbf{n})$$

Again, these terms reduce significantly for the case where $\mathbf{u}_0 \cdot \mathbf{n} = 0$. If the boundary does not have curvature (planar boundary) then it is equal to zero. If the boundary is planar and the impedance condition is used inside the flow, for example at an outflow condition, then it reduces to the normal gradient of the velocity normal to the surface.

ABOUT THE MOVING WALL BOUNDARY CONDITION

The Linearized Euler, Frequency Domain Interface and The Linearized Euler, Transient Interface have a Moving Wall physics feature and its theory is included here.

Myers' equation (Ref. 1) gives the expressions used for a boundary condition at a moving wall. In the frequency domain, it is given by

$$\mathbf{u}\cdot\mathbf{n} = -i\omega v_{\mathrm{n}} - \mathbf{u}_0\cdot\nabla(v_{\mathrm{n}}) + v_{\mathrm{n}}\mathbf{n}\cdot(\mathbf{n}\cdot\nabla\mathbf{u}_0)$$

The inward normal displacement is given by

$$v_{\mathrm{n}} = -n\cdot v_{\mathrm{n}} = (i\omega)^{-1}u_n \qquad u_n = -n\cdot u_{\mathrm{b}}$$

where u_n is the inward normal velocity is.

In the time domain, the condition on the normal velocity is given by

$$\mathbf{u}\cdot\mathbf{n} = -\frac{\partial v_{\mathrm{n}}}{\partial t} - \mathbf{u}_0\cdot\nabla(v_{\mathrm{n}}) + v_{\mathrm{n}}\mathbf{n}\cdot(\mathbf{n}\cdot\nabla\mathbf{u}_0)$$

where the inward normal displacement is given by $v_n = -\mathbf{n}\cdot\mathbf{v}$ (SI unit: m).

Scattered Field Formulation for LE and LNS

The governing equations, given above in Equation 5-7 and Equation 5-6, are defined in the general scattered field formulation. Assuming that all the acoustic fields are the sum of a background (prescribed) field and the resulting scattered field, the total fields can be written as

$$p_{\mathrm{t}} = p_{\mathrm{s}} + p_{\mathrm{b}} \qquad \mathbf{u}_{\mathrm{t}} = \mathbf{u}_{\mathrm{s}} + \mathbf{u}_{\mathrm{b}} \qquad T_{\mathrm{t}} = T_{\mathrm{s}} + T_{\mathrm{b}} \qquad \rho_{\mathrm{t}} = \rho_{\mathrm{s}} + \rho_{\mathrm{b}}$$
$$p \equiv p_{\mathrm{s}} \qquad \mathbf{u} \equiv \mathbf{u}_{\mathrm{s}} \qquad T \equiv T_{\mathrm{s}} \qquad \rho \equiv \rho_{\mathrm{s}}$$

where subscript "t" stands for total, subscript "s" for scattered, and subscript "b" for background. The scattered field is the field solved for, that is the dependent variables p, $\mathbf{u}$, T, and ρ (depending on the interface). If no background acoustic field is defined (the default), the scattered field is equal to the total field. The governing equations and all boundary conditions are expressed in terms of the total fields. The background acoustic fields are defined by adding the Background Acoustic Fields feature in the LE interface or the Background Acoustic Fields feature in the LNS interface. The features have a user defined option where all fields can be entered, either as analytical expressions or defined in terms of a solution to another study or model.

Linearized Potential Flow

The equations presented here are the linearized potential flow equations. This restricts the applications of the physics interface to systems where the background flow is well described by a compressible potential flow, that is, a flow that is inviscid, barotropic, and irrotational. The sound sources also need to be external to the flow or at least they need to be represented by simple well defined sources. Application areas typically include modeling of how jet engine noise is influenced by the mean flow.

The basic dependent variable is the velocity potential ϕ conventionally defined by the relationship

$$\mathbf{u} = \nabla\phi$$

where $\mathbf{u} = \mathbf{u}(\mathbf{r}, t)$ is the particle velocity associated with the acoustic wave motion. The *total* particle velocity is given by

$$\mathbf{u}_{\text{tot}}(\mathbf{r}, \mathbf{t}) = \mathbf{u}_0(\mathbf{x}) + \mathbf{u}(\mathbf{r}, t) \tag{5-9}$$

where $\mathbf{u}_0$ denotes the local background mean flow velocity and $\mathbf{u}$ is the acoustic velocity perturbation. In the following the variables with a zero subscript are the background mean flow values. The acoustic perturbation quantities have no super nor subscript. In acoustic textbooks a subscript "1" or a prime (superscript) is sometimes used for the acoustic quantities.

The dynamic equations for this mean-flow field are described in the next subsection on the Compressible Potential Flow. For now, just assume $\mathbf{u}_0$ 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 $\mathbf{u}_0 = \nabla\Phi$.

The linearized equation for the velocity potential ϕ, governing acoustic waves in a background flow with mean background velocity $\mathbf{u}_0$, mean background density ρ_0, and mean background speed of sound c_0, is

$$-\frac{\rho_0}{c_0^2}\frac{\partial}{\partial t}\left(\frac{\partial\phi}{\partial t} + \mathbf{u}_0 \cdot \nabla\phi\right) + \nabla \cdot \left[\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t} + \mathbf{u}_0 \cdot \nabla\phi\right)\mathbf{u}_0\right] = 0 \tag{5-10}$$

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 5-9. Then, linearizing the resulting equations in the acoustic perturbation and eliminating all acoustic variables except the velocity potential gives Equation 5-10.

Thus, the density ρ in this equation is the time-independent part. The corresponding acoustic part is $\rho(\mathbf{r}, t) = p(\mathbf{r}, t)/{c_0}^2$ where p is the acoustic pressure, given by

$$p(\mathbf{r}, t) = -\rho_0\left(\frac{\partial \phi}{\partial t} + \mathbf{u}_0 \cdot \nabla\phi\right)$$

Hence, once Equation 5-10 has been solved for the velocity potential, the acoustic pressure can easily be calculated.

When transformed to the frequency domain, the wave Equation 5-10 reads

$$-\frac{\rho_0}{c_0^2} i\omega(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi) + \nabla \cdot \left[\rho_0 \nabla\phi - \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi)\mathbf{u}_0\right] = 0$$

while the acoustic pressure is

$$p(\mathbf{r}) = -\rho_0(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi)$$

Typical boundary conditions include:

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

FREQUENCY DOMAIN EQUATIONS

In the frequency domain, the velocity potential ϕ is assumed to be a harmonic wave of the form

$$\phi(\mathbf{r}, t) = \phi(\mathbf{r})e^{i\omega t}$$

The governing frequency domain — or time-harmonic — equation is

$$-\frac{\rho_0}{c_0^2} i\omega(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi) + \nabla \cdot \left[\rho_0 \nabla\phi - \frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0 \cdot \nabla\phi)\mathbf{u}_0\right] = 0$$

In 2D, where

$$\phi(\mathbf{r}, t) = \phi(x, y)e^{i(\omega t - k_z z)}$$

the out-of-plane wave number k_z enters the equations when the ∇ operators are expanded:

$$-i\omega\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0\cdot\nabla\phi - ik_z u_z\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0\cdot\nabla\phi - ik_z u_z\phi)\right)$$
$$+ \rho_0 k_z^2\phi + ik_z u_z\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0\cdot\nabla\phi - ik_z u_z\phi) = 0$$

The default value of the out-of-plane wave number is 0, that is, no wave propagation perpendicular to the 2D plane. In a mode analysis the equations are solved for k_z.

For 2D axisymmetric components

$$\phi(\mathbf{r}, t) = \phi(r, z)e^{i(\omega t - m\varphi)}$$

the azimuthal wave number m similarly appears in the equation as a parameter:

$$-i\omega\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0\cdot\nabla\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{u}_0\frac{\rho_0}{c_0^2}(i\omega\phi + \mathbf{u}_0\cdot\nabla\phi)\right) + \rho_0\frac{m^2}{r^2}\phi = 0$$

The background velocity field $\mathbf{u}_0$ cannot have an azimuthal component because the flow is irrotational.

TIME-DEPENDENT EQUATION

In the time domain, the physics interface solves for the velocity potential ϕ with an arbitrary transient dependency. The following equation governs the acoustic waves in a mean potential flow:

$$-\frac{\rho_0}{c_0^2}\frac{\partial}{\partial t}\left(\frac{\partial\phi}{\partial t} + \mathbf{u}_0\cdot\nabla\phi\right) + \nabla\cdot\left[\rho_0\nabla\phi - \frac{\rho_0}{c_0^2}\left(\frac{\partial\phi}{\partial t} + \mathbf{u}_0\cdot\nabla\phi\right)\mathbf{u}_0\right] = 0 \qquad (5\text{-}11)$$

Here ρ_0 (SI unit: $\mathrm{kg/m^3}$) is the background mean flow density, $\mathbf{u}_0$ (SI unit: m/s) denotes the background mean flow velocity, and c_0 (SI unit: m/s) refers to the speed of sound. The software solves the equation for the velocity potential ϕ, with SI unit $\mathrm{m^2/s}$. The validity of this equation relies on the assumption that ρ_0, $\mathbf{u}_0$, and c_0 are approximately constant in time, while they can be functions of the spatial coordinates.

The background velocity field $\mathbf{u}_0$ cannot have an azimuthal component because the flow is irrotational.

BOUNDARY MODE ANALYSIS

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

$$-i\omega\frac{\rho_0}{c_0^2}(i\omega\phi + (\mathbf{u}_t\cdot\nabla\phi) - ik_z u_n\phi) + \nabla\cdot\left(\rho_0\nabla\phi - \mathbf{u}_t\frac{\rho_0}{c_0^2}(i\omega\phi + (\mathbf{u}_t\cdot\nabla\phi) - ik_z u_n\phi)\right) + \rho_0 k_z^2\phi + ik_z u_n\frac{\rho_0}{c_0^2}(i\omega\phi + (\mathbf{u}_0\cdot\nabla\phi) - ik_z u_n\phi) = 0 \tag{5-12}$$

for the eigenmodes, ϕ, and eigenvalues, $\lambda = -ik_z$, on a bounded two-dimensional domain, Ω, given well-posed edge conditions on $\partial\Omega$. In this equation, ϕ is the velocity potential, ρ_0 is the background mean flow density, c_0 is the speed of sound, ω is the angular frequency, and k_z is the out-of-plane wave number or propagation constant. Furthermore, $\mathbf{u}_t$ denotes the background mean velocity field in the tangential plane while u_n is the background mean velocity component in the normal direction.

Although the out-of-plane wave number is called k_z, the two-dimensional surface on which Equation 5-12 is defined does not necessarily have to be normal to the z-axis for 3D geometries.

Compressible Potential Flow

Consider a compressible and inviscid fluid in some domain Ω. The motion and state of the fluid is described by its velocity $\mathbf{V}$, density ρ, pressure p, and total energy per unit volume e. Its dynamics is governed by the Euler equations, expressing the conservation of mass, momentum, and energy (see General Governing Equations):

$$\begin{aligned}\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{V}) &= 0\\ \rho\left(\frac{\partial\mathbf{V}}{\partial t} + (\mathbf{V}\cdot\nabla)\mathbf{V}\right) + \nabla p &= \mathbf{F}\\ \frac{\partial e}{\partial t} + \nabla\cdot((e+p)\mathbf{V}) &= 0\end{aligned} \tag{5-13}$$

Here, a volume force $\mathbf{F}$ has been included on the right-hand side of the momentum equation, whereas a possible heat-source term on the right-hand side of the energy equation (the last one) has been set to zero.

To close this system of five equations with six unknowns, an equation of state is required. Here, this is taken to be the equation for an ideal barotropic fluid,

$$p = p_{\text{ref}}\left(\frac{\rho}{\rho_{\text{ref}}}\right)^{\gamma}$$

where $\gamma = C_p/C_V$ is the ratio of specific heats, C_p is the specific heat at constant pressure and C_V at constant volume, while p_{ref} and ρ_{ref} are reference quantities for the pressure and the density, respectively. An alternative form of the ideal-fluid state equation is

$$p = \rho(\gamma - 1)e$$

The assumption that the fluid is barotropic means that $p = p(\rho)$. Taking the total time derivative and using the chain rule, leads to the relation

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

where, using the equation of state,

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

defines the speed of sound in the ideal fluid.

Assuming the flow to be irrotational, there exists a *velocity potential* field Φ, such that $\mathbf{V} = \nabla\Phi$. If, in addition, the volume force is assumed to be given by $\mathbf{F} = -\rho\nabla\Psi$, where Ψ is referred to as the force potential, the second of Equation 5-13 can be integrated to yield the *Bernoulli equation*

$$\frac{\partial\Phi}{\partial t} + \left(\frac{1}{2}|\nabla\Phi|^2 + \frac{\gamma}{\gamma-1}\frac{p_{\text{ref}}}{\rho}\left(\frac{\rho}{\rho_{\text{ref}}}\right)^{\gamma} + \Psi\right) = \frac{1}{2}v_{\text{ref}}{}^2 + \frac{\gamma}{(\gamma-1)}\frac{p_{\text{ref}}}{\rho_{\text{ref}}} + \Psi_{\text{ref}}$$

In this equation, two additional reference quantities have entered: the velocity v_{ref}, and the force potential Ψ_{ref}, both valid at the same reference point as p_{ref} and ρ_{ref}. Note, in particular, that neither the pressure p, nor the energy per unit volume e, appears in this equation.

TIME DEPENDENT STUDY

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

$$\frac{\partial \Phi}{\partial t} + \left(\frac{1}{2}|\nabla\Phi|^2 + \frac{\gamma}{\gamma-1}\frac{p_{\text{ref}}}{\rho}\left(\frac{\rho}{\rho_{\text{ref}}}\right)^{\gamma} + \Psi\right) = \frac{1}{2}v_{\text{ref}}{}^2 + \frac{\gamma}{(\gamma-1)}\frac{p_{\text{ref}}}{\rho_{\text{ref}}} + \Psi_{\text{ref}}$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \nabla \Phi) = 0 \qquad c = \sqrt{\gamma \frac{p}{\rho}} \qquad \gamma \equiv C_p / C_V$$

where γ is the ratio of specific heats and Ψ denotes a possible domain force potential (SI unit: J/kg). In this equation, subscript **ref** signifies reference quantities that apply at a specific point or surface. Thus, p_{ref} is a reference pressure, ρ_{ref} is a reference density, v_{ref} is a reference velocity, and Ψ_{ref} is a reference force potential.

STATIONARY STUDY

In a stationary study, the same equation is used, but all time derivatives are set to zero, such that

$$\left(\frac{1}{2}|\nabla\Phi|^2 + \frac{\gamma}{\gamma-1}\frac{p_{\text{ref}}}{\rho}\left(\frac{\rho}{\rho_{\text{ref}}}\right)^{\gamma} + \Psi\right) = \frac{1}{2}v_{\text{ref}}{}^2 + \frac{\gamma}{(\gamma-1)}\frac{p_{\text{ref}}}{\rho_{\text{ref}}} + \Psi_{\text{ref}}$$

$$\nabla \cdot (\rho \nabla \Phi) = 0$$

References for the Aeroacoustics Branch Interfaces

1. M. K. Myers, "On the basic boundary condition in the presence of flow", *J. of Sound and Vibration*, vol. 71, p. 429, 1980.

2. W. Eversman, "The boundary condition at an impedance wall in a non-uniform duct with potential mean flow", *J. of Sound and Vibration*, vol. 246, p. 63, 2001.

3. D.T. Blackstock, *Fundamentals of Physical Acoustics,* John Wiley & Sons, 2000.

4. H. Bruus, *Theoretical Microfluidics*, Oxford University Press, 2010.

5. G.K. Bachelor, *An Introduction to Fluid Dynamics*, Cambridge University Press, 2000.

6. L. D. Landau and E. M. Lifshitz, *Fluid Mechanics, Course on Theoretical Physics Volume 6*, Butterworth-Heinemann, 2003.

7. B. Lautrup, *Physics of Continuous Matter, Exotic and Every Day Phenomena in the Macroscopic World,* 2nd ed., CRC Press, 2011.

8. C. M. Bender and S. A. Orszag, *Advanced Mathematical Methods for Scientists and Engineers: Asymptotic Methods and Perturbation Theory*, Springer, 1999.

9. C. Bogey, C. Bailly, and D. Juvé, "Computation of Flow Using Source Terms in Linearized Euler's Equations", *AIAA Journal*, vol. 40, p. 235, 2002.

10. P. P. Rao and P. J. Morris, "Use of Finite Element Methods in Frequency Domain Aeroacoustics", *AIAA Journal*, vol. 44, p. 1643, 2006.

11. A. Agarwal, P. J. Morris, and R. Mani, "Calculation of Sound Propagation in Nonuniform Flows: Suppression of Instability Waves", *AIAA Journal*, vol. 42, p.80, 2004.

12. C. Bailly and D. Juvé, "Numerical Solution of Acoustic Propagation Problems Using Linearized Euler Equations", *AIAA Journal*, vol. 38, p. 22, 2000.

13. C. W. Tam, "Computational Aeroacoustics: Issues and Methods", *AIAA Journal*, vol. 33, 1995.

14. G. Hauke and T. J. R. Hughers, "A comparative study of different sets of variables for solving compressible and incompressible flows", *Comput. Methods Appl. Mech. Engrg.*, vol. 153, pp. 1–44, 1998.

15. M. K. Myers, "An Exact Energy Corollary for Homentropic Flows", *J. Sound. Vib.*, vol. 109, pp. 277–284, 1986.

16. M. K. Myers, "Transport of energy disturbances in arbitrary steady flows", *J. Fluid Mech.*, vol. 226, pp. 383–400, 1991.

17. C. K. W. Tam and Z. Dong, "Radiation and Outflow Boundary Conditions for Direct Computation of Acoustic and Flow Disturbances in Nonuniform Mean Flow", J. *of Comp. Acoustics* 4, pp. 175–201, 1996.

18. L. Du, A. Holmberg, M. Karlsson, and M Åbom, "Sound amplification at a rectangular T-junction with merging mean flows", *J. Sound Vib.*, vol. 367, pp. 69–83 2016.

19. J. Gikadi, S. Föller, T. Sattelmayer, "Impact of turbulence on the prediction of linear aerodynamic interactions: Acoustic response of a turbulent shear layer", *J. Sound Vib.*, vol. 333, pp. 6548–6559, 2014.

20. S. Redonnet and G. Cunha, "An advanced hybrid method for the acoustic prediction", *Adv. Eng. Softw.*, vol. 88, pp. 30–52, 2015.

21. J. Gikadi, "Prediction of Acoustic Modes in Combustors using Linearized Navier–Stokes Equations in the Frequency Domain", PhD thesis, TUM, 2013.

22. O. C. Zienkiewicz, R. L. Taylor, and P. Nithiarasu, "The Finite Element Method for Fluid Dynamics", Butterworth–Heinemann, 7th edition, 2014.

23. G, Hauke, "Simple stabilizing matrices for the computation of compressible flows in primitive variables," *Comp. Methods. Appl. Mech. Engrg.*, vol. 190, pp. 6881–6893, 2001.

24. D. L. Sutliff, "Rotating Rake Turbofan Duct Mode Measurement system", NASA/TM–2005-213828

25. H. D. Meyer and E. Envia, "Aeroacoustic Analysis of Turbofan Noise Generation", NASA Contractor Report 4715, 1996.

26. S. W. Rienstra, Fundamentals of duct acoustics: notes of course "Progress in Simulation, Control and Reduction of Ventilation Noise" November 16-18, 2015, Von Karman Institute for Fluid Dynamics, VKI Lecture Series, 2016-02.

6

Thermoviscous Acoustics Interfaces

This chapter describes the physics and multiphysics interfaces found under the **Thermoviscous Acoustics** branch ( ).

- The Thermoviscous Acoustics, Frequency Domain Interface
- The Thermoviscous Acoustics, Transient Interface
- The Thermoviscous Acoustics, Boundary Mode Interface
- The Thermoviscous Acoustics, SLNS Approximation Interface
- The Acoustic–Thermoviscous Acoustic Interaction, Frequency Domain Interface
- The Thermoviscous Acoustic–Solid Interaction, Frequency Domain Interface
- The Thermoviscous Acoustic–Shell Interaction, Frequency Domain Interface
- The Thermoviscous Acoustic–Thermoelasticity Interaction Multiphysics Interfaces
- Modeling with the Thermoviscous Acoustics Branch
- Theory Background for the Thermoviscous Acoustics Branch

The Thermoviscous Acoustics, Frequency Domain Interface

The **Thermoviscous Acoustics, Frequency Domain (ta)** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, is used to compute the acoustic variations of pressure, velocity, and temperature. This physics interface is required to accurately model acoustics in geometries of small dimensions, often referred to as *microacoustics*. Near walls, viscous losses and thermal conduction become important because boundary layers exist. The thicknesses of these boundary layers are also known as the viscous and thermal penetration depth. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations. It is, for example, used when modeling the response of transducers like microphones, miniature loudspeakers, and receivers. Other applications include analyzing feedback in hearing aids and in mobile devices, or studying the damped vibrations of MEMS structures. At very small dimensions or at low ambient pressures, the transition to nonideal wall conditions can be treated with the dedicated Slip Wall and Interior Slip Wall conditions.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. The harmonic variation of all fields and sources is given by $e^{i\omega t}$ using the $+i\omega$ convention. Linear acoustics is assumed. Nonlinear effects can be included when modeling in the time domain using The Thermoviscous Acoustics, Transient Interface and the Nonlinear Thermoviscous Acoustics Contributions feature.

The equations defined by the *Thermoviscous Acoustics, Frequency Domain* interface are the linearized Navier–Stokes equations in quiescent background conditions solving the continuity, momentum, and energy equations (first order perturbation equations). Thermoviscous acoustics is also known as *viscothermal acoustics* or sometimes *thermoacoustics* (not to be confused with the field discussing heating and cooling using acoustics). Due to the detailed description necessary when modeling thermoviscous acoustics, the model simultaneously solves for the acoustic pressure p, the acoustic velocity variation $\mathbf{u}$ (particle velocity), and the acoustic temperature variations T. It is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

The *Thermoviscous Acoustics, Frequency Domain* interface is formulated in the so-called scattered field formulation where the total field (subscript t) is the sum of the scattered field (the field solved for, p, $\mathbf{u}$, and T) and a possible background acoustic field (subscript b), such that

$$p_t = p + p_b \qquad \mathbf{u}_t = \mathbf{u} + \mathbf{u}_b \qquad T_t = T + T_b$$

When no Background Acoustic Fields feature is present (the background field values are zero per default) the total field is simply the field solved for, $p_t = p$, $\mathbf{u}_t = \mathbf{u}$, and $T_t = T$. All governing equations and boundary conditions are formulated in the total field variables.

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

For good modeling strategies, solver suggestions, postprocessing information, as well as tips and tricks, see the Modeling with the Thermoviscous Acoustics Branch section.

On the Thermoviscous Acoustics Interface

The *Thermoviscous Acoustics, Frequency Domain* interface solves, as mentioned, the full linearized Navier–Stokes (momentum), continuity, and energy equations. It solves for the propagation of compressible linear waves in a general viscous and thermally conductive fluid. The length scale at which the thermoviscous acoustic description is necessary is given by the thickness of the viscous boundary layer (the viscous penetration depth), which is

$$\delta_v = \sqrt{\frac{\mu}{\pi f \rho_0}}$$

and the thickness of the thermal boundary layer (the thermal penetration depth)

$$\delta_{th} = \sqrt{\frac{k}{\pi f \rho_0 C_p}}$$

where the definition of the symbols f, μ, ρ_0, k, and C_p can be found in Table 6-1. The thickness of both boundary layers depends on the frequency f and decreases with

increasing frequency. The ratio of the two length scales is related to the nondimensional Prandtl number Pr, by

$$\frac{\delta_v}{\delta_{th}} = \sqrt{\frac{\mu C_p}{k}} = \sqrt{\mathrm{Pr}}$$

which define the relative importance of the thermal and viscous effects for a given material. In air at 20°C and 1 atm, the viscous boundary layer thickness is 0.22 mm at 100 Hz while it is only 55 µm in water under the same conditions. The Prandtl number is 0.7 in air and 7 in water.

Evaluate the value of the viscous and thermal boundary layer thickness as well as the Prandtl number in postprocessing. They are defined by the variables `ta.d_visc`, `ta.d_therm`, and `ta.Pr`, respectively.

The physical quantities commonly used in the thermoviscous acoustics interfaces are defined in Table 6-1 below.

TABLE 6-1: THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN PHYSICAL QUANTITIES.

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Pressure (acoustic)	p	pascal	Pa
Total acoustic pressure	p_t	pascal	Pa
Scattered acoustic pressure	p_s	pascal	Pa
Temperature variation (acoustic)	T	kelvin	K
Total temperature variation	T_t	kelvin	K
Scattered temperature variation	T_s	kelvin	K
Acoustic velocity field	$\mathbf{u} = (u, v, w)$	meter/second	m/s
Total acoustic velocity field	$\mathbf{u}_t$	meter/second	m/s
Scattered acoustic velocity field	$\mathbf{u}_s$	meter/second	m/s
Dynamic viscosity	μ	pascal·second	Pa·s
Bulk viscosity	μ_B	pascal·second	Pa·s
Thermal conductivity	k	watt/meter·kelvin	W/(m·K)
Heat capacity at constant pressure	C_p	joule/meter^3·kelvin	J/(m^3·K)

TABLE 6-1: THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN PHYSICAL QUANTITIES.

QUANTITY	SYMBOL	SI UNIT	ABBREVIATION
Isothermal compressibility	β_T	1/pascal	1/Pa
Coefficient of thermal expansion (isobaric)	α_p	1/kelvin	1/K
Ratio of specific heats	γ	(dimensionless)	1
Frequency	f	hertz	Hz
Wave number	k	1/meter	1/m
Equilibrium background pressure	p_0	pascal	Pa
Equilibrium background density	ρ_0	kilogram/meter^3	kg/m^3
Equilibrium background temperature	T_0	kelvin	K
Speed of sound	c	meter/second	m/s
Specific acoustic impedance	Z	pascal·second/meter	Pa·s/m
Acoustic impedance	Z_{ac}	pascal·second/meter^3	Pa·s/m^3

As the Thermoviscous Acoustics interface solves for pressure, velocity, and temperature, models can easily become large and contain many DOFs. See Solver Suggestions for Large Thermoviscous Acoustics Models for suggestions on how to solve large thermoviscous acoustic models.

SETTINGS

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

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

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

EQUATION

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

- For **Study controlled**, the frequency solved for is automatically taken from the solver; and the scaling of the equations is optimized for the numerical performance of the different solvers and study types.
- For **Frequency domain,** the **Frequency** is taken **From solver** (default) or can be entered when **User defined** is selected. You can also manually enter the scaling parameter Δ under the **Thermoviscous Acoustics Equation Settings** section.
- For **Mode analysis**, you can manually enter the **Mode analysis frequency** (applicable in 2D and 2D axisymmetry).

THERMOVISCOUS ACOUSTICS EQUATION SETTINGS

Adiabatic Formulation

Click to select **Adiabatic formulation** to use an adiabatic equation of state and disable the temperature degree of freedom for the thermoviscous acoustic equations. This formulation is applicable when the thermal losses can be disregarded; this is often the case in liquids, like water. In gases, like air, on the other hand the full formulation is necessary. When **Adiabatic formulation** is selected, all temperature conditions and options are disabled in the user interface.

Out-of-Plane Mode Extension

In 2D and 2D axisymmetric models, click to select **Out-of-plane mode extension** in order to model the third dimension implicitly. When selected, an additional degree of freedom, for the velocity, is solved for in the out-of-plane direction. The behavior is determined by entering the **Out-of-plane wave number** k_z in 2D models; or by entering the **Azimuthal mode number** m (sometimes referred to as the circumferential mode number) in 2D axisymmetric models. The latter defines an azimuthal wave number $k_m = m/r$. All out-of-plane gradients of the fields are then defined in terms of the wave number. The solved equations closely follow the ones used in the Formulation for the Boundary Mode Interface.

Scaling Factor

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

The default **Scaling factor** Δ is $1/(i\omega)$. This value corresponds to the equations for a *Frequency Domain* study when the equations are study controlled. To get the equations corresponding to an *Eigenfrequency* study, change the **Scaling factor** Δ to 1. Changing the scaling factor influences the coupling to other physics.

SOUND PRESSURE LEVEL SETTINGS

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

TYPICAL WAVE SPEED FOR PERFECTLY MATCHED LAYERS

Enter a value or expression for the typical wave speed for perfectly matched layers c_{ref} (SI unit: m/s). The default is `ta.c` and the value is automatically taken from the material model. If several materials or material models are used, it is best practice to add one PML for each. This will ensure that the typical wavelength is continuous within each PML feature.

STABILIZATION

To display this section, click the **Show More Options** button ( ) and select **Stabilization**. Select **No stabilization applied** (the default), **Galerkin least-squares (GLS) stabilization**, or **Streamline upwind Petrov-Galerkin (SUPG) stabilization**. When linear thermoviscous acoustic problems are solved (like in the frequency domain) the numerical problem is stable with the default P1-P2-P2 discretization. Enabling stabilization will ensure stability also for other combinations of discretization orders.

GLOBAL PORT SETTINGS

Select the **Port sweep settings** as:

- **No port sweep** (the default).
- **Activate port sweep** to enable the port sweep functionality.
- The **Legacy port sweep** option is only present for backward compatibility reasons.

The **Activate port sweep** option is used to compute the full scattering matrix when Port conditions are used. For more details see The Port Sweep Functionality subsection. The section only exists for 3D, 2D, and 2D axisymmetry.

Select the **Mode shape normalization** as **Amplitude normalization** (the default) or **Power normalization**. This setting controls if the mode shapes are normalized to have a unit maximum pressure amplitude or carry unit power. The selection determines how the scattering matrix is to be interpreted.

DISCRETIZATION

From the list, select the element order and type (Lagrange or serendipity) for the **Pressure**, the **Velocity field**, and the **Temperature variation**, respectively. The default is **Linear** for the pressure and **Quadratic serendipity** for the velocity and the temperature.

- For numerical stability reasons, the element order for the pressure should be one less than the element order for the velocity. Unless Stabilization is enabled, in which case, for example, a P1-P1-P1 discretization can be used.
- In fluids where the thermal and viscous boundary layer thickness are of the same order of magnitude (where the Prandtl number $\mathbf{Pr}$ is of the order 1, like in air), it is recommended to use the same element order for the temperature and the velocity. Both fields vary equally over the same length scale in the acoustic boundary layers near walls.

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

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields): the **Pressure** p, the **Velocity field** $\mathbf{u}$ and its components, and the **Temperature variation** T. The names can be changed but the names of fields and dependent variables must be unique within a model.

- Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface
- Theory Background for the Thermoviscous Acoustics Branch

In the *COMSOL Multiphysics Reference Manual*, see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- *Uniform Layer Waveguide*: Application Library path **Acoustics_Module/Verification_Examples/uniform_layer_waveguide**
- *Transfer Impedance of a Perforate*: Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/transfer_impedance_perforate**
- *Generic 711 Coupler — An Occluded Ear-Canal Simulator*: Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/generic_711_coupler**
- *Viscous Damping of a Microperforated Plate in the Slip Flow Regime*: Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/viscous_damping_mpp**
- *MEMS Microphone with Slip Wall*: Application Library path **Acoustics_Module/Electroacoustic_Transducers/mems_microphone_slip_wall**

Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface

The Thermoviscous Acoustics, Frequency Domain Interface has these domain, boundary, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

	The **Continuity** node with this physics interface is available as a pair boundary condition. This gives continuity in pressure, temperature variation, velocity, and in the flux on a pair boundary between thermoviscous acoustic domains.

Some of the following nodes are available from the **Mechanical** and **Thermal** submenus (listed in alphabetical order):

- Adiabatic
- Axial Symmetry
- Background Acoustic Fields
- Boundary Stress
- Heat Flux
- Heat Source
- Impedance
- Initial Values
- Interior Lumped Speaker Boundary
- Interior Impedance
- Interior Slip Wall
- Interior Temperature Variation
- Interior Velocity
- Interior Wall
- Isothermal
- Lumped Port
- Lumped Speaker Boundary
- No Slip
- No Stress
- Periodic Condition
- Port
- Pressure (Adiabatic)
- Slip (Perfect)
- Slip Wall
- Surface Tension
- Symmetry
- Temperature Variation
- Thermoviscous Acoustics Model
- Velocity
- Wall

Thermoviscous Acoustics Model

Use the **Thermoviscous Acoustics Model** node to define the model inputs (the background equilibrium temperature and pressure) and the material properties of the fluid (dynamic viscosity, bulk viscosity, thermal conductivity, heat capacity at constant pressure, and equilibrium density) necessary to model the propagation of acoustic compressible waves in a thermoviscous acoustic context. Extended inputs are available for the coefficient of thermal expansion and the compressibility, which enables modeling of any constitutive relation for the fluid.

MODEL INPUTS

This section contains field variables that appear as model inputs. The fields for the **Equilibrium pressure** p_0 and the **Equilibrium temperature** T_0 are always active as they enter the governing equations explicitly. If material properties depend on other model inputs they will automatically appear in this section.

Select **User defined** (the default), **Common model input**, or an existing variable from another physics interface.

The **Equilibrium pressure** p_0 (SI unit: Pa) has the default value set to 1 atm. The **Equilibrium temperature** T_0 (SI unit: K) has the default value set to 293.15 K (that is, 20°C).

Details about the Model Input and the Default Model Inputs are found in the Global and Local Definitions chapter of the *COMSOL Multiphysics Reference Manual*.

FLUID PROPERTIES

Define the material properties of the fluid by selecting an **Equilibrium density** — **Ideal gas**, **From material**, or **User defined**.

- If **From material** is selected (the default), the equilibrium density, and its dependence on the equilibrium pressure p_0 and temperature T_0, is taken from the defined material. Make sure that the Thermal Expansion and Compressibility settings are correct.
- For **Ideal gas** also select the Gas constant type — select Specific gas constant R_s (SI unit: J/(kg·K) or Mean molar mass M_n (SI unit: kg/mol)
- For **User defined**, enter a value or expression for the **Equilibrium density** $\rho_0(p_0, T_0)$ (SI unit: kg/m^3). The default is `ta.p0/(287[J/kg/K]*ta.T0)`, which is the ideal gas law.

The other thermoviscous acoustic model parameters defaults use values **From material**. For **User defined**, enter another value or expression for:

- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Bulk viscosity** μ_B (SI unit: Pa·s). The bulk viscosity parameter describes the difference between the mechanical and thermodynamic pressures. It is associated with losses due to expansion and compression.

- **Thermal conductivity** k (SI unit: W/(m·K)).
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)). This is the specific heat capacity or heat capacity per unit mass.

For a discussion about the air and water materials as commonly used in acoustics and other material properties, such as the bulk viscosity, see the Acoustic Properties of Fluids chapter of this manual.

THERMAL EXPANSION AND COMPRESSIBILITY

One of the main characteristics of an acoustic wave is that it is a compressional wave. In the detailed thermoviscous acoustic description, this property is closely related to the constitutive relation between the density, the pressure, and the temperature. This results in the important (linear) relation for the acoustic density variation

$$\rho_t = \rho_0(\beta_T p_t - \alpha_p T_t)$$

where ρ_t is the total density variation, p_t is the total acoustic pressure, T_t is the total acoustic temperature variation, β_T is the (isothermal) compressibility of the fluid, and α_p the (isobaric) coefficient of thermal expansion (sometimes named α_0). If this constitutive relation is not correct, then no waves propagate or possibly they propagate at an erroneous speed of sound. The default behavior is to define both quantities from the speed of sound and the ratio of specific heats (using the **From speed of sound** option) which are material properties often more readily available.

Note that when the **Adiabatic formulation** option is selected, under the Thermoviscous Acoustics Equation Settings section, the equation of state reduces to

$$\rho_t = \rho_0 \beta_s p_t = \rho_0 \frac{\beta_T}{\gamma} p_t .$$

When the **From equilibrium density** option is selected for the coefficient of thermal expansion and the compressibility, both values are derived from the equilibrium density $\rho_0(p_0, T_0)$ using their defining relations

$$\beta_T = \frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial p_0}\right]_{T_0} \qquad \alpha_p = -\frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial T_0}\right]_{p_0}$$

If the equilibrium density ρ_0 is a user-defined constant value, is picked up from another physics interface, or the material model does not define both a pressure and temperature dependence for ρ_0, the coefficient of thermal expansion and/or the

compressibility will evaluate to 0. Then the default **From speed of sound** option or the **User defined** options should be used.

If the material is air, the **From equilibrium density** option works well as the equilibrium density $\rho_0 = \rho_0(p_0, T_0)$ is defined as a function of both pressure and temperature.

For the water material the coefficient of thermal expansion is well defined as $\rho_0 = \rho_0(T_0)$, while the compressibility should be defined using the default **From speed of sound** option.

The **Thermal Expansion and Compressibility** section is displayed if **From material** or **User defined** is selected as the **Equilibrium density** under **Thermoviscous Acoustics Model**. For the **Ideal gas** option, the parameters are readily defined.

Select an option from the **Coefficient of thermal expansion** α_p list — **From material**, **From equilibrium density**, **From speed of sound** (the default), or **User defined**. For **User defined** enter a value for α_p (SI unit: $1/\mathrm{K} = \mathrm{K}^{-1}$).

Select an option from the **Isothermal compressibility** β_T list — **From equilibrium density**, **From isentropic compressibility**, **From speed of sound** (the default), or **User defined**. For **User defined**, enter a value for β_T (SI unit: $1/\mathrm{Pa} = \mathrm{Pa}^{-1}$).

The different options for defining the coefficient of thermal expansion and the compressibility stem from their thermodynamic definitions:

$$\beta_T = \frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial p_0}\right]_T = \frac{1}{\rho_0}\frac{\gamma}{c^2} = \gamma\beta_s$$

$$\alpha_p = -\frac{1}{\rho_0}\left[\frac{\partial \rho_0}{\partial T_0}\right]_p = \frac{1}{c}\sqrt{\frac{C_p(\gamma-1)}{T_0}}$$

For each of the following, and based on the above selection, the default is taken **From material**. For **User defined** enter another value or expression in the text field.

- **Speed of sound** c (SI unit: m/s).

- **Ratio of specific heats** γ (dimensionless). The default is 1.
- **Isentropic compressibility** β_s (SI unit: $1/\text{Pa} = \text{Pa}^{-1}$).

See the Theory Background for the Thermoviscous Acoustics Branch section for a detailed description of the governing equations and the constitutive relations.

To visualize the acoustic velocity (the particle velocity) three important variables exist for the velocity amplitude:

- The real part of the total acoustic velocity `ta.v_real`.
- The imaginary part (out-of-phase component) of the total acoustic velocity `ta.v_imag`.
- The RMS value of the total acoustic velocity `ta.v_rms`.

To visualize the dissipated energy due to viscosity and thermal conduction in postprocessing, three postprocessing variables exist:

- The viscous power dissipation density `ta.diss_visc`.
- The thermal power dissipation density `ta.diss_therm`.
- The total thermo-viscous power dissipation density `ta.diss_tot`.

In certain cases, it can be interesting not to include thermal conduction in the model and treat all processes as adiabatic (isentropic). This is, for example, relevant for most liquids where the thermal boundary layer is much thinner than the viscous. Not solving for the temperature field T also saves some degrees of freedom (DOFs). This is achieved by selecting the **Adiabatic formulation** option under the Thermoviscous Acoustics Equation Settings section.

See also Solver Suggestions for Large Thermoviscous Acoustics Models for suggestions on how to select an iterative solver for large problems.

Background Acoustic Fields

When the **Background Acoustic Fields** feature is added to a domain, it is possible to define the value of the background acoustic field variables p_b, $\mathbf{u}_b$, and T_b. Use this feature to set up scattering problems. To define fields at the inlet of a waveguide, it is recommended to use the Port condition.

MODEL INPUTS

This section contains field variables that appear as model inputs. The fields are always active as the equilibrium (background) temperature T_0 enters the governing energy equation explicitly. It is used for the **Plane wave** option.

BACKGROUND ACOUSTIC FIELDS

Select the **Acoustic field type** — **User defined** (the default) or **Plane wave**.

When **User defined** is selected, enter expressions for the **Background acoustic pressure** p_b, the **Background acoustic velocity** $\mathbf{u}_b$, and the **Background temperature variation** T_b. These can be analytical expressions or values of dependent variables solved in a previous study that defines the background field.

When **Plane wave** is selected, enter the **Pressure Amplitude** $|p_b|$ (given at $\mathbf{x} = \mathbf{0}$) and the wave direction vector $\mathbf{e}_k$ (this vector is automatically normalized). Finally, select where the **Material data** should be taken from; the default is **From material model** (the same as the domain model material) or select a material from the list. This will set up the background pressure, velocity, and temperature variations for a plane wave with the correct thermoviscous attenuation compatible with the governing equations.

For many applications, the **Plane wave** option available in the Port condition is a good alternative to the **Background Acoustic Fields** feature. The port automatically ensures a nonreflective behavior for the reflected outgoing waves.

Dedicated postprocessing variables exist for the total, background, and scattered fields, including all the acoustic fields, sound pressure level, and intensity.

Heat Source

Use the **Heat Source** node to define the heat source for the thermoviscous acoustics model. This adds a domain heat source Q to the right-hand side of the energy equation.

HEAT SOURCE

Enter a value for the **Heat source** Q (SI unit: W/m^3).

Initial Values

The **Initial Values** node adds initial values for the sound pressure, velocity field, and temperature variation. If necessary, add more **Initial Values** nodes from the **Physics** toolbar to give initial values in, for example, other domains.

INITIAL VALUES

Enter values or expressions for the **Pressure** p (SI unit: Pa), **Velocity field** $\mathbf{u}$ (SI unit: m/s), and **Temperature variation** T (SI unit: K).

Axial Symmetry

In a 2D axisymmetric geometry, the **Axial Symmetry** condition is automatically added on the axis of symmetry at $r = 0$. On this boundary, the radial component of the total velocity is set equal to zero

$$u_t = 0 \qquad \mathbf{u}_t = (u_t, v_t, w_t)$$

Wall

Use the **Wall** node to model the most common conditions at solid surfaces. This is the default boundary condition. This condition contains both a mechanical and a thermal selection. The default is a no-slip and isothermal condition applicable in most cases. More advanced conditions at boundaries can be set up by combining any of the **Mechanical** and **Thermal** conditions available. To model a wall on an interior boundary, use the Interior Wall condition.

For very small dimensions or at very low ambient pressures (in the Knudsen number regime from 0.001 to 0.1), the wall conditions will behave nonideally; for this purpose, use the Slip Wall boundary condition. This is the regime known as slip-flow in CFD.

The **Wall** condition is also the default condition added when the physics interface is set up. Note that the selections in this default condition can be changed (they are not locked to all boundaries). When used together with a perfectly matched layer (PML), it is recommended to use the no-slip option at the outer boundary of the layer. Specifically, using the **Slip (perfect)** condition in combination with a rational scaling in the PML will lead to a singularity and an error.

The no-slip condition is the origin of the viscous boundary layer and the isothermal condition is the origin of the thermal boundary layer. It is within these acoustic boundary layers that the main dissipation happens. See the Theory Background for the Thermoviscous Acoustics Branch section for more details.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip (perfect)**. See the No Slip and Slip (Perfect) conditions for further details.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. If **Slip (perfect)** is selected for the **Mechanical condition** and **Weak constraints** is selected for the **Constraints**, then the default Nitsche (penalty like) formulation (**Automatic**) is switched to a Lagrange multiplier formulation instead. See the Slip (Perfect) condition for details. For the **No-slip** and **Isothermal** selections a weak formulation is used instead of a pointwise constraint.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Pressure (Adiabatic)

Use the **Pressure (Adiabatic)** node to specify a prescribed pressure p_{bnd}, that acts as a pressure source at the boundary, typically an inlet or outlet. In the frequency domain,

p_{bnd} is the amplitude of a harmonic pressure source. The adiabatic condition states that no heat flows into or out of the boundary:

$$p_t = p_{bnd}$$

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -p_{bnd}\mathbf{n}$$

$$-\mathbf{n}\cdot(-k\nabla T_t) = 0$$

This condition is in general not physically correct on a solid wall because solids are generally better thermal conductors than air.

PRESSURE

Enter the value of the **Pressure** p_{bnd} (SI unit: Pa) at the boundary. The default is 0 Pa.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Symmetry

The **Symmetry** node for The Thermoviscous Acoustics, Frequency Domain Interface adds a boundary condition that represents symmetry. It corresponds to the Slip (Perfect) condition for the mechanical degrees of freedom and the Adiabatic condition for the temperature variation.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Slip Wall

The **Slip Wall** boundary condition is used to model the effective nonideal wall conditions that exist in the slip-flow regime, when the Knudsen number is in the range from 0.001 to 0.1. The **Slip Wall** conditions is used for systems with very small geometrical dimensions or systems running at very low ambient pressures. This is, for example, relevant when modeling MEMS transducers and other microdevices. To model a slip wall on an interior boundary, use the Interior Slip Wall condition.

In the slip flow regime, the linearized Navier-Stokes equations (the equations solved for thermoviscous acoustics) can still be used to model the acoustics, except within a thin layer of fluid adjacent to the walls. Within this layer, which is known as the Knudsen layer, the fluid is significantly rarefied. The effect of the Knudsen layer on the continuum part of the acoustic perturbation equations can be modeled by means of a slip model condition at the wall. In the slip wall formulation used here, the slip velocity depends linearly on the tangential wall stress (the traction) and also an a component related to thermal creep. The total acoustic velocity $\mathbf{u}_t$ and total acoustic temperature variations T_t at the wall are in the general case given by:

$$\mathbf{u}_t = \mathbf{u}_w + \mathbf{u}_{slip} \qquad \mathbf{u}_{slip} = -\frac{\sigma_s \lambda_0}{\mu}\tau_{n,t} + \mathbf{u}_{therm}$$

$$\tau_{n,t} = \tau \cdot \mathbf{n} - (\mathbf{n}^T \tau \mathbf{n})\mathbf{n}$$

$$\mathbf{u}_{therm} = \sigma_T \frac{\mu}{\rho_0 T_0}\left(\nabla T_t - (\mathbf{n} \cdot \nabla T_t)\mathbf{n} - \frac{\rho_0 T_t + \rho_t T_0}{\rho_0 T_0}(\nabla T_0 - (\mathbf{n} \cdot \nabla T_0)\mathbf{n})\right)$$

$$T_t = -\zeta_T(\lambda_0 \mathbf{n} \cdot \nabla T_t + \lambda_1 \mathbf{n} \cdot \nabla T_0)$$

where $\mathbf{u}_w$ is the wall velocity (fixed or moving), σ_s is the viscous slip coefficient, λ_0 is the mean free path, σ_T is the thermal slip coefficient, $\mathbf{u}_{therm}$ is the thermal creep, and ζ_T is the temperature jump coefficient. The solved set of equations represents a linearized version of the equations solved in The Slip Flow Interface available with the Microfluidics Module.

When the **Slip Wall** boundary condition is placed next to a constraint feature, it can be necessary to handle the shared points/edges. This can be done by excluding the points/edges from the constraint feature. The **Wall** boundary condition with **No Slip** and/or **Isothermal** is an example of a constraint feature that can require handling. See Suppressing Constraints on Lower Dimensions for details.

MECHANICAL

Select the **Mechanical condition** as **Fixed wall** (the default) or **Moving wall**. For the **Moving wall** option select **User defined** (the default) enter the **Velocity of the moving wall** $\mathbf{u}_w$ (SI unit: m/s). Use this option to define a source, for example, a vibrating structure. The fixed option represents $\mathbf{u}_w = \mathbf{0}$. Announced variables for a wall velocity can also be selected when applicable.

When setting up a coupling to a vibrating structure including the slip wall effects, select the announced solid velocity from the list, for example, **Velocity (solid/lemm1)**. The coupling back to the solid is then achieved by applying a **Boundary Load** to the structure and selecting the announced **Force per unit area** variable, for example, **Acoustic slip wall traction per unit area (ta/slw1)**.

THERMAL

Select the **Thermal condition** as **Isothermal** (the default) or **Temperature variation**. For the **Temperature variation** option enter the **Temperature variation** of the wall T_w (SI unit: K). The isothermal option represents $T_w = 0$.

When **Isothermal** is selected, choose to select (the default) or clear the **Neglect thermal creep** checkbox. When thermal creep is included, an additional contribution is added to the slip velocity. The effect is typically negligible for the default isothermal wall condition and when there are no gradients in the equilibrium temperature T_0. Thermal creep is always included when a temperature variation is specified at the wall.

SLIP MODEL

Select how to define the **Slip coefficients** used in the slip wall formulation. The options are to use **Maxwell's model** (the default), **User defined**, or **Slip length**.

- For **Maxwell's model** enter the **Tangential momentum accommodation coefficient** α_v (default value: 0.9). This model is applicable for gases and is based on mean-free-path considerations.
- For **User defined** enter the three free parameters of the model: the **Viscous slip coefficient** σ_s (default value: 1.1), the **Thermal slip coefficient** σ_T (default value: 1.2), and the **Temperature jump coefficient** ζ_T (default value: 2.2). The default values

correspond to the default values of Maxwell's model for air at typical ambient conditions. For more details on the parameters see Ref. 10.

The **Thermal slip coefficient** field is not present if **Isothermal** and **Neglect thermal creep** is selected. No temperature settings are present when **Adiabatic formulation** is selected.

- For **Slip length**, the slip velocity is defined through the classical slip length parameter. Enter the **Slip length** L_s (default value: 1 mm) and the two thermal coefficients if thermal creep is included.

  The **Thermal slip coefficient** field is not present if **Isothermal** and **Neglect thermal creep** is selected. No temperature settings are present when **Adiabatic formulation** is selected.

- Theory Background for the Thermoviscous Acoustics Branch.
- Theory for the Slip Wall Boundary Condition

- *Viscous Damping of a Microperforated Plate in the Slip Flow Regime*. Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/viscous_damping_mpp**
- *MEMS Microphone with Slip Wall*. Application Library path **Acoustics_Module/Electroacoustic_Transducers/mems_microphone_slip_wall**

CONSTRAINT SETTINGS

The **Constraint** type for the **Slip Wall** is always to use **Weak constraints**, and the default for the **Apply reaction terms on** option is **Individual dependent variables**. With these default settings, coupling a thermoviscous domain to a vibrating structure using the above mentioned steps will work consistently. It is also possible to choose **All physics (symmetric)** for the latter. This option does, however, not guarantee a consistent two-way coupling to other physics.

Port

The **Port** boundary condition is used to excite and absorb acoustic waves that enter or leave waveguide structures, like small ducts or channels, in an acoustic model. The thermoviscous port formulation ensures that the nontrivial mode shapes of the acoustic velocity and thermal fields are captured correctly.

A given port condition supports one specific propagating mode. To provide the full acoustic description, combine several port conditions on the same boundary. Typically, only the plane wave mode is propagating in small structures where the thermoviscous representation is necessary. The port condition provides a superior nonreflecting or radiation condition for waveguides compared to a simple impedance condition or a perfectly matched layer (PML) configuration. The same port boundary condition feature should not be applied to several waveguide inlets and outlets. The port condition supports S-parameter (scattering parameter) calculation but it can also be used as a source to just excite a system.

The **Port** boundary condition exists for 3D, 2D, and 2D axisymmetric models.

Only one port should be excited at a time if the purpose is to compute S-parameters. The S-parameters are defined as `ta.S11`, `ta.S21`, and so on, and can be used in postprocessing.

- An evaluation group for the **Scattering Coefficient** exists as a **Predefined Plot**. When evaluating, change the **Data Series Operation** for the port sweep parameter (`PortName`) to **Sum** to get the proper data formatting.

In other cases, having more than one port exciting the system might be wanted, but the S-parameter variables cannot be correctly computed. So, when several ports are excited, the S-parameter output is turned off.

If only two ports are added to the system and a port sweep is performed, the system transfer matrix and system impedance matrices are also computed.

On a given boundary, a combination of ports will define the total acoustic fields (sum of incident and outgoing pressure, temperature, and velocity waves) as

$$p_t = \sum_{i \in \text{bnd}} (S_{ij} + A^{\cdots} e^{\cdot\cdot} \delta_{ij}) p_i$$

$$\mathbf{u}_t = \sum_{i \in \text{bnd}} (S_{ij}\mathbf{u}_i + A^{\text{in}} e^{i\varphi} \delta_{ij}(\mathbf{u}_i - 2\mathbf{n}(\mathbf{n} \cdot \mathbf{u}_i)))$$

$$T_t = \sum_{\cdot\,\cdot\,\cdot} (S_{ij} + A^{\text{in}} e^{i\varphi} \delta_{ij}) T_i$$

where the summation "i" is over all ports on the given boundary "`bnd`", S_{ij} is the scattering parameter, A^{in} is the amplitude of the incident field and φ the phase (at port "j"), and p_i, $\mathbf{u}_i$, and T_i are the mode shape of the i^{th} port. The mode shape is

normalized to have either a unit maximum amplitude (for the pressure p_i) or carry unit power (see the normalization option in the Global Port Settings section). For both definitions, the scattering parameter S_{ij} defines the amplitude of mode i when a system is exited at port j (with mode j). For the power scaling, $|S_{ij}|^2$ directly gives the power of the given mode. This corresponds to a multimode expansion of the solution on the given boundary. The scattering parameters are automatically calculated when an acoustic model is set up with just one port exciting the system. To get the full scattering matrix The Port Sweep Functionality can be used.

PORT PROPERTIES

Enter a unique **Port name**. Only nonnegative integer numbers can be used as **Port name** as it is used to define the elements of the S-parameter matrix. The numeric port names are also required for the port sweep functionality. The port name is automatically incremented by one every time a port condition is added.

Select a **Type of port**: **User defined** (the default), **Numeric (0,0)-mode**, **Circular (0,0)-mode**, **Slit (0,0)-mode**, or **Plane wave**. Depending on the selection, different options appear in the **Port Mode Settings** section (see below). Use the **Circular (0,0)-mode** for a port with a circular cross section in 3D or 2D axisymmetry and the **Slit (0,0)-mode** option in 2D. If the port has a different cross section than either of these, use the **User defined** option or the **Numeric (0,0)-mode** port. The **Plane wave** option represents a situation where the boundary layers are not included; this is, for example, for a wave propagating in free space or when slip (perfect) and adiabatic (or symmetry) conditions are applied to all adjacent boundaries.

PORT MODE SETTINGS

Depending on the option selected in the **Type of port** (see above):

- For **User defined**, enter user defined expressions for the **Mode shape** p_n, $\mathbf{u}_n$, T_n, and the **Mode wave number** k_n (SI unit: rad/m). The mode shape will automatically be scaled before it is used in the port condition. Use the user-defined option to enter a known analytical expression or to use the solution from The Thermoviscous Acoustics, Boundary Mode Interface. The solutions from the boundary mode analysis can be referenced using the `withsol()` operator.
- The **Numeric (0,0)-mode** port options is used for waveguides of arbitrary cross sections. In this case, the shape of the propagating plane-wave mode (0,0) is solved on the port face. The boundary conditions for the mode are taken from the adjacent waveguide boundaries. This automatic detection works for slip (perfect), no-slip, adiabatic, isothermal, and symmetry conditions (including the same options when

selected in the **Wall** condition). If all the adjacent wave guide boundaries are slip and adiabatic (or symmetry) then use the **Plane wave** option.

For this option, a special solver sequence is automatically generated since the port mode shape (the port variables `psi`, `Psi_th`, `Psi_v`, and `vip`) should be solved before the domain problem (the main degrees of freedom `p`, `u`, `T`, and `Sparam1`). If an iterative solver suggestion is to be used, keep the linear solver for the first segregated step and then select the iterative suggestion for the second segregated step. If the **Numeric (0,0)-mode** port option is selected after a solver has already been generated, then select **Reset solver to default** in the study to get the correct settings.

- The **Circular (0,0)-mode** port option is used for waveguides of circular cross section. The analytical mode is a plane-wave mode (0,0) given by a constant cross section pressure, no-slip condition for the velocity, and isothermal condition for the temperature. An example of the propagating mode shape in a cylindrical waveguide is seen below.

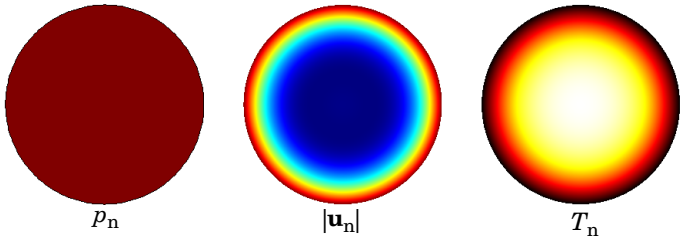


Figure 6-1: Plane wave mode for a circular duct of 1 mm diameter at f = 250 Hz.

Select how the **Circle radius** of the cross section is defined, either **Automatic** (the default) or **User defined**. The latter option can for some geometry configurations increase the numerical precision of the computed mode.

- The **Slit (0,0)-mode** port option only exists in 2D on a boundary. In 2D, the geometry is assumed infinite in the out-of-plane direction and represents a slit. The analytical mode is a plane-wave mode (0,0) given by a constant cross section pressure, no-slip condition for the velocity, and isothermal condition for the temperature.
- The **Plane wave** port option represents a situation where the boundary layers are not included in the mode shape, this is, for example, for a wave propagating in free space

or when slip (perfect) and adiabatic (or symmetry) conditions are applied to all adjacent boundaries.

The solution for the numeric port is obtained using the generalized low reduced frequency (LRF) approximation commonly used in thermoviscous acoustics. The theory is based on an order of magnitude analysis of the full thermoviscous equations and on the assumption of a plane propagating mode (constant pressure in cross section). The circular and slit options use an analytical solution for the given cross section.

The mode shapes based on LRF approximation are valid as long as the wavelength is much larger than the waveguide cross section ($\lambda >> a$) and the wavelength is much larger than the boundary layer thickness ($\lambda >> \delta_{visc}$ and $\lambda >> \delta_{therm}$).

PORT GEOMETRY

The settings in this section ensure that the correct port area (including symmetries) is used for computing important quantities like power, volume flow, and transfer matrix elements, associated with the port

Select the **Port area** as **Use symmetries** (the default) or **Selected boundaries**.

- When **Use symmetries** is selected, symmetry conditions adjacent to the port will automatically be taken into account if the **Port area multiplication factor** is set to **Automatic** (the default); if set to **User defined**, enter the area multiplication factor A_{scale} manually.
- When **Selected boundaries** is selected, the port will have the area of the selected boundaries, without taking any symmetry conditions into account.

The **Port** condition can only be applied to connected boundaries and the boundaries should typically be coplanar. The port condition can in principle be applied to a curved continuous boundary, but care should be taken that the setup is mathematically and physically consistent.

The **Port** boundary conditions should in general be placed at least one waveguide diameter from other geometry features to ensure that only a pure fully developed propagating wave exists at the port. Note also that the port condition does not treat possible evanescent waves.

The geometry restriction typically has to be tightened for the **Plane wave** option, where no attenuation exists at the waveguide walls. In this case, the port should be placed at least two viscous wavelengths from other geometry features. This is to ensure that reflected evanescent viscous waves are fully dissipated. In this case, at the lowest frequency $f_{\min}$ studied in the model, place the port at a distance:

$$2L_v = 4\pi\delta_v = 2\pi\sqrt{\frac{\mu}{\pi\rho_0 f_{\min}}}$$

INCIDENT MODE SETTINGS

Activate if the given port is excited by an incident wave of the given mode shape. For the first **Port** condition added in a model, the **Incident wave excitation at this port** is set to **On**. For subsequent conditions added the excitation is set to **Off** per default. If more than one port in a model is excited, the S-parameter calculation is not performed.

When the **Incident wave excitation at this port** is set to **On**, then select how to define the incident wave. Select the **Define incident wave** as **Pressure amplitude** (the default) or **Power** (**Power per unit length** in 2D models)

- For **Pressure amplitude**, enter the amplitude A_p^{in} (SI unit: Pa) of the incident wave. This is in general defined as the maximum pressure amplitude for a given mode shape.
- For **Power**, enter the power P^{in} (SI unit: W) of the incident mode. In 2D models, this will be a **Power per unit length** (SI unit: W/m).
- Enter the phase φ (SI unit: rad) of the incident wave. This phase contribution is multiplied with the amplitude defined through the above two options. Those inputs can be complex numbers.

Note that when the **Activate port sweep** option is selected at the physics level, the options in the **Incident Mode Settings** section are deactivated. This is because this option

automatically sends in a mode of unit amplitude, sweeping through one port at the time.

All ports with incident wave excitation turned off have an arrow symbol, added in the Graphics window, that points outward. When incident wave excitation is turned on, the arrow symbol points inward.

If a waveguide is cut with one or several symmetry planes, care should be taken when setting up the port modes and when postprocessing.

For the **Circular** and **Slit** options, make sure to only select modes that are actually symmetric according to the symmetry planes.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details

THE PORT SWEEP FUNCTIONALITY

The port sweep functionality is used to reconstruct the full scattering matrix S_{ij} by automatically sweeping the port excitation through all the ports included in the model. When the port sweep is activated, the options in the **Incident Mode Settings** in the port conditions are deactivated and COMSOL controls which port is excited with an incident mode.

The port sweep functionality is activated at the main physics interface level by selecting **Activate port sweep** in the Global Port Settings section. Enter the **Sweep parameter name**, the default is `PortName`. Create a parameter with the same name under **Global Definitions** > **Parameters 1**. This is the name of the parameter to be used in a parametric sweep; it should represent the **Port name** integer values (defined when adding the port conditions). Add a parametric sweep study step and run the sweep over the `PortName` parameter with an integer number of values representing all the ports in the model. Once the model is solved, the full scattering matrix can be evaluated using the defined global variables `ta.S11`, `ta.S21`, `ta.S12`, and so on. The transmission loss (TL)

between two given ports is also computed, for example, the variable for the TL loss from port 1 to 2 is given by `ta.TL_12`.

- An evaluation group for the **Scattering Coefficient** exists as a **Predefined Plot**. The **Global Matrix Evaluation** is used to evaluate the full scattering matrix `acpr.S`. When evaluating, change the **Data Series Operation** for the port sweep parameter (typically `PortName`) to **Sum** to get the proper data formatting.
- An **Predefined Plot** evaluation group also exists for the **Transfer Matrix**. This plot evaluates the real and imaginary parts of the four matrix components separately. The output format can readily be exported to a file and used as interpolation data for subsequent use in other models, for example, in connection to a Lumped Port condition.

If only two ports are added to the thermoviscous model, COMSOL also automatically computes the transfer matrix of the system (variables `ta.T11`, `ta.T12`, `ta.T21`, `ta.T22`) and the impedance matrix of the system (`ta.Z11`, `ta.Z12`, `ta.Z21`, `ta.Z22`). These expressions are only true if plane wave modes are used. This is the case in nearly all configurations when working with microacoustic systems. For ports in thermoviscous acoustics, the **Numeric (0,0)-mode**, **Circular (0,0)-mode**, **Slit (0,0)-mode**, and **Plane wave** options are for plane waves only, that is, the (0,0) mode with varying boundary conditions. Higher-order modes can only be introduced with the **User defined** option. The transfer matrix representation is often used in electroacoustic modeling using lumped representations.

- *Wax Guard Acoustics: Transfer Matrix Computation*. Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/wax_guard_acoustics**
- *Transfer Impedance of a Perforate*. Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/transfer_impedance_perforate**

Lumped Port

The **Lumped Port** feature connects a waveguide or duct inlet/outlet to a lumped representation element. This can be an *Electrical Circuit* (lumped electroacoustic representation), a two-port network defined through a transfer matrix, or a lumped

representation of a waveguide. Basically, it couples the end of a waveguide to an exterior system that has a given acoustic lumped representation. Several representations and sources exist to describe the lumped system. When using the lumped port representation, it is assumed that only plane pressure waves propagate in the acoustic waveguide, that is the (0,0)-modes.

PORT MODE SETTINGS

The **Port Mode Settings** correspond to the options set in the **Port Properties** and **Port Mode Settings** sections of the Port condition.

PORT GEOMETRY

The **Port Geometry** settings correspond to those set in the Port condition.

The **Lumped Port** condition can only be applied to connected boundaries and the boundaries should typically be coplanar. The port condition can in principle be applied to a curved continuous boundary, but care should be taken that the setup is mathematically and physically consistent.

The **Lumped Port** boundary conditions should in general be placed at least one waveguide diameter from other geometry features to ensure that only a pure fully developed propagating wave exists at the port. Note also that the port condition does not treat possible evanescent waves.

The geometry restriction typically has to be tightened for the **Plane wave** option, where no attenuation exists at the waveguide walls. In this case, the port should be placed at least two viscous wavelengths from other geometry features. This is to ensure that reflected evanescent viscous waves are fully dissipated. In this case, at the lowest frequency $f_{\min}$ studied in the model, place the port at a distance:

$$2L_v = 4\pi\delta_v = 2\pi\sqrt{\frac{\mu}{\pi\rho_0 f_{\min}}}$$

LUMPED PORT PROPERTIES

Select the type of lumped representation connected to the port by selecting the **Connection type** as **Circuit**, **Two-port** (the default), or **Waveguide**. The selection made here influences the settings and options in the **Connection Type** section (see below).

- The **Circuit** option is used to connect the port boundary to an *Electrical Circuit* interface (this requires the AC/DC module). The connection is set up by adding the **External I vs. U** element in the circuit model and selecting the lumped port as the **Electric potential**, for example, **Voltage from lumped port (acpr/lport1)**. The feature automatically introduces a unit transformation from electric to acoustic units. The classical equivalence between the voltage V (SI units: V) and the pressure p (SI unit: Pa) as well as the current I (SI unit: A) and the volume velocity Q (SI unit: m^3/s) is made.
- For the **Two-port** option, it is assumed that the port boundary is connected to a two-port system described through a transfer matrix $\mathbf{T}$. See the **Connection Type** section for details.
- For the **Waveguide** option, it is assumed that the port boundary is extended with a waveguide of constant and identical cross section. The connection is equivalent to the two-port as the waveguide is defined through the appropriate transfer matrix $\mathbf{T}$, of a waveguide. See the **Connection Type** section for details.

For the **Two-port** and the **Waveguide** options also select the **Exterior condition** as **Circuit**, **Ideal source** (the default), or **Impedance**. The selection made here influences what section is visible in the user interface. See below for a description of the options.

CONNECTION TYPE

This section is visible if **Two-port** or **Waveguide** is selected as the **Connection type**.

- For **Two-port**, select the **Two-port representation** as **Acoustic** (the default) or **Electroacoustic**. The selection indicates if the transfer matrix is from acoustic to acoustic variables or from electric to acoustic variables. The first can be used to include a transfer matrix of an acoustic subsystem, for example, extracted using the Port condition. The latter should, for example, be used if the transfer matrix represents a transducer with electric inputs and acoustic outputs.

  Enter expressions, for example, calling interpolation functions that depend on frequency, for the elements of the transfer matrix T_{11}, T_{12}, T_{21}, and T_{22}.

  For the **Acoustic** option, you can choose to invert the transfer matrix (reverse its direction) by selecting **Invert transfer matrix direction**. This option will invert and correct the signs of the transfer matrix $\mathbf{T}$ behind the scene.

- For **Waveguide**, select the **Waveguide model** as **Automatic** (the default) or **User defined**. The **Automatic** option will use the cross-section specific characteristic impedance and fluid properties of the boundary for the waveguide model. If **User defined** is selected, enter values for the **Wave number** k, the **Specific characteristic acoustic impedance** Z. For both options, enter the **Waveguide length** L (SI unit: m) modeled with the lumped representation.

The transfer matrix $\mathbf{T}$ couples the inlet (1) and outlet (2) pressure p and flow rate Q, defined in the feature, according to the following usual convention:

$$\begin{bmatrix} p_1 \\ Q_1 \end{bmatrix} = \begin{bmatrix} T_{11} & T_{12} \\ T_{21} & T_{22} \end{bmatrix} \begin{bmatrix} p_2 \\ Q_2 \end{bmatrix}$$

What is inlet and outlet depends on if the port excites the system (with a source) or it is passive (with an end impedance).

For the **Two-port** option, variables exist for evaluating the pressure and sound pressure level at both ends of the two-port element. The variables are `ta.lport1.p1`, `ta.lport1.p2`, `ta.lport1.Lp1`, and `ta.lport1.Lp2`, where the physics tag and feature tag should be selected adequately. The number 1 and 2 in the variable name, refers to the inlet and outlet of the element.

SOURCE SETTINGS

This section is visible if **Ideal source** is selected as the **Exterior condition**.

Select the **Input source** as **Average pressure** (the default) or **Volume flow**. Enter values for the average pressure $p_{\text{av,in}}$ or the volume flow Q_{in}. These represent ideal sources equivalent to an ideal voltage or ideal current source.

IMPEDANCE SETTINGS

This section is visible if **Impedance** is selected as the **Exterior condition**.

In this case, the lumped port is not active and an impedance condition can be applied to terminate the lumped representation. This could, for example, be the acoustic

impedance of a microphone membrane. Select the **Impedance model** as **User defined** (the default) or **RCL**.

- For the **User defined** option select the input format as an **Acoustic impedance** Z_{ac} (SI unit: kg/m^4·s) or an **Impedance** Z (SI unit: Pa·s/m). If the electroacoustic representation is used for the two-port enter the **Electric impedance** Z (SI unit: Ω).
- The **RCL** option is described for the Impedance boundary condition of The Pressure Acoustics, Frequency Domain Interface.

CIRCUIT

This section gives information about the state of coupling/connection to an Electrical Circuit interface. If the **Lumped Port** is not connected, the text **Connect to 'External I vs. U' in the Electrical Circuit interface** text will be displayed. Once connected, a reference with a tag to the associated **External I vs. U** node, in the Electrical Circuit interface, is displayed.

EXCLUDED EDGES

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Lumped Speaker Boundary

The **Lumped Speaker Boundary** condition is used to model a miniature loudspeaker (dust cap, cone, surround assembly, and motor) or another micro-transducer using a lumped representation with a coupling to an **Electrical Circuit** interface. The mechanical and electric properties of the speaker can, for example, be described through a Thiele-Small parameter representation and associated lumped parameters.

The properties of a back volume can also be modeled using a lumped representation through either a simple equivalent acoustic compliance, a more general RCL circuit representation, or a user-defined acoustic impedance. On the other hand, if the air domains on both sides of the speaker are modeled explicitly (included in the model) then there are two possible setups:

1. If the speaker diaphragm is represented by an interior boundary (a "shell" with air domains on both sides); then use the Interior Lumped Speaker Boundary condition.
2. If the speaker diaphragm is represented by a domain (a "solid" with air domains on both sides); then use two **Lumped Speaker Boundary** features, one applied to the upper side and one to the lower side. Make sure the speaker axis is the same on both

sides and that the back volume correction is disabled (the **No correction** option). Finally, in the **Electrical Circuit** add two **External I vs. U** nodes in series, when setting up the coupling.

For time domain modeling including possible nonlinear lumped effects, see the Lumped Speaker Boundary and Interior Lumped Speaker Boundary of The Thermoviscous Acoustics, Transient Interface.

SPEAKER GEOMETRY

Define how the **Speaker area** (projected) of the speaker is computed by selecting either **Selected boundaries** (the default) if all boundaries are present, or **Use symmetries** if the speaker surface is only partially represented.

When **Use symmetries** is selected, select the **Speaker area multiplication factor** A_{scale} as **Automatic** (the default, the model is analyzed for the presence of symmetry conditions); or **User defined** and enter a value (default is 1). The settings are required in order to compute the effective area of the speaker which is used to compute the acoustic load and radiated power.

Define the **Speaker axis direction** $\mathbf{e}_{\text{ax}}$ by selecting **Automatic** (the default) or **User defined**. For the automatic option, the axis is computed as the average of the surface normals, this option is valid if a full speaker surface is selected. The **User defined** option should be used if the speaker surface is only partially represented and is the only option when **Use symmetries** is selected.

BACK VOLUME CORRECTION

The **Lumped Speaker Boundary** does not model the acoustics on the back side of the speaker cone explicitly. The properties can be entered either through a volume giving a simple compliance effect, through a simple RCL model, or through a user defined acoustic impedance of the back volume.

Select the **Acoustic impedance of the back volume** as **Volume compliance** (the default), **RCL**, **User defined**, or **No correction**.

- For **Volume compliance**, enter a **Volume** V_{back} (SI unit: m^3). The volume represents an equivalent acoustic compliance $V_{\text{back}}/(\rho c^2)$.
- For **RCL**, enter the values of the **Equivalent acoustic resistance** R_{ac} (SI unit: $\text{kg}/(\text{m}^4\cdot\text{s})$), the **Equivalent acoustic compliance** C_{ac} (SI unit: $\text{m}^4\cdot\text{s}^2/\text{kg}$), and the **Equivalent acoustic inertance** L_{ac} (SI unit: kg/m^4).

- For **User defined**, enter an expression for the **Acoustic impedance** Z_{ac} (SI unit: $kg/(m^4 \cdot s)$) of the back volume. This can, for example, be a constant, an analytical expression, or interpolation data.
- For **No correction**, the correction impedance is 0.

CIRCUIT

This section gives information about the state of coupling/connection to an Electrical Circuit interface. If the **Lumped Speaker Boundary** is not connected, the text **Connect to 'External I vs. U' in the Electrical Circuit interface** text will be displayed. Once connected a reference with a tag to the associated **External I vs. U** node, in the Electrical Circuit interface, is displayed.

MECHANICAL

Select **Use slip** to get a tangential perfect slip condition on the velocity; the velocity is only coupled in the normal direction to the surface. When not selected (the default), the full velocity component is coupled.

THERMAL

Select the **Thermal condition** as **Isothermal** (default) or **Adiabatic**.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

!	When the **Lumped Speaker Boundary** is placed next to a **Wall** or **Interior Wall**, with **No slip** as the **Mechanical condition**, it is necessary to exclude common points (in 2D axisymmetry) or common edges (in 3D). This is done in the **Excluded Points/Edges** section. To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and physics interfaces being modeled. This feature works well for cases like opposing parallel boundaries. In other cases, use the **Destination Selection** section to control the destination. By default, it contains the source and destination selection that COMSOL Multiphysics identifies. The periodic condition automatically defines a mapping between the source $\mathbf{x}_s$ and the destination $\mathbf{x}_d$ points on the boundaries.

BOUNDARY SELECTION

The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODICITY SETTINGS

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

- For **Continuity**, the values of the field variables at destination are set equal to the source: $p(\mathbf{x}_d) = p(\mathbf{x}_s)$, $\mathbf{u}(\mathbf{x}_d) = \mathbf{u}(\mathbf{x}_s)$, and $T(\mathbf{x}_d) = T(\mathbf{x}_s)$. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that corresponding velocity components are connected.
- For **Antiperiodicity**, the values of the field variables on the destination are set equal to the values on the source with the sign reversed: $p(\mathbf{x}_d) = -p(\mathbf{x}_s)$, $\mathbf{u}(\mathbf{x}_d) = -\mathbf{u}(\mathbf{x}_s)$, and $T(\mathbf{x}_d) = -T(\mathbf{x}_s)$. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that corresponding velocity components are connected.

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

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

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

  This condition is used to model any geometry that has a cyclic periodic structure such as a microphone or a loudspeaker driver. Setting the azimuthal mode number determines what mode is analyzed. The response of the full system to an external excitation is in general a linear combination of many different modes. The option exists in 3D and 2D.

- For **User defined**, select the checkbox for any of the field variables as needed. Then for each selection, choose the **Type of periodicity** — **Continuity** or **Antiperiodicity**. If the source and destination boundaries are rotated with respect to each other, a transformation is automatically performed, so that corresponding velocity components are connected.

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

!	To get correct behavior of the **Periodic Condition**, in particular the **Floquet periodicity** and the **Cyclic symmetry** conditions, it is recommended that the source and destination meshes are identical. This can be achieved using the **Identical Mesh** mesh feature. When the **Periodic Condition** stretches across regions with a mix of default material models, PMLs, background pressure fields, or background acoustic fields, it is recommended to add one **Periodic Condition** for each set of such boundaries.

!	Note that the **Periodic Condition** sets up periodicity in the dependent variables but not in the Lagrange multipliers, if such are added by any constraints. The periodicity for these should be handled manually.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF DESTINATION

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog. For information about the **Orientation of Destination** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Mapping Between Source and Destination** section, see Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Interior Wall

The **Interior Wall** boundary condition is used to model a wall condition on an interior boundary. It is, similarly to the Wall boundary condition, available on exterior boundaries except that it applies on both sides (up and down) of an internal boundary. It allows discontinuities (in pressure, velocity, and temperature) across the boundary. Use the **Interior Wall** boundary condition to avoid meshing thin structures by applying this slit-like condition on interior curves and surfaces instead. Add this node from the **Interior Conditions** submenu.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip (perfect)**. See the No Slip and Slip (Perfect) conditions for further details.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

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

If **Slip (perfect)** is selected for the **Mechanical condition** and **Weak constraints** is selected as **Constraint**, the default Nitsche (penalty like) formulation is switched to a Lagrange multiplier formulation instead. For the **No-slip** and **Isothermal** selections a weak formulation is used instead of a pointwise constraint.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Interior Impedance

Use the **Interior Impedance** node to apply a transfer impedance condition on an interior boundary. Add this node from the **Interior Conditions** submenu.

The condition relates the normal stress at the two sides of the boundary (up and down) to the normal velocity via the normal specific transfer impedance Z_n. The condition allows discontinuities (a slit) in pressure and temperature depending on the condition chosen for the temperature.

$$(\sigma_{up} - \sigma_{down})\mathbf{n} = Z_n(\mathbf{u}_t \cdot \mathbf{n})\mathbf{n}$$

$$\sigma = \left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla \cdot \mathbf{u}_t)\mathbf{I}\right]$$

$$Z_n = \frac{p_{t,down} - p_{t,up}}{\mathbf{u}_t \cdot \mathbf{n}}$$

MECHANICAL

Enter a value or expression for the **Specific transfer impedance** Z_n (SI unit: Pa·s/m).

Select a condition for the **Tangential velocity** — **Slip** (the default) or **No slip**. The **No Slip** option will apply an additional constraint for the velocity in the tangential direction on the two sides of the interior boundary.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Interior Velocity

Use the **Interior Velocity** node to specify a velocity on an interior boundary in thermoviscous acoustics. Add this node from the **Interior Conditions** submenu.

The condition can be used to specify sources, for example, the velocity of a diaphragm in a miniature transducer that is modeled using a lumped circuit model. The velocity components can be prescribed independently, and there is an option to force continuity in pressure across the boundary, as well as options for the thermal conditions.

COORDINATE SYSTEM SELECTION

Select the **Coordinate system** to define the orientation of the prescribed quantities (the velocity). The default is the **Global coordinate system**; any other applicable system will be listed.

MECHANICAL

To define a prescribed velocity for each space direction, select one or more of the **Prescribed in x direction**, **Prescribed in y direction**, and **Prescribed in z direction** checkboxes. Then enter a value or expression for the prescribed velocities u_{0x}, u_{0y}, or u_{0z} (SI unit: m/s), here given in a 3D component and for the **Global coordinate system**

selection. The prescribed values that you specify are interpreted in the selected **Coordinate system** and space dimension.

Select the **Continuous pressure** checkbox if the velocity condition is to be used as a source where the pressure should not be slit.

THERMAL

Select a **Thermal condition** — **Isothermal** (the default) or **Adiabatic**. See the Isothermal and Adiabatic conditions for further details.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For the **Interior Velocity** condition the **Constraint method** is set to **Nodal**.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Interior Temperature Variation

Use the **Interior Temperature Variation** condition to model a thermal type of source on an interior boundaries, $T_t = T_{bnd}$. This can, for example, be a thin sheet that has a harmonically varying temperature, given by T_{bnd}, for example, due to ohmic heating. This temperature fluctuation will give rise to acoustic waves that can be modeled using the Thermoviscous Acoustics interface and this boundary condition. Add this node from the **Interior Conditions** submenu.

MECHANICAL

Select a **Mechanical condition** — **No slip** (the default) or **Slip (perfect)**. See the No Slip and Slip (Perfect) conditions for further details.

THERMAL

Enter a value or expression for the **Temperature variation** T_{bnd} (SI unit: K) that has to be applied at the interior boundary.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Interior Lumped Speaker Boundary

The **Interior Lumped Speaker Boundary** is used to model a miniature loudspeaker (dust cap, cone, surround assembly, and motor) or another microtransducer using a lumped representation with a coupling to an **Electrical Circuit** interface. Add this node from the **Interior Conditions** submenu. The mechanical and electric properties of the speaker can, for example, be prescribed through a Thiele-Small representation and associated lumped parameters. The interior version of the Lumped Speaker Boundary includes the effects of the fluid domains on both sides of the speaker boundary explicitly. For time domain modeling including possible nonlinear lumped effects, see the Lumped Speaker Boundary and Interior Lumped Speaker Boundary of The Thermoviscous Acoustics, Transient Interface.

When the **Interior Lumped Speaker Boundary** is placed next to a **Wall** or **Interior Wall**, with **No slip** as the **Mechanical condition**, it is necessary to exclude common points (in 2D axisymmetry) or common edges (in 3D). This is done in the **Excluded Points/Edges** section. To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

SPEAKER GEOMETRY

Define how the **Speaker area** (projected) of the speaker is computed by selecting either **Selected boundaries** (the default) if all boundaries are present, or **Use symmetries** if the speaker surface is only partially represented.

When **Use symmetries** is selected, select the **Speaker area multiplication factor** A_{scale} as **Automatic** (the default, the model is analyzed for the presence of symmetry conditions); or **User defined** and enter a value (default is 1). The settings are required in order to compute the effective area of the speaker which is used to compute the acoustic load and radiated power.

Define the **Speaker axis direction** $\mathbf{e}_{\text{ax}}$ by selecting **Automatic** (the default) or **User defined**. For the automatic option the axis is computed as the average of the surface normals, this option is valid if a full speaker surface is selected. The **User defined** option

should be used if the speaker surface is only partially represented and is the only option when **Use symmetries** is selected.

For the **Automatic** option select **Reverse automatic speaker axis direction** if necessary. The axis orientation can be visualized by plotting the axis components `ta.ilsb1.e_axx`, `ta.ilsb1.e_axy`, and `ta.ilsb1.e_axz`. (using the proper item and physics scope).

For the **Circuit**, **Mechanical**, **Thermal**, **Constraint Settings**, **Exclude Edges**, and **Exclude Points** sections see the Lumped Speaker Boundary.

Interior Slip Wall

Use the **Interior Slip Wall** condition to model an interior wall with the same formulation as the Slip Wall condition. Add this node from the **Interior Conditions** submenu. This condition can be used to model thin interior walls to the fluid domain. The settings are the same as for the Slip Wall condition.

Surface Tension

The **Surface Tension** feature adds the necessary condition to model an interface between two fluids including surface tension effects. Add this node from the **Interior Conditions** submenu. This acoustic (perturbation) formulation of the Young-Laplace equation relies on a linearization around the stationary shape of the interface. The feature is important when modeling small curved interfaces between two different immiscible fluids, like microbubbles or microdrops.

The condition prescribes a jump in the normal stress at the interface between two fluids, while the temperature variation and acoustic velocity is continuous. The condition reads

$$\mathbf{n} \cdot (\mathbf{T}_1 - \mathbf{T}_2) = -\sigma \kappa^{\mathrm{aco}}$$

$$\mathbf{T} = -p_{\mathrm{t}}\mathbf{I} + \mathbf{K}$$

$$\kappa^{\mathrm{aco}} = d_{\mathrm{n}}(\kappa_1^2 + \kappa_2^2) + \nabla_{||}^2 d_{\mathrm{n}} \qquad d_{\mathrm{n}} = \frac{1}{i\omega}(\mathbf{u}_{\mathrm{t}} \cdot \mathbf{n})$$

where σ is the surface tension coefficient, $\mathbf{T}$ is the total stress in the fluid, and κ^{aco} is the acoustic perturbation to the steady state curvature of the interface. The steady state main curvatures of the interface are κ_1 and κ_2. The normal acoustic displacement of the interface is d_{n}.

SURFACE TENSION

Select the **Surface tension coefficient** from the list of built-in options or enter a user-defined value. Select to pick from the **Library coefficients, for liquid/gas interfaces** and then select from the built-in list, pick from the **Library coefficients, for liquid/liquid interfaces** and select from the built-in list, or select **User defined**. For **User defined** enter the surface tension coefficient σ (SI unit: N/m).

If performing an eigenfrequency analysis where the **Surface Tension** is present, some care has to be taken to ensure good numerical behavior. It is necessary to change the scaling factor used for the equations. To change the scaling factor select the **Thermoviscous Acoustics, Frequency Domain** node and expand the Equation section. Set the **Equation form** to **Frequency domain**, and then locate the **Thermoviscous Acoustics Equations Settings** section and set the **Scaling factor** to `ta.iomega`.

Eigenmodes in Air Bubble with Surface Tension. Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/eigenmodes_air_bubble**

No Slip

The **No Slip** node sets up a no-slip condition for the acoustic velocity fluctuations on a hard wall. Add this node from the **Mechanical** submenu. The condition reads $\mathbf{u}_t = \mathbf{0}$.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Mechanical and thermal boundary conditions contribute such that a condition can be set on the velocity and temperature simultaneously (they do not override). Thermal type condition override each other and the same is true for mechanical type conditions.

Slip (Perfect)

Use the **Slip (perfect)** node to prescribe zero normal velocity, and therefore also zero tangential stress, condition on the boundary. Add this node from the **Mechanical** submenu. This represents the ideal or perfect slip case, as opposed to nonideal condition that can be modeled with the Slip Wall feature. The condition reads

$$\mathbf{n} \cdot \mathbf{u}_t = \mathbf{0}$$

$$\sigma_n - (\sigma_n \cdot \mathbf{n})\mathbf{n} = \mathbf{0}$$

$$\sigma_n = \left[-p_t\mathbf{I} + \mu(\nabla \mathbf{u}_t + (\nabla \mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla \cdot \mathbf{u}_t)\mathbf{I}\right]\mathbf{n}$$

This results in a no-penetration condition where no viscous boundary layer is created. Use the condition in places where the viscous losses in the boundary layer are unimportant. Here, it is not necessary to mesh the boundary layer, resulting in fewer mesh elements.

When running a model with a perfectly matched layer (PML), it is recommended to not combine a **Slip (perfect)** condition with a rational scaling in the PML, as this will lead to a singularity and an error.

Per default, the **Slip (perfect)** condition uses a Nitsche formulation (also often referred to as discontinuous Galerkin or penalty formulation) to prevent locking problems on curved surfaces.

CONSTRAINT SETTINGS

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

Select **Weak constraints** as the **Constraint** if you want to switch from the default Nitsche formulation (**Automatic**) to a Lagrange multiplier formulation for the perfect slip condition (this formulation is not suited for an iterative solver). The Lagrange multiplier formulation also prevents locking problems.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Velocity

Use the **Velocity** node to define the prescribed velocities $\mathbf{u}_0$ on the boundary. Add this node from the **Mechanical** submenu. The condition reads $\mathbf{u}_t = \mathbf{u}_0$. This condition is useful, for example, when modeling a vibrating wall/source.

COORDINATE SYSTEM SELECTION

Select the **Coordinate system** to define the orientation of the prescribed quantities (the velocity). The default is the **Global coordinate system**, any other applicable system will be listed.

VELOCITY

To define a prescribed velocity for each space direction, select one or more of the **Prescribed in x direction**, **Prescribed in y direction**, and **Prescribed in z direction** checkboxes. Then enter a value or expression for the prescribed velocities u_{0x}, u_{0y}, or u_{0z} (SI unit: m/s), here given in a 3D component and for the **Global coordinate system selection**. The prescribed values that you specify are interpreted in the selected **Coordinate system** and space dimension.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For the **Velocity** condition the **Constraint method** is set to **Nodal**.

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

No Stress

Use the **No Stress** node to set the total surface stress equal to zero. Add this node from the **Mechanical** submenu. The condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = \mathbf{0}$$

Boundary Stress

Use the **Boundary Stress** node to define the total surface (boundary) stress through a stress tensor, a traction force, or a pressure. Add this node from the **Mechanical** submenu. The stress condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = \sigma_p\mathbf{n}$$

where σ_p is a symmetric stress tensor, the product $\sigma_p\mathbf{n}$ defines a traction force. The traction condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = \mathbf{F}_{tr}$$

where $\mathbf{F}_{tr}$ is the surface traction force. The pressure condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -p_{bnd}\mathbf{n}$$

where p_{bnd} is the pressure applied to the boundary (acting in the normal direction).

STRESS

Select a **Stress type** — **Stress tensor**, **Traction** (the default), or **Pressure**. For **Stress tensor** enter the (symmetric) **Stress tensor** σ_p (SI unit: N/m^2) components, for **Traction** enter the traction force $\mathbf{F}_{tr}$ (SI unit: N/m^2) components, and for **Pressure** enter the boundary pressure p_{bnd} (SI unit: Pa).

Impedance

Use the **Impedance** node to specify a normal specific impedance Z_n on a boundary. Add this node from the **Mechanical** submenu. The condition has a user defined option as well as the option to define a simple RCL impedance model. The latter is useful for modeling the interaction with, for example, a microphone represented by a simple mass-spring-damper system.

The boundary condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -Z_n(\mathbf{u}_t\cdot\mathbf{n})\mathbf{n}$$

The condition relates the normal stress at the boundary to the normal velocity, through the value if the specific impedance. The tangential velocity can optionally be constrained to zero.

MECHANICAL

Select the Impedance model as **User defined** (the default) or **RCL**. For a transient model only the **User defined** option exists.

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

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

  Notice the matching diagram and **Equation** section information for each choice. Then enter the **Equivalent acoustic resistance** R_{ac} (SI unit: kg/(m^4·s)), the **Equivalent acoustic compliance** C_{ac} (SI unit: m^4·s^2/kg) and the **Equivalent acoustic inertance** L_{ac} (SI unit: kg/m^4).

Select a condition for the **Tangential velocity** — **Slip (perfect)** (the default) or **No slip**. The **No Slip** option will apply an additional constraint for the velocity in the tangential direction on the boundary. When **Slip (perfect)** is selected, the impedance mimics the behavior of a corresponding pressure acoustics model with a normal impedance condition.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Isothermal

Use the **Isothermal** node to model a wall that is assumed to be a good thermal conductor and backed by a large heat reservoir kept at constant temperature. This implies that the harmonic temperature variations vanish: $T_t = 0$. Add this node from the **Thermal** submenu.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Adiabatic

Use the **Adiabatic** node to define a situation with no heat flow into or out of the boundary. Add this node from the **Thermal** submenu. The condition reads

$$-\mathbf{n} \cdot (-k\nabla T_t) = 0$$

Temperature Variation

Use the **Temperature Variation** node to define the temperature variation on the boundary T_{bnd}. Add this node from the **Thermal** submenu. In the frequency domain, this is the amplitude of a harmonic temperature variation, $T_t = T_{bnd}$.

TEMPERATURE VARIATION

Enter a value or expression for the **Temperature variation** T_{bnd} (SI unit: K).

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Coupling the temperature field in the fluid to, for example, a detailed heat transfer in solids model can be achieved using the Thermoviscous Acoustic–Thermal Perturbation Boundary multiphysics coupling.

Heat Flux

Use the **Heat Flux** node to define a thermal source given by an inward normal heat flux $q_n = -\mathbf{n} \cdot \mathbf{q}$ through the boundary. Add this node from the **Thermal** submenu. The condition reads

$$-\mathbf{n} \cdot (-k\nabla T_t) = q_n$$

HEAT FLUX

Enter a value for the **Inward normal heat flux** q_n (SI unit: W/m^2).

The Thermoviscous Acoustics, Transient Interface

The **Thermoviscous Acoustics, Transient (tatd)** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, is used to compute the transient evolution of the acoustic variations in pressure, velocity, and temperature. The interface is the time domain equivalent of The Thermoviscous Acoustics, Frequency Domain Interface. This physics interface is required to accurately model acoustics in geometries of small dimensions, of millimeter and submillimeter size, often referred to as microacoustics. Near walls, viscous losses and dissipation due to thermal conduction become important because boundary layers exist. The thicknesses of these boundary layers are known as the viscous and thermal penetration depth. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations. It is, for example, used when modeling the response of transducers like microphones, miniature loudspeakers and receivers. Other applications include analyzing feedback in hearing aids, smartphones, and other mobile devices, and studying the damped vibrations of MEMS structures.

The physics interface solves the equations in the time domain. The model can be extended to model nonlinear effects by adding the Nonlinear Thermoviscous Acoustics Contributions feature. In the time domain, it is also possible to model nonlinear effects due to topology changes, like nonlinear squeeze film damping. This is achieved when combining the interface with the *Moving Mesh* functionality.

The equations defined by the Thermoviscous Acoustics, Transient interface are the first and second order perturbation formulation of the Navier–Stokes equations in quiescent background conditions solving the continuity, momentum, and energy equations. Due to the detailed description necessary when modeling thermoviscous acoustics, the model simultaneously solves for the acoustic pressure p, the acoustic velocity variation $\mathbf{u}$ (particle velocity), and the acoustic temperature variations T. It is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries.

The Thermoviscous Acoustics, Transient interface is, as the frequency domain variant, formulated in the so-called scattered field formulation where the total field (subscript t) is the sum of the scattered field (the field solved for, p, $\mathbf{u}$, and T) and a possible background acoustic field (subscript b), such that

$$p_t = p + p_b \qquad \mathbf{u}_t = \mathbf{u} + \mathbf{u}_b \qquad T_t = T + T_b$$

The scattered field formulation is not applicable in domains where the Nonlinear Thermoviscous Acoustics Contributions are included. When no Background Acoustic Fields feature is present (the background field values are zero per default) the total field is simply the acoustic field solved for, $p_t = p$, $\mathbf{u}_t = \mathbf{u}$, and $T_t = T$. All governing equations and boundary conditions are formulated in the total field variables.

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

For good modeling strategies, solver suggestions, postprocessing information, as well as tips and tricks, see the Modeling with the Thermoviscous Acoustics Branch section.

For more details about the physics interface see On the Thermoviscous Acoustics Interface subsection, under The Thermoviscous Acoustics, Frequency Domain Interface section. Details about the governing equations are found in the Theory Background for the Thermoviscous Acoustics Branch section.

SETTINGS

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

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

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

EQUATION

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

- For **Study controlled**, the scaling of the equations is optimized for the numerical performance of the different solvers and study types.

THERMOVISCOUS ACOUSTICS EQUATIONS SETTINGS

See Thermoviscous Acoustics Equation Settings.

STABILIZATION

To display this section, click the **Show More Options** button ( ) and select **Stabilization**. Select **No stabilization applied** (the default), **Galerkin least-squares (GLS) stabilization**, or **Streamline upwind Petrov-Galerkin (SUPG) stabilization**. When linear thermoviscous acoustic problems are solved, the problem is stable (with the default P1-P2-P2 discretization), but as soon as the Nonlinear Thermoviscous Acoustics Contributions feature is used, stabilization may be required. For weakly nonlinear problems, no stabilization is necessary, but for moderate and highly nonlinear problems using stabilization is essential. In most of those cases, use the **Galerkin least-squares (GLS) stabilization** option.

TRANSIENT SOLVER AND MESH SETTINGS

Enter the **Maximum frequency to resolve** to be resolved by the solver and the physics-controlled mesh (when used) in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. Select the **Time stepping** (method) as **Fixed (preferred)** the default and recommended or **Free**. The **Free** option is in general not recommended for wave problems. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

Details about Transient Solver Settings are found in the section Modeling with the Thermoviscous Acoustics Branch.

DISCRETIZATION

From the list, select the element order and type (Lagrange or serendipity) for the **Pressure**, the **Velocity field**, and the **Temperature variation**, respectively. The default is **Linear** for the pressure and **Quadratic serendipity** for the velocity and the temperature.

- For numerical stability reasons, the element order for the pressure should one less than the element order for the velocity. Unless Stabilization is enabled, in which case, for example, a P1-P1-P1 discretization can be used.
- In fluids where the thermal and viscous boundary layer thickness are of the same order of magnitude (where the Prandtl number Pr is of the order 1, like in air), it is recommended to use the same shape order for the temperature and the velocity. Both fields vary equally over the same length scale in the acoustic boundary layers near walls.

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

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Pressure** p, the **Velocity field** $\mathbf{u}$ and its components, and the **Temperature variation** T. The names can be changed but the names of fields and dependent variables must be unique within a model.

Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Transient Interface

The Thermoviscous Acoustics, Transient Interface has these domain, boundary, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

	In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

	The Continuity node with this physics interface is available as a pair boundary condition. This gives continuity in pressure, temperature variation, velocity, and in the flux on a pair boundary between thermoviscous acoustic domains.

All of the boundary conditions available with the Thermoviscous Acoustics, Transient interface are the same as for the Frequency Domain version, and are described there (see the following list). Some of the following nodes are available from the **Mechanical** and **Thermal** submenus (listed in alphabetical order):

- Adiabatic[1]
- Axial Symmetry[1]
- Background Acoustic Fields
- Boundary Stress[1]
- Heat Flux[1]
- Heat Source[1]
- Initial Values[1]
- Interior Lumped Speaker Boundary
- Interior Impedance[1]
- Interior Temperature Variation[1]
- Interior Velocity[1]
- Interior Wall[1]
- Isothermal[1]
- Nonlinear Thermoviscous Acoustics Contributions
- Impedance
- Lumped Speaker Boundary
- No Slip[1]
- No Stress[1]
- Periodic Condition[1]
- Pressure (Adiabatic)[1]
- Slip (Perfect)[1]
- Slip Wall[1]
- Surface Tension[1]
- Symmetry[1]
- Temperature Variation[1]
- Thermoviscous Acoustics Model
- Velocity[1]
- Wall[1]

[1]Described for the Thermoviscous Acoustics, Frequency Domain interface

Thermoviscous Acoustics Model

Use the **Thermoviscous Acoustics Model** node to define the model inputs (the background equilibrium temperature and pressure) and the material properties of the fluid (equilibrium density, dynamic viscosity, bulk viscosity, thermal conductivity, and heat capacity at constant pressure) necessary to model the transient propagation of acoustic compressible waves in a thermoviscous acoustic context. Extended inputs are available for the coefficient of thermal expansion and the compressibility, which enables modeling of any constitutive relation for the fluid.

The equations solved are the linear (first order) perturbation equations for the conservation of mass, momentum, and energy. The model also assumes a linear equation of state relating acoustic density, pressure, and temperature fluctuations. The governing equations are given by:

$$\frac{\partial \rho_t}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u}_t) = 0$$

$$\rho_0 \frac{\partial \mathbf{u}_t}{\partial t} = \nabla \cdot \sigma + \mathbf{F} \qquad \sigma = -p_t \mathbf{I} + \mu(\nabla \mathbf{u}_t + (\nabla \mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla \cdot \mathbf{u}_t)\mathbf{I} \tag{6-1}$$

$$\rho_0 C_p \left(\frac{\partial T_t}{\partial t} + \mathbf{u}_t \cdot \nabla T_0\right) - T_0 \alpha_p \left(\frac{\partial p_t}{\partial t} + \mathbf{u}_t \cdot \nabla p_0\right) = -\nabla \cdot (-k \nabla T_t) + Q$$

$$\rho_t = \rho_0(\beta_T p_t - \alpha_p T_t)$$

the equations are formulated in terms of the total acoustic fields p_t, $\mathbf{u}_t$, and T_t. If the Background Acoustic Fields feature is added, for modeling scattering problems or as a source, the total fields are the sum of the background acoustic fields and the scattered acoustic fields. The equations can be extended to include nonlinear effects by adding the Nonlinear Thermoviscous Acoustics Contributions.

The **Model Inputs**, the **Thermoviscous Acoustics Model**, and the **Thermal Expansion and Compressibility** sections are the same as for the frequency domain interface. They are described under the Thermoviscous Acoustics Model section in The Thermoviscous Acoustics, Frequency Domain Interface.

Note that for the transient interface, it is equally important as for the frequency domain interface, to set the material parameters correctly. The values of the coefficient of thermal expansion and the isothermal compressibility are prone to errors, so make sure to take a thorough look at the Thermal Expansion and Compressibility section.

Nonlinear Thermoviscous Acoustics Contributions

The **Nonlinear Thermoviscous Acoustics Contributions** feature adds the necessary contributions to the governing Equation 6-1 in order to model nonlinear effects in a transient thermoviscous simulation. The contributions allow modeling of vortex shedding that may happen at sudden expansions, like in a perforate, a grill, or at a miniature sound port. Vortex shedding will in general introduce distortion in the measured response of a system, with the generation of harmonics. The feature can also capture the nonlinear effect associated with high sound pressure levels that require a nonlinear representation of the equation of state (pressure, density, and temperature relation). The nonlinear contributions to the left-hand side of Equation 6-1 are:

$$\ldots + \nabla \cdot (\rho_t \mathbf{u}_t) = \ldots$$

$$\ldots + \rho_t \frac{\partial \mathbf{u}_t}{\partial t} + \rho_0(\mathbf{u}_t \cdot \nabla \mathbf{u}_t) = \ldots$$

$$\ldots + C_p\left(\rho_t \frac{\partial T_t}{\partial t} + \rho_0(\mathbf{u}_t \cdot \nabla T_t) + \rho_t(\mathbf{u}_t \cdot \nabla T_0)\right)$$

$$-\alpha_p\left(T_t \frac{\partial p_t}{\partial t} + T_0(\mathbf{u}_t \cdot \nabla p_t) + T_t(\mathbf{u}_t \cdot \nabla p_0)\right) = \ldots$$

These represent all the second order nonlinear contributions, of the acoustic perturbation fields, to the governing equations (keeping a linear equation of state). This includes the nonlinear convective terms that are necessary to model vortex shedding and other similar effects with detachment. The contributions are important for modeling local high particle velocity situations, when the linearity condition $|\mathbf{u}_t| << c$ is no longer fulfilled. The contributions to the energy equation are not added when the **Adiabatic formulation** is used.

- *Nonlinear Slit Resonator*. Application Library path **Acoustics_Module/Nonlinear_Acoustics/nonlinear_slit_resonator**
- *Nonlinear Transfer Impedance of a Tapered Orifice*. Application Library path **Acoustics_Module/Nonlinear_Acoustics/nonlinear_transfer_impedance**

For systems where the linearity condition on the density $\rho_t << \rho_0$ is no longer fulfilled, the density expansion can be changed to include second order terms (see below). This will allow modeling of so-called cumulative effects.

The **Nonlinear Thermoviscous Acoustics Contributions** feature is not compatible with the Background Acoustic Fields feature. The superposition principle is not valid in a nonlinear model.

Note that when solving nonlinear models, it is often necessary to use numerical stabilization. Turn it on in the Stabilization section. Per default no stabilization is used. Using stabilization also allows to switch to a P1-P1-P1 discretization which can be more efficient in transient models. Remember to use an adequate mesh for a lower order discretization, especially in the acoustic boundary layers.

MODEL INPUTS

The model inputs for the **Equilibrium pressure** p_0 and the **Equilibrium temperature** T_0 are always visible as they contribute to the governing equations.

DENSITY REPRESENTATION

In models with high local sound pressure levels, the linear equation of state may no longer be valid. This happens when the linearity condition on the density $\rho_t << \rho_0$ is no longer fulfilled. If necessary, change the default **Density expansion** from **First order** to **Second order**.

When **Second order** is selected, additional inputs to the model are necessary. For the general case, the second order derivatives of the equilibrium density $\rho_0 = \rho_0(p_0,T_0)$ with respect to pressure p_0 an temperature T_0 are needed. They contribute to the second-order Taylor expansion of the density. Per default, they are taken **From equilibrium density**; this implies that the dependency of the density on pressure and temperature should be correct.

If the **Adiabatic** formulation is used (see Thermoviscous Acoustics Equation Settings), the user interface inputs correspond to the Nonlinear Acoustics (Westervelt) Contributions equation in Pressure Acoustics, Transient. Select to specify the **Parameter of nonlinearity** (default), the **Ratio of specific heats (for gases)**, or the **Coefficient of nonlinearity**.

VISCOUS HEATING

Click to select **Include viscous dissipation** (disabled per default). This will add a right-hand side heat source to the energy equation. The viscous dissipation is a nonlinear (second order) effect and can only be included in the nonlinear model.

Background Acoustic Fields

When the **Background Acoustic Fields** feature is added to a domain, it is possible to define the value of the background acoustic field variables p_b, $\mathbf{u}_b$, and T_b. Using this feature, it is possible to set up scattering problems as well as defining acoustic fields at an inlet of a waveguide (using a small domain at the inlet). The feature cannot be used together with the Nonlinear Thermoviscous Acoustics Contributions, as splitting into background and scattered field is not possible for nonlinear equations.

BACKGROUND ACOUSTIC FIELDS

Enter expressions for the **Background acoustic pressure** p_b, the **Background acoustic velocity** $\mathbf{u}_b$, and the **Background temperature variation** T_b. These can be analytical expressions or values of dependent variables solved in a previous study that defines the background field. The time-domain version of the background acoustics field feature does not include the plane wave option available for the frequency domain interface.

Impedance

Use the **Impedance** condition to specify a normal specific impedance Z_n on a boundary. Add this node from the **Mechanical** submenu. The boundary condition reads

$$\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\mathbf{n} = -Z_n(\mathbf{u}_t\cdot\mathbf{n})\mathbf{n}$$

The condition relates the normal stress at the boundary to the normal velocity, through the value if the specific impedance. The tangential velocity can optionally be constrained to zero.

MECHANICAL

Enter a value or expression for the **Specific impedance** Z_n (SI unit: Pa·s/m).

Select a condition for the **Tangential velocity** — **Slip (perfect)** (the default) or **No slip**. The **No Slip** option will apply an additional constraint for the velocity in the tangential direction on the boundary. When **Slip (perfect)** is selected, the impedance mimics the behavior of a corresponding pressure acoustics model with a normal impedance condition.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

Lumped Speaker Boundary

The **Lumped Speaker Boundary** condition is used to model a loudspeaker (dust cap, cone, surround assembly, and motor) or another transducer using a lumped representation with a coupling to an **Electrical Circuit** interface. The mechanical and electric properties of the speaker can, for example, be described through a Thiele-Small parameter representation and associated lumped parameters.

In the time domain, nonlinear effects can be included in the lumped model through parameters that depend on the axial position x or velocity v of the speaker. These are, for example, the typically measured $BL(x)$, $C_{MS}(x)$, or $R_{MS}(v)$. Predefined variables exist for the axial velocity `tatd.lsb1.v_ax` and an ODE is solved for the associated axial position `tatd.lsb1.x_ax` (use the appropriate physics and feature tag). These variables can be directly used in expressions in the electric components in the **Electrical Circuit** model as they are globally defined.

The properties of a back volume can also be modeled using a lumped representation through either a simple equivalent acoustic compliance or a more general RCL circuit representation. On the other hand, if the air domains on both sides of the speaker are modeled explicitly (included in the model) then there are two possible setups:

1. If the speaker diaphragm is represented by an interior boundary (a "shell" with air domains on both sides): use the Interior Lumped Speaker Boundary condition.
2. If the speaker diaphragm is represented by a domain (a "solid" with air domains on both sides): use two **Lumped Speaker Boundary** features, one applied to the upper side and one to the lower side. Make sure the speaker axis is the same on both sides and that the back volume correction is disabled (the **No correction** option). Finally, in the **Electrical Circuit** add two **External I vs. U** nodes in series, when setting up the coupling.

For the **Speaker Geometry**, **Circuit**, **Mechanical**, **Thermal**, **Constraint Settings**, **Exclude Edges**, and **Exclude Points** sections, see Lumped Speaker Boundary in The Thermoviscous Acoustics, Frequency Domain Interface.

For the **Back Volume Correction** and the **Initial Value** sections, see Lumped Speaker Boundary in The Pressure Acoustics, Transient Interface.

!	When the **Lumped Speaker Boundary** is placed next to a **Wall** or **Interior Wall**, with **No slip** as the **Mechanical condition**, it is necessary to exclude common points (in 2D axisymmetry) or common edges (in 3D). This is done in the **Excluded Points/Edges** section. To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

Interior Lumped Speaker Boundary

The **Interior Lumped Speaker Boundary** is used to model a microtransducer (dust cap, cone, surround assembly, and motor) or another transducer using a lumped representation with a coupling to an **Electrical Circuit** interface. The mechanical and electric properties of the speaker can, for example, be described through a Thiele-Small representation and associated lumped parameters. The interior version of the Lumped Speaker Boundary includes the effects of the fluid domains on both sides of the speaker boundary explicitly. Add this node from the **Interior Conditions** submenu.

In the time domain, nonlinear effects can be included in the lumped model through parameters that depend on the axial position x or velocity v of the speaker. These are, for example, the typically measured $BL(x)$, $C_{MS}(x)$, or $R_{MS}(v)$. Predefined variables exist for the axial velocity `actd.ilsb1.v_ax` and an ODE is solved for the associated axial position `actd.ilsb1.x_ax` (use the appropriate physics and feature tag). These variables can be directly used in expressions in the electric components in the **Electrical Circuit** model as they are globally defined

For the **Speaker Geometry**, **Circuit**, **Mechanical**, **Thermal**, **Constraint Settings**, **Exclude Edges**, and **Exclude Points** sections see Interior Lumped Speaker Boundary in The Thermoviscous Acoustics, Frequency Domain Interface.

For the **Initial Value** sections see Interior Lumped Speaker Boundary in The Pressure Acoustics, Transient Interface.

!	When the **Interior Lumped Speaker Boundary** is placed next to a **Wall** or **Interior Wall**, with **No slip** as the **Mechanical condition**, it is necessary to exclude common points (in 2D axisymmetry) or common edges (in 3D). This is done in the **Excluded Points/Edges** section. To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

The Thermoviscous Acoustics, Boundary Mode Interface

The **Thermoviscous Acoustics, Boundary Mode (tabm)** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, is used to compute and identify propagating and nonpropagating modes in waveguides and ducts. The interface performs a boundary mode analysis on a given boundary, including the thermal and viscous loss effects that are important in the acoustic boundary layer near walls.

The interface is applied at boundaries which represent the cross section or the inlet of a waveguide or duct of small dimensions. It solves for the acoustic variations of pressure p, velocity $\mathbf{u}$, and temperature T, as well as the out-of-plane wave number k_{n} of the modes. Near walls, viscous losses and thermal conduction become important because boundary layers exist. The thickness of these layers is known as the viscous and thermal penetration depth. For this reason, it is necessary to include thermal conduction effects and viscous losses explicitly in the governing equations. The Thermoviscous Acoustics, Boundary Mode interface is, for example, used when setting up sources in systems with small ducts like hearing aids or mobile devices. It can also be used to identify the propagating wave number and characteristic impedance of a duct cross section and use that information in the homogenized Narrow Region Acoustics model in The Pressure Acoustics, Frequency Domain Interface.

The propagation wave number is defined by the postprocessing variable `tabm.kn` and the (lumped) characteristic impedance by the variable `tabm.Zc`. Both are global variables.

The Thermoviscous Acoustics, Boundary Mode interface solves the equations defined by the linearized Navier–Stokes equations (linearized continuity, momentum, and energy equations), in quiescent background conditions, on boundaries, searching for the out-of-plane wave numbers at a given frequency. All gradients in the governing equations are expressed in terms of the in-plane gradient and the out-of-plane wave number that is being solved for. Due to the detailed description necessary when modeling thermoviscous acoustics, the model simultaneously solves for the acoustic pressure p, the velocity variation $\mathbf{u}$ (particle velocity), and the acoustic temperature

variations T. The interface is available on boundaries for 3D and on edges for 2D axisymmetric geometries.

For details about the governing equations, see the Theory Background for the Thermoviscous Acoustics Branch.

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

SETTINGS

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

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

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

EQUATION

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

- For **Study controlled**, the frequency used for the mode analysis study is given in the study.
- For **Mode analysis**, you can set the frequency manually. The default **Mode analysis frequency** f is 100 Hz.

SOUND PRESSURE LEVEL SETTINGS

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

DISCRETIZATION

From the list, select the element order for the **Pressure**, the **Velocity field**, and the **Temperature variation**. The default uses **Linear** elements for the pressure and **Quadratic serendipity** for the velocity field and the temperature variations.

In order for the system to be numerically stable, it is important that the order for the pressure degree of freedom (DOF) is one lower than the velocity field. Per default, the velocity components and the temperature share the same element order as they vary similarly over the same length scale in the acoustic boundary layer. Therefore, both typically require the same spatial accuracy. There is no stabilization option for the boundary mode interface.

Increase the element order to, for example, quadratic for the pressure and cubic for the velocity and temperature dofs. This can add additional accuracy but it also adds additional degrees of freedom compared to the default setting and will be computationally more expensive.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Pressure** p, the **Velocity field** $\mathbf{u}$ and its components, and the **Temperature variation** T. The names can be changed but the names of fields and dependent variables must be unique within a model.

- Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface
- Theory Background for the Thermoviscous Acoustics Branch

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Boundary Mode Interface

The Thermoviscous Acoustics, Boundary Mode Interface has these boundary and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

	In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

	The Continuity node with this physics interface is available as a pair boundary condition. This gives continuity in pressure, temperature variation, velocity and in the flux on a pair boundary between thermoviscous acoustics domains.

All of the boundary conditions available with the Thermoviscous Acoustics, Boundary Mode interface are the same as for the Frequency Domain and Transient version, and are described there (see the following list). The Thermoviscous Acoustics Model

feature is described here. Some of the following nodes are available from the **Mechanical** and **Thermal** submenus (listed in alphabetical order):

- Adiabatic[1]
- Boundary Stress[1]
- Heat Flux[1]
- Heat Source[1]
- Initial Values[1]
- Isothermal[1]
- Impedance
- No Slip[1]
- No Stress[1]
- Pressure (Adiabatic)[1]
- Slip (Perfect)[1]
- Symmetry[1]
- Temperature Variation[1]
- Thermoviscous Acoustics Model
- Velocity[1]
- Wall[1]

[1]Described for the Thermoviscous Acoustics, Frequency Domain interface

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Thermoviscous Acoustics Model

Use the **Thermoviscous Acoustics Model** node to define the model inputs (the background equilibrium temperature and pressure) and the material properties of the fluid (dynamic viscosity, bulk viscosity, thermal conductivity, heat capacity at constant pressure, and equilibrium density) necessary to model the propagation of acoustic compressible waves in a thermoviscous acoustic context. Extended inputs are available for the coefficient of thermal expansion and the compressibility, which enables modeling of any constitutive relation for the fluid.

The **Model Inputs**, the **Thermoviscous Acoustics Model**, and the **Thermal Expansion and Compressibility** sections are the same as for the frequency domain and transient interfaces. They are described under the Thermoviscous Acoustics Model section in The Thermoviscous Acoustics, Frequency Domain Interface.

Note that for the boundary mode interface, it is equally important as for the frequency domain and transient interfaces to set the material parameters correctly. The values of the coefficient of thermal expansion and the isothermal compressibility are prone to

errors, so make sure to take a thorough look at the Thermal Expansion and Compressibility section.

BOUNDARY MODE SETTINGS

This section contains an input for the **Propagation direction** $\mathbf{n}$ of the modes analyzed with the interface. Typically, this direction is equal to the surface or boundary normal. The default value is (`tabm.nx`, `tabm,ny`, `tabm,nz`) in 3D and (`tabm.nr`, `tabm.nz`) in 2D axisymmetric. When setting up models, this vector defines the positive propagation direction for the waves and thus the sign of the wave number.

Impedance

Use the **Impedance** node to specify a normal specific impedance Z_n on a boundary. Add this node from the **Mechanical** submenu. The condition has a user defined option for specifying the impedance.

MECHANICAL

Specify the **Specific impedance** Z_n (SI unit: Pa·s/m). The default value is the characteristic specific impedance of the medium `tabm.rho0*tabm.c`.

CONSTRAINT SETTINGS

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

EXCLUDED EDGES/POINTS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. See Suppressing Constraints on Lower Dimensions for details.

The Thermoviscous Acoustics, SLNS Approximation Interface

The **Thermoviscous Acoustics, SLNS Approximation (slns)** interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, is used to compute the propagation of acoustic waves using the so-called *Sequential Linearized Navier-Stokes* or *SLNS* approximation to include the thermoviscous boundary layer losses in a computationally efficient manner. The interface solves the governing equations in the frequency domain.

The SLNS formulation is based on a Helmholtz decomposition of the full set of governing equations of the The Thermoviscous Acoustics, Frequency Domain Interface (also sometimes referred to as FLNS or full linearized Navier-Stokes). The resulting three scalar Helmholtz equations are solved sequentially. These are for the viscous scaling function Ψ_v, the thermal scaling function Ψ_{th}, and the acoustic pressure p. The method is computationally efficient and will capture most thermoviscous losses correctly. This interface is in particular suited for large system simulations. The main assumption for the decomposition is that the acoustical wavelength is much larger than the thickness of the viscous and the thermal boundary layers.

The formulation does not capture losses fully in regions with so-called redirectional flow. This happens at sudden geometry changes, like a sudden expansion. For many models, these additional losses will be minimal compared to the pure boundary layer losses in the remaining regions of the model. The method solves the thermoviscous problem, by including all the rotational (incompressible) flow components into the boundary layers only, through the scaling functions. In regions with redirectional flow, rotational components exist outside the boundary layers and are not captured. For detailed modeling of, for example, perforates or microperforated plates use the full formulation of The Thermoviscous Acoustics, Frequency Domain Interface.

In some sense, the Thermoviscous Acoustics, SLNS Approximation interface is a generalization of pressure acoustics with the Narrow Region functionality and the Thermoviscous Boundary Layer Impedance functionality, for generalized geometries.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. The harmonic variation of all fields and sources is given by $e^{i\omega t}$ using the $+i\omega$ convention. Linear acoustics is assumed.

The Thermoviscous Acoustics, SLNS Approximation interface is available for 3D, 2D, and 2D axisymmetric geometries.

The Thermoviscous Acoustics, SLNS Approximation interface is formulated in the scattered field formulation where the total field (subscript `t`) is the sum of the scattered field and a possible background acoustic field (subscript `b`).

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Thermoviscous Acoustics Model**, **Wall**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Thermoviscous Acoustics, SLNS Approximation** to select physics features from the context menu.

SETTINGS

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

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

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

EQUATION

See settings for Equation for the Pressure Acoustics, Frequency Domain interface.

THERMOVISCOUS ACOUSTICS EQUATION SETTINGS

If needed select the **Adiabatic formulation** of the equations.

SOUND PRESSURE LEVEL SETTINGS

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

GLOBAL PORT SETTINGS

See settings for Global Port Settings for the Pressure Acoustics, Frequency Domain interface.

DISCRETIZATION

From the list, select the element order for the **Pressure**, the **Viscous scaling function**, and the **Thermal scaling function**. The default uses **Quadratic** (Lagrange) for all the fields.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields), the **Acoustic pressure** p (`p`), the **Viscous scaling function** Ψ_v (`Psiv`), and the **Thermal scaling function** Ψ_{th} (`Psith`). The names can be changed but the names of fields and dependent variables must be unique within a model.

For details about the governing equations see the Theory for the SLNS (Sequential Linearized Navier–Stokes) Formulation in the Theory Background for the Thermoviscous Acoustics Branch section.

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

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

Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, SLNS Approximation Interface

The Thermoviscous Acoustics, SLNS Approximation Interface has these boundary and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

The **Continuity** node with this physics interface is available as a pair boundary condition. This gives continuity in the pressure, the scaling functions, and the flux on a pair boundary between thermoviscous acoustic domains.

The boundary conditions available with the Thermoviscous Acoustics, SLNS Approximation interface include a subset of the conditions available for The Pressure Acoustics, Frequency Domain Interface (see the following list) as well as a dedicated Wall feature. Special **Override** and **Contribution** rules apply in order for boundary conditions to include the desired thermoviscous mechanical *no slip* or *slip (perfect)* as well as thermal *isothermal* or *adiabatic* behavior.

- Axial Symmetry
- Background Pressure Field
- Dipole Domain Source
- Impedance
- Initial Values
- Lumped Port
- Lumped Speaker Boundary
- Monopole Domain Source
- Normal Acceleration
- Normal Displacement
- Normal Velocity
- Periodic Condition
- Pressure
- Symmetry
- Thermoviscous Acoustics Model
- Wall

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Thermoviscous Acoustics Model

Use the **Thermoviscous Acoustics Model** node to define the model inputs (the background equilibrium temperature and pressure) and the material properties of the fluid (speed of sound, density, heat capacity at constant pressure, ratio of specific heats, thermal conductivity, dynamic viscosity, and bulk viscosity) necessary to model the propagation of acoustic compressible waves in a thermoviscous acoustic context using the SLNS approach.

The model sets up the three Helmholtz equations that result from a Helmholtz decomposition of the full thermoviscous acoustic governing equations. The three equations are solved sequentially, and represent the viscous and thermal incompressible waves and the compressible acoustic wave. The main assumption for the decomposition is that the acoustical wavelength is much larger than the thickness of the viscous and thermal boundary layers.

The three equations solved are for the viscous scaling function Ψ_v, the thermal scaling function Ψ_{th}, and the total acoustic pressure p_t. The equations are

$$\begin{aligned}
\nabla\cdot(\nabla\Psi_v)+k_v^2(\Psi_v-1) &= 0 \qquad & k_v^2 &= -\frac{i\omega\rho}{\mu}\\
\nabla\cdot(\nabla\Psi_{th})+k_{th}^2(\Psi_{th}-1) &= 0 \qquad & k_{th}^2 &= -\frac{i\omega\rho C_p}{k_{cond}}\\
\nabla\cdot\left(-\frac{1}{\rho_c}(\Psi_v\nabla p_t-\mathbf{q}_d)\right)-\frac{\Psi'_{th}k^2p_t}{\rho_c} &= Q_m \qquad & k^2 &= \left(\frac{\omega}{c_c}\right)^2\\
\Psi'_{th} &= \gamma-(\gamma-1)\Psi_{th}
\end{aligned} \tag{6-2}$$

where k_v is the viscous wave number, k_{th} is the thermal wave number, and k is the wave number of the compressional acoustic wave. Ψ'_{th} is the modified thermal scaling function. For the adiabatic formulation, the modified thermal scaling function is set equal to 1 and is not solved for. The bulk thermal and viscous losses are always included through the complex-valued density ρ_c and speed of sound c_c, for details see the Thermally Conducting and/or Viscous Fluid Model for the The Pressure Acoustics, Frequency Domain Interface. Note that for notational purposes in the equation display, the thermal conductivity is denoted k_{cond}, as to not confuse the symbol with the wave number k. The condition for the validity of the Equation 6-2 reads

$$\begin{aligned}
\left|\frac{\nabla^2 p_t}{p_t}\right| &\ll \frac{1}{\delta_v^2} \qquad & \delta_v^2 &= \frac{2\mu}{\rho_0\omega}\\
\left|\frac{\nabla^2 p_t}{p_t}\right| &\ll \frac{1}{\delta_{th}^2} \qquad & \delta_{th}^2 &= \frac{2k_{cond}}{C_p\rho_0\omega}
\end{aligned}$$

where δ_v is the viscous boundary layer thickness and δ_{th} is the thermal boundary layer thickness.

The two equations for the scaling functions depend only on the geometry, frequency, and material properties, as well as the choice of **No slip** and **Slip (perfect)** condition at the Wall. The equations can be solved sequentially with the equation for the acoustic

pressure last. When the default solver is generated, the equations are solved sequentially, which gives the desired performance.

MODEL INPUTS

This section contains field variables that appear as model inputs. The fields for the **Equilibrium pressure** p_0 and the **Equilibrium temperature** T_0 are always visible. If material properties depend on these, they are automatically used. Select **User defined** (the default), **Common model input**, or an existing variable from another physics interface.

Details about the Model Input and the Default Model Inputs are found in the Global and Local Definitions chapter of the *COMSOL Multiphysics Reference Manual*.

FLUID PROPERTIES

For each of the following material properties the default values are taken **From material**. For **User defined**, enter other values or expressions for any or all options.

- **Speed of sound** c (SI unit: m/s).
- **Density** ρ (SI unit: kg/m^3).
- **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)).
- **Ratio of specific heats** γ (dimensionless).
- **Thermal conductivity** k (SI unit: W/(m·K)). Note that the quantity is called k_{cond} in the equation display.
- **Dynamic viscosity** μ (SI unit: Pa·s).
- **Bulk viscosity** μ_B (SI unit: Pa·s).

Background Pressure Field

The **Background Pressure Field** feature has the same settings and options as the Background Pressure Field in the The Pressure Acoustics, Frequency Domain Interface.

Initial Values

The **Initial Values** node adds an initial value for the pressure, the viscous scaling function, and the thermal scaling function.

Monopole Domain Source

Use the **Monopole Domain Source** node to add a user-defined domain source term Q_m to the governing equation. See the Monopole Domain Source in the The Pressure Acoustics, Frequency Domain Interface.

Dipole Domain Source

Use the **Dipole Domain Source** node to add the domain source term $\mathbf{q}_d$ to the governing equation. See the Dipole Domain Source in the The Pressure Acoustics, Frequency Domain Interface.

Axial Symmetry

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

Wall

Use the **Wall** node to model the most common conditions at solid surfaces. This is the default boundary condition. This condition contains both a mechanical and a thermal selection. The default is a no-slip and isothermal condition applicable in most cases.

For the acoustic pressure, the wall condition represents the natural Neumann condition. For the viscous scaling function, a no-slip condition represents a constraint $\Psi_v = 0$, while for the thermal scaling function, an isothermal condition represents a constraint $\Psi_{th} = 0$.

MECHANICAL

Select the **Mechanical condition** as **No slip** (the default) or **Slip (perfect)**.

The choice made for the mechanical condition is inherited by the Normal Acceleration, Normal Velocity, Normal Displacement, Impedance, and Lumped Speaker Boundary features if added to the same boundary.

THERMAL

Select the **Thermal condition** as **Isothermal** (the default) or **Adiabatic**.

The choice made for the thermal condition is inherited by the Normal Acceleration, Normal Velocity, Normal Displacement, Impedance, and Lumped Speaker Boundary features if added to the same boundary.

CONSTRAINT SETTINGS

These settings are shown when the **Advanced Physics Options** is selected. It controls how the constraints are handled.

Normal Acceleration

The **Normal Acceleration** condition adds an acceleration source at a boundary. See the Normal Acceleration in the The Pressure Acoustics, Frequency Domain Interface.

Note that the **Normal Acceleration** condition contributes with the Wall feature. It specifically inherits the defined **Mechanical condition** (**No slip** or **Slip (perfect)**) and **Thermal condition** (**Isothermal** or **Adiabatic**) from the Wall feature. For specific combinations of mechanical and thermal conditions with the source, add a separate **Wall** feature on the source boundary. When **No slip** is used together with the source, the formulation — used in the SLNS formulation for thermoviscous acoustics — can result in a solution that looks unphysical near the moving wall. However, these artifacts are only local effects that do not influence the overall results.

Normal Velocity

The **Normal Velocity** condition adds a velocity source at a boundary. See the Normal Velocity in the The Pressure Acoustics, Frequency Domain Interface.

Note that the **Normal Velocity** condition contributes with the Wall feature. It specifically inherits the defined **Mechanical condition** (**No slip** or **Slip (perfect)**) and **Thermal condition** (**Isothermal** or **Adiabatic**) from the Wall feature. For specific combinations of mechanical and thermal conditions with the source, add a separate **Wall** feature on the source boundary. When **No slip** is used together with the source, the formulation — used in the SLNS formulation for thermoviscous acoustics — can result in a solution that looks unphysical near the moving wall. However, these artifacts are only local effects that do not influence the overall results.

Normal Displacement

The **Normal Displacement** condition adds an acceleration source at a boundary. See the Normal Displacement in the The Pressure Acoustics, Frequency Domain Interface.

Note that the **Normal Displacement** condition contributes with the Wall feature. It specifically inherits the defined **Mechanical condition** (**No slip** or **Slip (perfect)**) and **Thermal condition** (**Isothermal** or **Adiabatic**) from the Wall feature. For specific

combinations of mechanical and thermal conditions with the source, add a separate **Wall** feature on the source boundary. When **No slip** is used together with the source, the formulation — used in the SLNS formulation for thermoviscous acoustics — can result in a solution that looks unphysical near the moving wall. However, these artifacts are only local effects that do not influence the overall results.

Pressure

The **Pressure** condition adds a pressure source at a boundary. See the Pressure in the The Pressure Acoustics, Frequency Domain Interface.

For the **Pressure** condition the boundary is always assumed to have a **Slip (perfect)** mechanical behavior and an **Adiabatic** thermal behavior.

Impedance

The **Impedance** node adds an impedance boundary condition with the option to select between several built-in impedance models and engineering relations. See the Impedance in the The Pressure Acoustics, Frequency Domain Interface.

Note that the **Impedance** condition contributes with the Wall feature. It specifically inherits the defined **Mechanical condition** (**No slip** or **Slip (perfect)**) and **Thermal condition** (**Isothermal** or **Adiabatic**) from the Wall feature. For specific combinations of mechanical and thermal conditions with the **Impedance** condition, add a separate **Wall** feature on the source boundary.

Symmetry

The **Symmetry** node adds a boundary condition where there is symmetry in the pressure as well as the thermal and viscous scaling fields. Use this condition to reduce the size of a model by cutting it in half where there are symmetries.

Periodic Condition

The **Periodic Condition** node adds a periodic boundary condition that can be used to reduce the model size by using symmetries and periodicities in the geometry and physics modeled. See the Periodic Condition in the The Pressure Acoustics, Frequency Domain Interface.

Lumped Port

The **Lumped Port** feature connects a waveguide or duct inlet/outlet to a lumped representation element. This can be an *Electrical Circuit*, a two-port network defined through a transfer matrix, or a lumped representation of a waveguide. Basically, it couples the end of a waveguide to an exterior system that has a given lumped representation. See the Lumped Port in the The Pressure Acoustics, Frequency Domain Interface.

For the **Lumped Port** condition the boundary is always assumed to have a **Slip (perfect)** mechanical behavior and an **Adiabatic** thermal behavior.

Lumped Speaker Boundary

The **Lumped Speaker Boundary** condition is used to model a loudspeaker (dust cap, cone, surround assembly, and motor) or another transducer using a lumped representation with a coupling to an **Electrical Circuit** interface. See the Lumped Speaker Boundary in the The Pressure Acoustics, Frequency Domain Interface.

Note that the **Lumped Speaker Boundary** condition contributes with the Wall feature. It specifically inherits the defined **Mechanical condition** (**No slip** or **Slip (perfect)**) and **Thermal condition** (**Isothermal** or **Adiabatic**) from the Wall feature. For specific combinations of mechanical and thermal conditions with the source, add a separate **Wall** feature on the source boundary. When **No slip** is used together with the source, the formulation — used in the SLNS formulation for thermoviscous acoustics — can result in a solution that looks unphysical near the moving wall. However, these artifacts are only local effects that do not influence the overall results.

The Acoustic–Thermoviscous Acoustic Interaction, Frequency Domain Interface

The **Acoustic–Thermoviscous Acoustic Interaction, Frequency Domain** multiphysics interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, combines the Thermoviscous Acoustics, Frequency Domain and Pressure Acoustics, Frequency Domain interfaces together with the Acoustic–Thermoviscous Acoustic Boundary multiphysics coupling. To couple the two physics interfaces and use Thermoviscous Acoustics only where necessary can reduce the computational cost of a model.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. Linear acoustics is assumed.

When a predefined **Acoustic–Thermoviscous Acoustic Interaction, Frequency Domain** interface is added from the **Acoustics** > **Thermoviscous Acoustics** branch of the **Model Wizard** or the **Add Physics** windows, the **Thermoviscous Acoustics, Frequency Domain** and **Pressure Acoustics, Frequency Domain** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling **Acoustic–Thermoviscous Acoustic Boundary**.

	For details about all the multiphysics couplings in the Acoustics Module, see Multiphysics Couplings.

	The use of multiphysics couplings helps improve the flexibility and design options for your modeling. For details, see The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual.*

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When a predefined multiphysics interface is used, for example **Acoustic–Thermoviscous Acoustic Interaction, Frequency Domain**, the **Selection** on the multiphysics couplings is automatically set to **All boundaries**. In this way, the multiphysics coupling is

automatically active on all boundaries with **Thermoviscous Acoustics, Frequency Domain** on one side and **Pressure Acoustics, Frequency Domain** on the other.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available multiphysics couplings, but you need to manually select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Coupling

See Acoustic–Thermoviscous Acoustic Boundary in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Thermoviscous Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface.
- The available physics features for The Pressure Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface

- Theory Background for the Pressure Acoustics Branch
- Theory Background for the Thermoviscous Acoustics Branch

The Thermoviscous Acoustic–Solid Interaction, Frequency Domain Interface

The **Thermoviscous Acoustic–Solid Interaction, Frequency Domain** multiphysics interface ( ), found under the **Thermoviscous Acoustics** branch ( ) when adding a physics interface, combines the Thermoviscous Acoustics, Frequency Domain and Solid Mechanics interfaces. The physics interface solves for, and has predefined couplings between, the displacement field in the solid and the acoustic variations in the fluid domains.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. Linear acoustics is assumed.

When a predefined **Thermoviscous Acoustic–Solid Interaction, Frequency Domain** interface is added from the **Acoustics** > **Thermoviscous Acoustics** branch of the **Model Wizard** or the **Add Physics** windows, the **Thermoviscous Acoustics, Frequency Domain** and **Solid Mechanics** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling **Thermoviscous Acoustic–Structure Boundary**.

For details about all the multiphysics couplings in the Acoustics Module, see Multiphysics Couplings.

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When a predefined multiphysics interface is used, for example **Thermoviscous Acoustic–Solid Interaction, Frequency Domain**, the **Selection** on the multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all boundaries with **Thermoviscous Acoustics, Frequency Domain** on one side and **Solid Mechanics** on the other.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. You can choose the available multiphysics couplings, but you need to manually select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Coupling

See Thermoviscous Acoustic–Structure Boundary in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Thermoviscous Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface.

- Theory Background for the Thermoviscous Acoustics Branch

- Regarding the available structural-mechanics features and their theory background, see The Solid Mechanics Interface.

The Thermoviscous Acoustic–Shell Interaction, Frequency Domain Interface

This physics interface requires a Structural Mechanics Module license. For theory and physics interface feature descriptions relating to the Shell interface, see the *Structural Mechanics Module User's Guide*.

The **Thermoviscous Acoustic–Shell Interaction, Frequency Domain** multiphysics interface ( ), found under the **Acoustics** > **Thermoviscous Acoustics** branch ( ) when adding a physics interface, combines the Thermoviscous Acoustics, Frequency Domain and Shell interfaces. The physics interface solves for and has a predefined coupling between the displacement field of the shell and the acoustic variations in the fluid domains. It can be used, for example, for modeling the vibrating response of micromirrors in MEMS applications. The physics interface is available for 3D and 2D axisymmetric geometries, and it is capable of modeling the coupled thermoviscous acoustics and shell vibrations in the frequency domain.

The physics interface solves the equations in the frequency domain assuming all fields and sources to be harmonic. Linear acoustics is assumed.

When a predefined **Thermoviscous Acoustic–Shell Interaction, Frequency Domain** interface is added from the **Acoustics** > **Thermoviscous Acoustics** branch of the **Model Wizard** or the **Add Physics** windows, the **Thermoviscous Acoustics, Frequency Domain** and **Shell** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling **Thermoviscous Acoustic–Structure Boundary**. This multiphysics coupling automatically couples the thermoviscous acoustic domain to exterior shells (on exterior boundaries) and to interior shells (on interior boundaries).

For details about all the multiphysics couplings in the Acoustics Module, see Multiphysics Couplings.

SETTINGS FOR PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

When a predefined multiphysics interface is used, for example **Thermoviscous Acoustic–Shell Interaction, Frequency Domain**, the **Selection** on the multiphysics coupling is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all boundaries with **Thermoviscous Acoustics, Frequency Domain** on one side (exterior shells) or on both sides (interior shells) of **Shell** boundaries.

However, if physics interfaces are added one at a time, followed by the multiphysics couplings, these modified settings are not automatically included.

For example, if the single interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose the available multiphysics couplings, but you need to manually select on which boundaries they need to be applied, or select **All boundaries** to recover the predefined behavior.

Multiphysics couplings are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

PHYSICS INTERFACES AND MULTIPHYSICS COUPLINGS

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Multiphysics Coupling

See Thermoviscous Acoustic–Structure Boundary in the Multiphysics Couplings chapter.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

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

- The available physics features for The Thermoviscous Acoustics, Frequency Domain Interface are listed in the section Domain, Boundary, and Pair Nodes for the Thermoviscous Acoustics, Frequency Domain Interface.
- The available physics features for The Shell and Plate Interfaces are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*.

- Theory Background for the Thermoviscous Acoustics Branch
- Theory for the Shell and Plate Interfaces in the *Structural Mechanics Module User's Guide*

The Thermoviscous Acoustic–Thermoelasticity Interaction Multiphysics Interfaces

These two multiphysics interfaces require the MEMS Module license. For theory and physics interface feature descriptions relating to the Thermoelasticity multiphysics interface, see the *Structural Mechanics Module User's Guide*.

The **Thermoviscous Acoustic–Thermoelasticity Interaction, Frequency Domain** ( ) and the **Thermoviscous Acoustic–Thermoelasticity Interaction, Transient** ( ) multiphysics interfaces, found under the **Acoustics** > **Thermoviscous Acoustics** branch ( ) when adding a physics interface, combines the Thermoviscous Acoustics, Frequency Domain or Transient interface with the Thermoelasticity multiphysics interface. The physics interface solves for and has predefined multiphysics couplings between the displacement and temperature field in the solid and the acoustic variations in pressure, velocity, and temperature in the fluid domain.

It can be used, for example, for modeling the vibrating response of MEMS devices including detailed description of the damping. The physics interface is available for 3D, 2D, and 2D axisymmetric geometries.

- In the *frequency domain*, the physics interface solves the equations using a frequency-domain perturbation approach assuming all fields and sources to be harmonic. The displacement and temperature field in the solid are linearized around a stationary solution. Linear acoustics is assumed.
- In the *time domain*, the physics interface solves the equations in the time domain by coupling the deviation from reference in the solid to the acoustic fields in the fluid.

Prestressed Micromirror Vibrations: Thermoviscous–Thermoelasticity Coupling. Application Library path **Acoustics_Module/Vibrations_and_FSI/micromirror_prestressed_vibration**

The multiphysics interface combines The Thermoviscous Acoustics, Frequency Domain Interface or The Thermoviscous Acoustics, Transient Interface with The Thermoelasticity Interface, using the Thermoviscous Acoustic–Thermoelasticity Boundary multiphysics coupling.

FREQUENCY DOMAIN

When solving the multiphysics problem in the frequency domain, it is necessary to use a frequency-domain perturbation approach. The reason is that the Heat Transfer in Solids physics (and Solid mechanics) is formulated in total fields, whereas the acoustic physics are formulated in perturbation variables. To solve the model, two approaches exist:

- First solve a **Stationary** study step followed by the **Frequency-Domain Perturbation** study step. In the stationary step, solve for the background temperature field in the solid (and potentially also the displacement). This solution is then taken as the linearization point for the frequency-domain perturbation problem.
- If the background temperature in the solid can be assumed constant, it is possible to only use the **Frequency-Domain Perturbation** step. In this case, change the **Value of linearization point** to **User controlled**, the linearization point will be taken from the **Initial Values** node in the heat transfer physics interface.

TIME DOMAIN

In the time domain, the situation is a bit different. Since the heat transfer interface solves for the total fields, the perturbation temperature is defined as the difference between the temperature solved for `T` and the reference temperature `ht.Tref`. The reference temperature is defined at the physics interface level in the **Physical Model** section. This value is typically a constant, but the value can be taken from a previously solved stationary problem. In that case, the reference value should also be used as the initial value for the transient simulation.

Modeling with the Thermoviscous Acoustics Branch

This section contains modeling tips and tricks, information about solvers, postprocessing variables, and meshing.

In this section:

- Physics-Controlled Mesh
- Meshing the Boundary Layer
- Solver Suggestions for Large Thermoviscous Acoustics Models
- Lagrange and Serendipity Shape Functions
- Transient Solver Settings
- Postprocessing Variables
- Suppressing Constraints on Lower Dimensions

Physics-Controlled Mesh

When modeling acoustic problems using the Thermoviscous Acoustics, Frequency Domain or the Thermoviscous Acoustics, Transient interfaces the computational mesh can be automatically set up using the **Physics-controlled mesh** functionality. For these two physics, the generated computational mesh will follow best practices. The mesh will automatically resolve the wave, a boundary layer mesh is added at walls, perfectly matched layer (PML) domains will have an appropriate mapped mesh, and periodic boundaries will be set up with copy mesh operations.

In the Settings window for the **Physics-Controlled Mesh** on the Mesh node in the Model Builder Tree the following controls exist. The **Maximum mesh element size control parameter** sets the maximum mesh element size. The **From physics** setting will use the speed of sound and the maximum frequency from the study (frequency domain) or the **Maximum frequency to resolve** (transient) to set a mesh that resolves the wavelength. The **Frequency** option lets you enter a specific **Maximum frequency** to resolve. For both options you can select the **Number of mesh elements per wavelength**. Finally, the **User defined** lets you set an actual maximum mesh size. The last option is often the most useful in microacoustic applications, as the wavelength is typically automatically

resolved, while it is the geometry that has to be resolved. Use this option to, for example, enter a length scale comparable to a fraction of a narrow waveguide diameter.

If a model contains a **Parametric Sweep**, the physics-controlled mesh will be regenerated for each new parameter value. This is of course important, if the parameter changes the geometry, the minimum wavelength, or the boundary layer thickness in the model. For example, if a sweep is done over a number of frequency bands, a proper mesh should be regenerated for each band.

In cases where the parameter has no influence on the mesh, remeshing will still take place, and may slow down the solution process. In this case, it is advised to instead use an **Auxiliary Sweep** (an inner sweep) found in the **Study Extensions** section.

Meshing the Boundary Layer

When modeling using the Thermoviscous Acoustics interfaces, several length scales become important when setting up the mesh.

- First, there is the wavelength which should still be resolved as in pressure acoustics, see Meshing: Resolving the Waves in Space in the Pressure Acoustics Interfaces chapter. This length scale is often resolved as the geometries are small when modeling thermoviscous acoustic phenomena in microacoustic applications.
- Secondly, there is the thickness of the viscous and thermal boundary layers. In order for the model to include the correct amount of damping, the boundary layers need to be resolved. Ideally this is done using a **Boundary Layers** mesh. The **Thickness of first boundary layer** (or **All layers**) and the **Number of boundary layers** should be set such that they resolve the boundary layer at the specific modeling frequency or for the frequency range. Remember that the boundary layer thickness scales as one over the square root of the frequency.

  The thermal and viscous boundary layer thickness length scale exists as a variable. In the frequency domain interface, `ta.d_visc` and `ta.d_therm` can be evaluated as function of frequency. In the transient interface, `tatd.d_visc` and `tatd.d_therm` are evaluated at the frequency given by **The maximum frequency to resolve** setting in Transient Solver and Mesh Settings.
- Finally, it is important to consider the thickness of the boundary layer compared to the physical dimensions of the model, for example, the viscous boundary layer

thickness δ_v compared to the tube radius a in a waveguide. This is sometimes known as the Womersley number

$$\mathrm{Wo} = \sqrt{\frac{\omega\rho a^2}{\mu}} = \frac{a\sqrt{2}}{\delta_v} = \frac{a}{\delta_{th}}\sqrt{\frac{2}{\mathrm{Pr}}}$$

where δ_v is the viscous boundary layer thickness, δ_{th} is the thermal boundary layer thickness, and Pr is the Prandtl number.

When the boundary layer is small compared to the geometry, the Womersley number is large, say $\mathrm{Wo} > 10$. Then the effect associated with the losses in the viscous boundary layer can normally be disregarded. In this case, the boundary layer do not need to be meshed and a **Slip** condition can be used instead of a **No-slip** condition. The same is true for the thermal boundary layer thickness compared to the tube radius. Here, an **Isothermal** condition can be replaced by an **Adiabatic** condition. If the **No-slip** or **Isothermal** conditions are still kept, then remember to add at least one boundary layer mesh that is of roughly the size of the acoustic boundary layer. If this is not done, erroneous losses can be introduced in the model.

On the other hand, whenever Wo is $O(1)$ or smaller, the boundary layer effects need to be included and the boundary layer needs to be meshed adequately as mentioned above.

A great deal of information about meshing and modeling with the Thermoviscous acoustics interface can be found in two COMSOL Blog posts. They are:

- https://www.comsol.com/blogs/theory-of-thermoviscous-acoustics-thermal-and-viscous-losses/
- https://www.comsol.com/blogs/how-to-model-thermoviscous-acoustics-in-comsol-multiphysics/

See the Boundary Layers section in the *COMSOL Multiphysics Reference Manual* for more details.

Solver Suggestions for Large Thermoviscous Acoustics Models

Solving thermoviscous acoustic problems can easily involve solving for many degrees of freedom (DOFs) as the model solves for the acoustic variations in both pressure,

velocity field (2 or 3 components), and temperature. First of all, it is important to restrict the use of the thermoviscous acoustics model to domains and regions of the model where it is necessary and couple to Pressure Acoustics using the multiphysics coupling. In the frequency domain, use the Narrow Region Acoustics feature or the Thermoviscous Boundary Layer Impedance feature of Pressure Acoustics to reduce the number of DOFs. Secondly, care should be taken when meshing the computational domain. If these two things have been carefully considered, the solver can be changed from its default *Direct* setting to use an *Iterative* solver suggestion. Depending on the model size and involved physics, two options are described below.

In both cases, a good starting point for setting up a new solver configuration is to right-click the study node and select **Show Default Solver**, then expand the **Solver Configuration** tree under **Stationary Solver** or **Time-Dependent Solver**. Predefined iterative solver suggestions are automatically generated. Per default, a direct solver is used and two iterative solvers are suggested and disabled. To turn on one of these approaches, right-click the solver and select **Enable** (or press F4).

- The first suggestion *(GMRES with Direct Precon.)* uses an iterative solver with two direct preconditioners. This method is typically faster than the direct solver and uses 20% less memory.
- The second suggestion *(GMRES with DD)* uses an iterative solver with the domain decomposition method. This method is very robust (also for multiphysics applications) and very memory efficient. The method can be slow if modified to be as memory efficient as possible (see below).

Both suggestions are described below. In liquids where thermal effects can be neglected, the model can be solved in the **Adiabatic formulation** and DOFs saved. Choosing different shape functions can also reduce the memory consumption, for example, switching from the default P1-P2-P2 discretization to an all linear discretization, if **Stabilization** is enabled.

- Iterative Solver with Direct Preconditioner using Hybridization
- Iterative Solver with Domain Decomposition
- Solving in the Adiabatic Case
- Choosing Shape Functions

ITERATIVE SOLVER WITH DIRECT PRECONDITIONER USING HYBRIDIZATION

The first iterative solver suggestion *(GMRES with Direct Precon.)* is useful for moderate 3D problems, that only involve thermoviscous acoustics. Using this solver

suggestion will save around 20% memory and can speed up the solution procedure by a factor 2 or 3.

The manual setup of the suggestion can be done as follows: Under the *Stationary Solver* node take the following steps: Add an **Iterative** solver with the GMRES solver. As preconditioner add the **Direct Preconditioner** and switch the solver to PARDISO. Expand the **Hybridization** section and select **Multi preconditioner** in the **Preconditioner variables** list add the *Pressure* and *Velocity field*. Add a second direct preconditioner with the same settings but now select only the *Temperature* as preconditioner variables. Set the **Pivoting Perturbation** to 1e-13 for both the pressure-velocity group and the temperature group. The reason for splitting the equations up in this manner is that the energy equation is only loosely coupled to the momentum and continuity equations.

See the Direct Preconditioner section in the *COMSOL Multiphysics Reference Manual* for more details.

The direct preconditioner approach is used in the model: *Transfer Impedance of a Perforate*. Application Library path **Acoustics_Module/Tutorials/transfer_impedance_perforate**

ITERATIVE SOLVER WITH DOMAIN DECOMPOSITION

The second iterative solver suggestion *(GMRES with DD)*, uses the more advanced *Domain Decomposition* approach, to handle large and very large 3D models. The method can be tuned depending on the hardware available. Two settings for the Domain Decomposition preconditioner are of interest:

- The **Maximum number of DOFs per subdomain** can be increased/decreased based on the memory available on your system.
- The **Recompute and clear subdomain data** can be set to **On** (instead of the default **Off**) if it is necessary to run a very memory-lean version of the solver. This will in turn increase the computation time. The **Recompute and clear subdomain data** is a list with the options **Automatic**, **Off**, and **On**. If the option **Automatic** is chosen, the recompute and clear mechanism is activated if there is an out-of-memory error during the domain decomposition setup phase. The setup is then repeated with recompute and clear activated (which can be costly in terms of time but is better than failure). A warning is given in this case. Use the default suggested **Off** option for an efficient solver of large problems when there is sufficient memory resources.

This type of approach should make it possible to solve large thermoviscous acoustics models, also including multiphysics interactions, using a minimum of RAM.

See the Domain Decomposition (Schwarz) section in the *COMSOL Multiphysics Reference Manual* for more details

SOLVING IN THE ADIABATIC CASE

In certain cases, it is a good assumption to not include thermal conduction in the model and treat all processes as adiabatic (isentropic). This is, for example, relevant for fluids where the thermal boundary layer is much thinner than the viscous, like in water. Not solving for the temperature field T also saves some degrees of freedom (DOFs).

This is achieved by selecting the **Adiabatic formulation** option under the Thermoviscous Acoustics Equation Settings section. When **Adiabatic formulation** is selected, all temperature options and conditions are disabled in the user interface.

CHOOSING SHAPE FUNCTIONS

In models with a structured mesh the default serendipity shape functions will generate less degrees of freedom over choosing Lagrange elements; see Lagrange and Serendipity Shape Functions below. In general, if a boundary layer mesh is used (to resolve the thermal and viscous boundary layers) or if a PML is used in the model, the mesh contains structured mesh regions.

Lagrange and Serendipity Shape Functions

In most of the physics interfaces in the Acoustics Module, and specifically in the Thermoviscous Acoustic interfaces, you can choose between two families of shape functions in the **Discretization** section: *Lagrange* and *serendipity*. The current default (except for the Thermoviscous Acoustics, SLNS approximation interface) is to use serendipity shape functions.

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

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

- 2D: Quadrilateral elements of discretization order higher than 1.
- 3D: Hexahedral, prism, and pyramid elements of discretization order higher than 1.

In the *COMSOL Multiphysics Reference Manual*:

- The Lagrange Element (shlag)
- The Nodal Serendipity Element (shnserp)

When coupling two physics interfaces that have the same DOFs like, for example, displacement, the same type of shape functions should be used in both interfaces to ensure conformity. Since there is no difference between the two families of shape functions in 1D, this is not an issue when connecting edges.

Transient Solver Settings

When solving transient wave problems, it is recommended to use fixed time stepping in the solver (the manual option in the solver configuration). In general, there is no point in using an automatic time-step control, which can be provided by the time-dependent solver. The tolerances in the automatic error control are difficult to tune in wave problems when there is weak but important high-frequency content. It is recommended to use the suggestion generated when the **Fixed (preferred)** method is chosen (the default) in the **Transient Solver Settings** section. The internal time step of the solver that is generated, when the **Maximal frequency to resolve** is set, will in most cases produce a solver that is adequate. It is assuming that the user has generated a mesh that properly resolves the same maximal frequency (minimal wavelength).

For further details see the Time Stepping: Resolving the Waves in Time section in Modeling with the Thermoviscous Acoustics Branch.

Postprocessing Variables

This section contains information about variables for:

- Viscous and Thermal Boundary Layer Variables
- Material Properties

- Stress Tensor Components
- Power Dissipation and Intensity Variables
- In and Out-of-Plane Variables for the Boundary Mode Interface

In the variable names, in the tables below, *phys_id* represents the interface name. For example, `ta` for the frequency domain interface, `tatd` for the transient interface, and `tabm` for the boundary mode interface.

VISCOUS AND THERMAL BOUNDARY LAYER VARIABLES

The viscous and thermal boundary layer thickness (thermal and viscous penetration depth) as defined in Theory Background for the Thermoviscous Acoustics Branch can be evaluated in postprocessing for the frequency-domain models. The same is the case for the Prandtl number relating the two length scales (available in both frequency and time domain).

TABLE 6-2: BOUNDARY LAYER VARIABLES.

VARIABLE	DESCRIPTION
phys_id.d_visc	Viscous boundary layer thickness
phys_id.d_therm	Thermal boundary layer thickness
phys_id.Pr	Prandtl number

MATERIAL PROPERTIES

Material properties are readily available for postprocessing in plots. In plots, click the **Replace Expression** icon and browse to the **Material properties** list under the thermoviscous acoustics interface. Important parameters to plot are the coefficient of thermal expansion *phys_id*.alpha0 and the isothermal compressibility *phys_id*.betaT. These should not evaluate to zero.

STRESS TENSOR COMPONENTS

The stress tensor components are defined as variables and can be evaluated in postprocessing or used to create exotic couplings between physics. In the table below only the xx-component of the stress tensor and the x-component of the stress are shown. Change the spatial reference accordingly.

TABLE 6-3: STRESS VARIABLES ONE COMPONENT IN 3D.

VARIABLE	DESCRIPTION
phys_id.K_stress_tensorxx	Viscous stress tensor, xx-component
phys_id.T_stress_tensorxx	Total stress tensor, xx-component

TABLE 6-3: STRESS VARIABLES ONE COMPONENT IN 3D.

VARIABLE	DESCRIPTION
phys_id.K_stressx	Viscous stress, x-component (on boundaries)
phys_id.T_stressx	Total stress, x-component (on boundaries)

POWER DISSIPATION AND INTENSITY VARIABLES

The thermoviscous acoustics interface models the energy dissipation process which stem from viscous and thermal dissipation processes. The amount of dissipated energy can be of interest as a results analysis variable or as a source term for a multiphysics problem.

The energy conservation-dissipation corollary describes the transport and dissipation of energy in a system (see Ref. 1 p. 516 or Ref. 6). In linear acoustics, this equation is derived by taking the dot product (scalar product) of the momentum and the velocity $\mathbf{v}$, adding it to the continuity equation, and then adding the entropy. After some manipulation and integration, the use of the divergence theorem yields Equation 6-3

$$\frac{\partial}{\partial t}\int_{\Omega} w dV + \int_{\partial\Omega} (\mathbf{i}\cdot\mathbf{n}) dA = -\int_{\Omega} \Delta dV \qquad \text{or} \qquad \frac{\partial w}{\partial t} + \nabla\cdot\mathbf{i} = -\Delta$$

$$w = \frac{1}{2}\rho_0 u^2 + \frac{1}{2}\frac{p^2}{\rho_0 c_0^2} + \frac{1}{2}\frac{\rho_0 T_0}{C_p} s^2$$

$$\mathbf{i} = p\mathbf{u} - [\mathbf{u}^{\mathsf{T}}\cdot\tau]^{\mathsf{T}} - \frac{k}{T_0} T\nabla T \tag{6-3}$$

$$\Delta = \tau{:}\nabla\mathbf{u} + \frac{k}{T_0}(\nabla T)^2 = \Delta_v + \Delta_t$$

where w is the disturbance energy of the control volume, $u = |\mathbf{u}|$ is the velocity, T is the temperature variation, p is the acoustic pressure variations, p_0 is the background equilibrium pressure, T_0 the background equilibrium temperature, ρ_0 the background density, c_0 the (isentropic) speed of sound, C_p the heat capacity at constant pressure (per unit mass), k the coefficient of thermal conduction, $\mathbf{i}$ is the instantaneous intensity (flux of energy out of a control volume), Δ is the dissipated energy per unit volume and time (SI unit: Pa/s = J/(m^{3}s) = W/m^3), s is the entropy, τ is the viscous stress tensor, $\tau : \nabla\mathbf{u}$ is the viscous dissipation function, and T indicates the transpose of a vector. Δ_v and Δ_t are the viscous and thermal contributions to the dissipation function. In Equation 6-3, we have made use of Ref. 6 for the expression for the intensity $\mathbf{I}$.

In the Thermoviscous Acoustics, Frequency Domain interface, the dissipation term Δ is directly given by the RMS value of the tensor expression

$$\Delta_v = \langle \tau : \nabla \mathbf{u} \rangle = \frac{1}{4}(\tau^* : \nabla \mathbf{u} + \tau : (\nabla \mathbf{u})^*) \quad (6\text{-}4)$$

where ":" in Equation 6-4 is the double dot operator (or total inner product) and * is the complex conjugate. In the above expressions, the time-averaged expressions for a product in the frequency domain is defined as:

$$\langle AB \rangle = \langle \mathrm{Re}(Ae^{i\omega t})\mathrm{Re}(Be^{i\omega t}) \rangle = \frac{1}{4}(A^*B + AB^*)$$

The power dissipation variables are defined in Table 6-4.

TABLE 6-4: POWER DISSIPATION VARIABLES.

VARIABLE	DESCRIPTION
phys_id.diss_therm	Thermal power dissipation density
phys_id.diss_visc	Viscous power dissipation density
phys_id.diss_tot	Total thermal and viscous power dissipation density

In the Thermoviscous Acoustics, Frequency Domain interface the (time averaged) intensity **I** is given by averaging the instantaneous intensity vector **i** in Equation 6-3 using the same time averaged products defined above. The intensity and intensity magnitude are defined in Table 6-5.

TABLE 6-5: INTENSITY VARIABLES.

VARIABLE	DESCRIPTION
phys_id.Ix	Intensity x-component (in 1D, 2D, and 3D)
phys_id.Iy	Intensity y-component (in 2D, and 3D)
phys_id.Iz	Intensity z-component (2D axisymmetric and 3D)
phys_id.Ir	Intensity r-component (2D axisymmetric)
phys_id.I_mag	Intensity magnitude

In the Thermoviscous Acoustics, Transient interface, the instantaneous intensity **i** variables is available for postprocessing. The instantaneous intensity and instantaneous intensity magnitude are defined in Table 6-6.

TABLE 6-6: INSTANTANEOUS INTENSITY VARIABLES.

VARIABLE	DESCRIPTION
phys_id.Iix	Instantaneous intensity x-component (in 1D, 2D, and 3D)
phys_id.Iiy	Instantaneous intensity y-component (in 2D, and 3D)
phys_id.Iiz	Instantaneous intensity z-component (2D axisymmetric and 3D)

TABLE 6-6: INSTANTANEOUS INTENSITY VARIABLES.

VARIABLE	DESCRIPTION
phys_id.Iir	Instantaneous intensity r-component (2D axisymmetric)
phys_id.Ii_mag	Instantaneous intensity magnitude

IN AND OUT-OF-PLANE VARIABLES FOR THE BOUNDARY MODE INTERFACE

Several dedicated variables exist for The Thermoviscous Acoustics, Boundary Mode Interface where quantities are defined in terms of their in-plane and out-of-plane values. For example, the intensity variable **I** has the following derived values

$$\mathbf{I}_{\text{ip}} = \mathbf{I} - (\mathbf{I} \cdot \mathbf{n})\mathbf{n}$$
$$\mathbf{I}_{\text{op}} = (\mathbf{I} \cdot \mathbf{n})\mathbf{n}$$

where ip stands for in-plane and op for out-of-plane. These two variables are named `tabm.Iip` and `tabm.Iop` (with spatial components `x`, `y`, and `z`). The magnitude of these two variables is given by `tabm.Iip_mag` and `tabm.Iop_mag`. In the same manner variables exist for the acceleration and the velocity.

Suppressing Constraints on Lower Dimensions

Sometimes, boundary conditions on two adjacent objects can come into conflict on a shared object (point or edge). For the boundary conditions in the thermoviscous physics that are based on constraints, you have the possibility to select that certain objects of lower dimensions should be excluded from the main selection. To do this, you must first click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. In the settings, a new section named **Excluded Edges** and/or **Excluded Points** will then appear (when applicable). In these sections, you can select geometrical objects which should be excluded from the main selection when the constraint is applied.

Velocity Condition

When using the mechanical Velocity condition you may get conflicts if it is located next to a no-slip condition (a wall). To remedy this, you can exclude the common edge or point.

Periodic Condition

When using the Periodic Condition, you may get conflicts or duplicates which makes the model either overconstrained or problematic for the automatic constraint elimination algorithm. If you are aware of such situations, it is good practice to remove

one of the potentially conflicting constraints. This could be in multiphysics models where you combine solid mechanics with thermoviscous acoustics and have edges where you both apply a **Periodic Condition** and the **Thermoviscous Acoustic–Structure Boundary** multiphysics coupling.

Particularly, if the geometry is curved there is a risk that these constraints are not identical from a numerical point of view. In this case, excluding the conflicting edge from the selected boundary will make the behavior unique and fully predictable.

Another example where constraints will come in conflict is if you want to, for example, use periodic condition on parts of the geometry using weak constraints, while keeping the default pointwise constraints on other parts. If the same mesh node has both types of constraints, the solution will fail, so you must exclude any common geometrical objects from the selection in one of the constraints.

Slip Wall

When using the Slip Wall feature you may get conflicts if it is located next to a no-slip condition (a **Wall** feature or a **Normal Acceleration/Velocity/Displacement** feature). To remedy this, you can exclude the common edge or point in the no-slip condition (**Wall** feature or a **Normal Acceleration/Velocity/Displacement** feature).

See also Excluded Surfaces, Excluded Edges, and Excluded Points in the *COMSOL Multiphysics Reference Manual*.

Theory Background for the Thermoviscous Acoustics Branch

The Thermoviscous Acoustics, Frequency Domain Interface, The Thermoviscous Acoustics, Transient Interface, and The Thermoviscous Acoustics, Boundary Mode Interface are designed for the analysis of acoustics in viscous and thermally conducting, compressible Newtonian fluids. The physics interface solves the linearized Navier–Stokes equation, the continuity equation, and the energy equation. This corresponds to a small parameter expansion of the dependent variables. The physics interface solves for the acoustic pressure variations p, the fluid velocity variations $\mathbf{u}$, and the acoustic temperature variations T. The interface uses a scattered field formulation where it is possible to define background acoustic fields. All equations and boundary conditions are formulated in the total fields and solve for the scattered fields.

The Thermoviscous Acoustics, Frequency Domain Interface is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries. The physics interface solves problems in the frequency domain, that is, Frequency Domain, Frequency-Domain Modal, and Eigenfrequency type analysis. In 2D and 1D axisymmetric systems, a Mode Analysis study is also available for the out-of-plane component.

The Thermoviscous Acoustics, Transient Interface is available for 3D, 2D, and 1D Cartesian geometries as well as for 2D and 1D axisymmetric geometries. The physics interface solves problems in the time domain using the Time Dependent analysis type.

The Thermoviscous Acoustics, Boundary Mode Interface is available on boundaries in 3D and 2D axisymmetric geometries. The physics interface helps identify and solve for propagating and nonpropagating modes using the Mode Analysis study.

In this section:

- The Viscous and Thermal Boundary Layers
- General Linearized Compressible Flow Equations
- Acoustic Perturbation and Linearization
- Scattered Field Formulation and Background Acoustic Fields
- Formulation for Eigenfrequency Studies
- Formulation for Mode Analysis in 2D and 1D Axisymmetry

- Formulation for the Boundary Mode Interface
- The Nonlinear Thermoviscous Acoustics Equations
- Theory for the SLNS (Sequential Linearized Navier–Stokes) Formulation
- Theory for the Slip Wall Boundary Condition
- References for the Thermoviscous Acoustics, Frequency Domain Interface

The Viscous and Thermal Boundary Layers

In general, a tangential harmonic oscillation of amplitude u_0 and frequency f applied to a wall at $z = 0$ creates a viscous wave of the form

$$u(z) = u_0 e^{-\sqrt{\frac{\pi f \rho_0}{\mu}}(1+i)z}$$

where, f is the frequency, ρ_0 is the static density, and μ is the dynamic viscosity. The viscous shear waves are therefore dispersive with wavelength L_v and the associated wavenumber k_v

$$L_v = 2\pi\sqrt{\frac{\mu}{\pi\rho_0 f}} = 2\pi\delta_v \qquad k_v^2 = -\frac{i\omega\rho_0}{\mu}$$

The viscous waves are highly damped since their amplitude decays exponentially with distance from the boundary (see Ref. 3). In fact, in just one wavelength, the amplitude decreases to about 1/500 of its value at the boundary. Therefore, the viscous boundary layer thickness can for most purposes be considered to be less than L_v. The length scale δ_v is the so-called viscous penetration depth or viscous boundary layer thickness.

Similarly, a harmonically oscillating temperature with amplitude T_0 and frequency f at $z = 0$ creates a thermal wave of the form

$$T(z) = T_0 e^{-\sqrt{\frac{\pi f \rho_0 C_p}{k}}(1+i)z}$$

where C_p is the (specific) heat capacity at constant pressure and k is the thermal conductivity. The wavelength and wavenumber are here

$$L_{th} = 2\pi\sqrt{\frac{k}{\pi\rho_0 f C_p}} = 2\pi\delta_{th} \qquad k_{th}^2 = -\frac{i\omega\rho_0 C_p}{k}$$

and a decay behavior similar to the viscous waves. The length scale δ_{th} is the thermal penetration depth.

The ratio of viscous wavelength to thermal wavelength is a nondimensional number related to the Prandtl number Pr, as

$$\frac{L_\mathrm{v}}{L_\mathrm{th}} = \sqrt{\frac{\mu C_p}{\mathrm{k}}} = \sqrt{\mathrm{Pr}}$$

In air, this ratio is roughly 0.8, while in water, it is closer to 2.7. Thus, at least in these important cases, the viscous and thermal boundary layers are of the same order of magnitude. Therefore, if one effect is important for a particular geometry, so is probably the other.

General Linearized Compressible Flow Equations

In general, the motion of a viscous compressible Newtonian fluid, including the energy equation, is governed by the set of equations listed in Equation 6-5 below.

$$\begin{gathered}
\frac{d\rho}{dt} + \rho(\nabla \cdot \mathbf{u}) = 0 \\
\rho\frac{d\mathbf{u}}{dt} = \nabla \cdot \sigma + \mathbf{F} \\
\rho C_\mathrm{p}\frac{dT}{dt} - \alpha_\mathrm{p} T\frac{dp}{dt} = -\nabla \cdot \mathbf{q} + \phi + Q \\
\sigma = -p\mathbf{I} + \tau = -p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^\mathrm{T}) - \left(\frac{2\mu}{3} - \mu_\mathrm{B}\right)(\nabla \cdot \mathbf{u})\mathbf{I} \\
\mathbf{q} = -\mathrm{k}\nabla T \\
\rho = \rho(p, T)
\end{gathered} \tag{6-5}$$

where the dependent variables are pressure p, velocity $\mathbf{u}$, temperature T, and density ρ. The first three equations are the continuity equation, the momentum equation (the Navier–Stokes equation), and the energy equation, respectively. The last three equation are the constitutive equations. They define the total stress tensor σ and the viscous stress tensor τ through Stokes expression, the Fourier heat conduction law, and an equation of state. See, for example, Ref. 1 to 7 for further details.

The material time derivatives (or total derivatives) d/dt are in the following expanded according to

$$\frac{dA(\mathbf{x}, t)}{dt} = \frac{\partial A}{\partial t} + \mathbf{u} \cdot \nabla A$$

where A is a dummy variable. The equation of state relates local values of pressure, density, and temperature and is therefore an algebraic equation or an ODE, rather than a PDE. A common form of state equations is to know the density as function of pressure and temperature, $\rho = \rho(p, T)$. In the following, it is assumed that the state equation has this form.

The basic properties of the fluid are the dynamic viscosity μ and thermal conductivity k. The coefficient μ_B is the bulk (or second) viscosity and describes losses due to compressibility (expansion and contraction of the fluid), where μ describes losses due to shear friction. The bulk viscosity can in some cases be used to model an empirically observed deviation from Stokes' assumption but is usually negligible compared to μ unless the motion is irrotational; see Ref. 3 and Ref. 4. These three properties are taken to be constant or at most weakly temperature-dependent. The specific heat at constant pressure C_p (per unit mass) and the (isobaric) coefficient of volumetric thermal expansion α_p (sometimes denoted α_0),

$$\alpha_p = -\frac{1}{\rho}\frac{\partial \rho}{\partial T}\bigg|_p$$

are both possibly functions of pressure and temperature.

In the energy equation

$$\phi = \tau(\mathbf{u}):S(\mathbf{u}) = \tau(\mathbf{u}):\nabla\mathbf{u}$$

is the viscous dissipation function — that is, the scalar contraction of the viscous stress tensor τ with the rate of strain tensor S. Both tensors are seen as functions of a velocity vector. If the mean velocity is zero, this term vanishes in the following linearization because it is homogeneous of second order in the velocity gradients. Otherwise, it acts as an oscillating source or sink (the term is included in the full Linearized Navier–Stokes interfaces). In the right sides of Equation 6-5, $\mathbf{F}$ and Q are a volume force and a heat source, respectively.

Acoustic Perturbation and Linearization

For small perturbations around a steady-state solution, the dependent variables and sources can be assumed to take on the following form:

$$
\begin{aligned}
\mathbf{u} &= \mathbf{u}_0(\mathbf{x}) + \mathbf{u}_1(t, \mathbf{x}) \\
p &= p_0(\mathbf{x}) + p_1(t, \mathbf{x}) \\
T &= T_0(\mathbf{x}) + T_1(t, \mathbf{x}) \\
\rho &= \rho_0(\mathbf{x}) + \rho_1(t, \mathbf{x}) \\
\mathbf{F} &= \mathbf{F}_0(\mathbf{x}) + \mathbf{F}_1(t, \mathbf{x}) \\
Q &= Q_0(\mathbf{x}) + Q_1(t, \mathbf{x})
\end{aligned}
$$

where the subscript "1" variables represent the acoustic perturbations (first-order perturbation) and subscript "0" the background mean flow quantities. Assuming zero mean flow $\mathbf{u}_0 = \mathbf{0}$ and after inserting into the governing Equation 6-5, the steady-state equations can be subtracted from the system, which is subsequently linearized to first order by ignoring terms quadratic in the acoustic variables. Dropping the subscript "1" for readability yields the thermoviscous acoustic equations:

$$
\begin{aligned}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u}) &= 0 \\
\rho_0 \frac{\partial \mathbf{u}}{\partial t} &= \nabla \cdot \left[-p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_{\mathrm{B}} \right) (\nabla \cdot \mathbf{u}) \mathbf{I} \right] + \mathbf{F} \\
\rho_0 C_{\mathrm{p}} \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T_0 \right) - T_0 \alpha_{\mathrm{p}} \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p_0 \right) &= -\nabla \cdot (-k \nabla T) + Q
\end{aligned}
\tag{6-6}
$$

where the unprimed variables are now the acoustic deviation from the steady state.

The density ρ is expressed in terms of the pressure and the temperature variations using the density differential (Taylor expansion about the steady quiescent values)

$$
\rho = p \left[\frac{\partial \rho}{\partial p} \right]_{T_0} + T \left[\frac{\partial \rho}{\partial T} \right]_{p_0} = \rho_0 (p \beta_{\mathrm{T}} - T \alpha_{\mathrm{p}})
\tag{6-7}
$$

The two thermodynamic quantities (the coefficient terms in square brackets) define the isobaric coefficient of thermal expansion α_{p} (sometimes named α_0) and the isothermal compressibility β_{T}, according to the following relations

$$
\begin{aligned}
\beta_{\mathrm{T}} &= \frac{1}{\rho_0} \left[\frac{\partial \rho_0}{\partial p} \right]_T = \frac{1}{K_{\mathrm{T}}} = \frac{1}{\rho_0 c^2} \frac{\gamma}{} = \gamma \beta_{\mathrm{s}} \\
\gamma &= \frac{C_{\mathrm{p}}}{C_{\mathrm{v}}} = \frac{K_{\mathrm{s}}}{K_{\mathrm{T}}} \\
\alpha_{\mathrm{p}} &= -\frac{1}{\rho_0} \left[\frac{\partial \rho_0}{\partial T} \right]_p
\end{aligned}
\tag{6-8}
$$

where K_s is the isentropic bulk modulus (sometimes named K_0), K_T the isothermal bulk modulus, C_v is the heat capacity at constant volume (per unit mass), c is the (isentropic) speed of sound, and γ is the ratio of specific heats (the adiabatic index). The isothermal compressibility β_T is related to the isentropic (or adiabatic) compressibility β_s (sometimes named β_0) and the coefficient of thermal expansion α_p via the thermodynamic relations

$$\beta_s = \beta_T - \frac{\alpha_p^2 T_0}{\rho_0 C_p} \qquad \beta_T = \gamma\beta_s \tag{6-9}$$

It is derived using the Maxwell relations; see, for example, Ref. 5 and Ref. 7.

From Equation 6-8 and Equation 6-9, the isothermal compressibility and the isobaric coefficient of thermal expansion can be expressed in terms of the speed of sound as

$$\beta_T = \frac{1}{\rho_0 c^2}\frac{\gamma}{} \qquad \alpha_p = \frac{1}{c}\sqrt{\frac{C_p(\gamma-1)}{T_0}} \tag{6-10}$$

The equations presented in Equation 6-6 and Equation 6-7 are the ones solved in the time domain in The Thermoviscous Acoustics, Transient Interface. Assuming small harmonic oscillations about a steady-state solution, the dependent variables and sources can be assumed to take on the following form

$$\begin{aligned} \mathbf{u} &= \mathbf{u}_0 + \mathbf{u}_1 e^{i\omega t} & p &= p_0 + p_1 e^{i\omega t} \\ T &= T_0 + T_1 e^{i\omega t} & \rho &= \rho_0 + \rho_1 e^{i\omega t} \\ \mathbf{F} &= \mathbf{F}_0 + \mathbf{F}_1 e^{i\omega t} & Q &= Q_0 + Q_1 e^{i\omega t} \end{aligned}$$

Inserting this into the governing equations and performing the linearization yield the equations solved in the frequency domain in The Thermoviscous Acoustics, Frequency Domain Interface:

$$\begin{gathered} i\omega\rho + \nabla\cdot(\rho_0\mathbf{u}) = 0 \\ i\omega\rho_0\mathbf{u} = \nabla\cdot\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_B\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right] + \mathbf{F} \\ \rho_0 C_p(i\omega T + \mathbf{u}\cdot\nabla T_0) - T_0\alpha_p(i\omega p + \mathbf{u}\cdot\nabla p_0) = -\nabla\cdot(-k\nabla T) + Q \\ \rho = \rho_0(p\beta_T - T\alpha_p) \end{gathered} \tag{6-11}$$

The system of equations implemented in the thermoviscous acoustics interfaces is further given in a scattered field formulation, as described below in Scattered Field Formulation and Background Acoustic Fields.

When the **Adiabatic formulation** option under the Thermoviscous Acoustics Equation Settings section the solved equations reduce to:

$$
\begin{aligned}
i\omega\rho + \nabla\cdot(\rho_0\mathbf{u}) &= 0\\
i\omega\rho_0\mathbf{u} &= \nabla\cdot\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_{\mathrm{B}}\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right] + \mathbf{F}\\
\rho &= \rho_0\beta_s p \qquad T = \frac{\alpha_{\mathrm{p}}T_0}{\rho_0 C_{\mathrm{p}}}p
\end{aligned}
\tag{6-12}
$$

where the temperature T depends directly on the acoustic pressure p.

IDEAL GAS

For an ideal gas, the equation of state $p = \rho RT$, where R is the specific gas constant, leads to

$$\beta_{\mathrm{T}} = \frac{1}{p_0} \qquad \alpha_0 = \frac{1}{T_0}$$

and the density

$$\rho = \rho_0\left(\frac{p}{p_0} - \frac{T}{T_0}\right)$$

Inserting these expressions and dividing the continuity equation by the reference density, Equation 6-11 in the frequency domain take on the following simplified form

$$
\begin{aligned}
i\omega\left(\frac{p}{p_0} - \frac{T}{T_0}\right) + \nabla\cdot\mathbf{u} &= 0\\
i\omega\rho_0\mathbf{u} &= \nabla\cdot\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_{\mathrm{B}}\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right]\\
i\omega(\rho_0 C_{\mathrm{p}}T - p) + \mathbf{u}\cdot\nabla p_0 + \rho_0 C_{\mathrm{p}}(\mathbf{u}\cdot\nabla T_0) &= -\nabla\cdot(-\mathrm{k}\nabla T) + Q
\end{aligned}
$$

This is, for example, the system of equations implemented in the Thermoviscous Acoustics, Frequency Domain interface when the ideal gas law is selected.

ISENTROPIC (ADIABATIC) IDEAL GAS CASE

If the process is assumed to be adiabatic and reversible — that is, isentropic — the thermal conductivity is effectively zero. Then also the temperature can be eliminated, giving for the ideal gas case:

$$
\begin{aligned}
i\omega\left(\frac{1}{p_0}-\frac{1}{\rho_0 C_p T_0}\right)p+\nabla\cdot\mathbf{u} &= 0\\
i\omega\rho_0\mathbf{u} &= \nabla\cdot\left[-p\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}})-\left(\frac{2\mu}{3}-\mu_{\mathrm{B}}\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right]
\end{aligned}
\tag{6-13}
$$

where we have assumed constant background properties. Defining the speed of sound c in analogy with the standard assumptions for linear acoustics (term in front of the pressure in the continuity equation), it is found that

$$
\frac{1}{\rho_0 c^2}=\frac{1}{p_0}-\frac{1}{\rho_0 C_p T_0}=\frac{1}{p_0}\left(1-\frac{R}{C_p}\right)=\frac{1}{\gamma p_0}
$$

or

$$
c=\sqrt{\frac{\gamma p_0}{\rho_0}}
$$

In the case with a general fluid, the corresponding relation is using Equation 6-8 and Equation 6-9:

$$
\frac{1}{\rho_0 c^2}=\frac{1}{K_0}=\frac{1}{\rho_0}\left[\frac{\partial\rho_0}{\partial p}\right]_T-\frac{T_0}{\rho_0 C_p}\left(\frac{1}{\rho_0}\left[\frac{\partial\rho_0}{\partial T}\right]_p\right)^2=\frac{1}{K_{\mathrm{T}}}-\frac{T_0\alpha_p^2}{\rho_0 C_p}
$$

where K_0 is the adiabatic bulk modulus, K_{T} the isothermal bulk modulus, and α_0 the coefficient of thermal expansion.

ISOTHERMAL CASE

If, on the other hand, the thermal conductivity is high, or the thermoviscous acoustic waves propagate in a narrow space between highly conductive walls, the temperature can be assumed to be constant (isothermal assumption) and the system of equations for an ideal gas becomes:

$$
i\omega\rho_0\mathbf{u}=\nabla\cdot\left[-p\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}})-\left(\frac{2\mu}{3}-\mu_{\mathrm{B}}\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right]
$$

$$
i\omega\left(\frac{1}{p_0}\right)p+\nabla\cdot\mathbf{u}=0
$$

which, again comparing to standard assumptions, gives

$$\frac{1}{\rho_0 c^2} = \frac{1}{p_0}$$

or equivalently

$$c = \sqrt{\frac{p_0}{\rho_0}}$$

Therefore, thermal conductivity and/or conducting walls decrease the apparent speed of sound in narrow domains.

THE HELMHOLTZ EQUATION

If the thermodynamic processes in the system are assumed to be adiabatic and viscosity can be neglected, Equation 6-13 reduces, for constant background properties, to

$$i\omega\rho_0\mathbf{u} = \nabla\cdot(-p\mathbf{I}) = \nabla p$$

$$i\omega\frac{1}{\rho_0 c^2}p + \nabla\cdot\mathbf{u} = 0$$

Now, taking the divergence of the momentum equations and inserting the expression for the divergence of the velocity, taken from the continuity equations, yields the Helmholtz equation for constant material properties:

$$\nabla^2 p + \frac{\omega^2}{c^2}p = 0$$

Scattered Field Formulation and Background Acoustic Fields

The governing equations, given above in Equation 6-8, are defined in the general scattered field formulation. Assuming that all the acoustic fields are the sum of a background (prescribed) field and the resulting scattered field, the total fields can be written as

$$p_t = p_s + p_b \qquad \mathbf{u}_t = \mathbf{u}_s + \mathbf{u}_b \qquad T_t = T_s + T_b$$
$$p \equiv p_s \qquad \mathbf{u} \equiv \mathbf{u}_s \qquad T \equiv T_s$$

where subscript t stands for total, subscript s for scattered, and subscript b for background. The scattered field is the field solved for (that is, the dependent variables p, $\mathbf{u}$, and T). If no background acoustic field is defined (the default), the scattered field is equal to the total field. The governing equations and all boundary conditions are

expressed in terms of the total fields. The equations solved for are thus, in general, for the frequency domain case given by

$$
\begin{aligned}
i\omega\rho_t + \nabla\cdot(\rho_0\mathbf{u}_t) &= 0\\
i\omega\rho_0\mathbf{u}_t &= \nabla\cdot\left[-p_t\mathbf{I} + \mu(\nabla\mathbf{u}_t + (\nabla\mathbf{u}_t)^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_{\mathrm{B}}\right)(\nabla\cdot\mathbf{u}_t)\mathbf{I}\right]\\
\rho_0 C_{\mathrm{p}}(i\omega T_t + \mathbf{u}_t\cdot\nabla T_0) - T_0\alpha_{\mathrm{p}}(i\omega p_t + \mathbf{u}_t\cdot\nabla p_0) &= -\nabla\cdot(-\mathrm{k}\nabla T_t) + Q\\
\rho_t &= \rho_0(\beta_{\mathrm{T}}p_t - \alpha_{\mathrm{p}}T_t)
\end{aligned}
\tag{6-14}
$$

The background acoustic fields are defined by adding the Background Acoustic Fields feature. The feature has a user-defined option where all fields can be entered, either as analytical expressions or defined in terms of a solution to another study or model. The background acoustic fields feature also has a plane-wave option (in the frequency domain interface) that defines a plane propagating attenuated wave.

PLANE WAVE SOLUTION

The plane wave option in the Background Acoustic Fields feature (in the frequency domain only) defines the following fields

$$
\begin{aligned}
p_{\mathrm{b}} &= |p_{\mathrm{b}}|e^{-k_{\mathrm{b}}(\mathbf{n}_k\cdot\mathbf{x})}\\
\mathbf{u}_{\mathrm{b}} &= \frac{\omega(\beta_{\mathrm{T}}p_{\mathrm{b}} - \alpha_{\mathrm{p}}T_{\mathrm{b}})}{k_{\mathrm{b}}}\mathbf{n}_k\\
T_{\mathrm{b}} &= \frac{i\omega\alpha_{\mathrm{p}}T_0 p_{\mathrm{b}}}{i\omega\rho_0 C_p + \mathrm{k}k_{\mathrm{b}}^2}
\end{aligned}
\tag{6-15}
$$

$$
k_{\mathrm{b}} = \frac{\omega}{c}\left(1 + \frac{i\omega b_{\mathrm{tv}}}{\rho_0 c^2}\right)^{\frac{1}{2}} \qquad b_{\mathrm{tv}} = \frac{4}{3}\mu + \mu_{\mathrm{B}} + \frac{(\gamma-1)\mathrm{k}}{C_{\mathrm{p}}}
$$

where k_{b} is the wave number of a plane propagating wave (in the direction $\mathbf{n}_k$) with viscous and thermal attenuation; see Ref. 3. The expressions given in Equation 6-15 are an exact solution to the governing equations Equation 6-14.

Formulation for Eigenfrequency Studies

When performing an eigenfrequency study, the governing equations (Equation 6-8) are on the form:

$$-\lambda\rho + \nabla\cdot(\rho_0\mathbf{u}) = 0$$

$$-\lambda\rho_0\mathbf{u} = \nabla\cdot\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}}) - \left(\frac{2\mu}{3} - \mu_{\mathrm{B}}\right)(\nabla\cdot\mathbf{u})\mathbf{I}\right]$$

$$-\rho_0 C_{\mathrm{p}}(\lambda T_{\mathrm{t}} + \mathbf{u}_{\mathrm{t}}\cdot\nabla T_0) + T_0\alpha_{\mathrm{p}}(\lambda p_{\mathrm{t}} + \mathbf{u}_{\mathrm{t}}\cdot\nabla p_0) = -\nabla\cdot(-\mathrm{k}\nabla T) + Q$$

$$\rho = \rho_0(\beta_{\mathrm{T}}p - \alpha_{\mathrm{p}}T)$$

where the eigenvalue is $\lambda = -i\omega$. It is important to note that there is a difference between regular pressure acoustics and thermoviscous acoustics in terms of what modes can exist and which modes are found during an eigenfrequency study. In pressure acoustics only the pure acoustic modes exist; the equations and assumptions ensured this. In thermoviscous acoustics, on the other hand, the equations are formulated for all small-signal components that can exist. This means that other nonacoustic modes also exist — that is, thermal and vorticity modes.

VORTICITY AND THERMAL MODES

When solving an eigenfrequency problem in thermoviscous acoustics, it is important to take a close look at the obtained eigenfrequencies and assess if they are acoustic or not. The nature of the solution is of the form

$$p(\mathbf{x}, t) \sim p(\mathbf{x})e^{-\lambda t} = p(\mathbf{x})e^{i\omega t} \qquad \lambda = \alpha + i\beta \qquad \omega = -\beta + i\alpha$$

where λ is the eigenvalue. Typically, eigenvalues exist near the positive real axis, where $\beta \approx 0$. These are exponentially decaying nonacoustic (nonoscillating) modes that stem from the thermal equation or the deviatoric part of the momentum equation (the nonpressure and nonvolume part of the stress tensor) also called the vorticity modes. The acoustic eigenvalues, on the other hand, lie close to the imaginary axis and are oscillating and slightly damped.

OTHER SPURIOUS MODES

Note that other spurious and nonacoustic modes can also exist when, for example, a PML layer is used to model an open boundary. These modes stem from unphysical phenomena and the scaling inside the PML layer. In all cases, it is a good idea to have an a priori knowledge of the location and type of the eigenvalues, maybe from solving a lossless pressure acoustics model, and also to look at the modes in terms of, for example, the pressure field.

Formulation for Mode Analysis in 2D and 1D Axisymmetry

The Mode Analysis study type is available for The Thermoviscous Acoustics, Frequency Domain Interface in 2D and 1D axisymmetric models. This type of study is used to determine the form of the *propagating acoustic modes* in *waveguide* structures. The analyzed 2D and 1D axisymmetric geometries can be thought of as the cross sections of a waveguide. This study is really restricted to the cross section of the waveguides. To calculate the modes in 3D structures at the inlet of the waveguide, use The Thermoviscous Acoustics, Boundary Mode Interface, described below.

The spatial dependency in the (out-of-plane) axial z direction along the waveguide is assumed to be of the form of a traveling wave with wave number k_z. The dependent variables in 2D are rewritten as

$$p = p(x,y)e^{-ik_z z}$$
$$\mathbf{u} = \mathbf{u}(x,y)e^{-ik_z z}$$
$$T = T(x,y)e^{-ik_z z}$$

and in 1D axisymmetric as

$$p = p(r)e^{-ik_z z}$$
$$\mathbf{u} = \mathbf{u}(r)e^{-ik_z z}$$
$$T = T(r)e^{-ik_z z}$$

Using this form of the dependent variables, differentiation with respect to z reduces to a multiplication with $-ik_z$. The *propagating modes* are determined by solving an eigenvalue problem in the variable $\lambda = -ik_z$.

The expression for the pressure can now be written retaining the harmonic time dependency, as

$$k_z = \beta - i\alpha$$
$$p = p(\mathbf{x})e^{-ik_z z}e^{i\omega t} = p(\mathbf{x})e^{-\alpha z}e^{i(\omega t - \beta z)}$$

where $\mathbf{x}$ is the in plane coordinate(s). The axial wave number is split into a real and an imaginary part. The imaginary part α of the wave number describes how fast the propagating modes decay along the waveguide, it is often referred to as the attenuation coefficient. In the above equation, it is assumed to be negative ($-\alpha$), leading to an exponentially decaying wave. The real part β is related to the phase speed c_{ph} of the propagating mode by $c_{ph} = \beta/\omega$. In thermoviscous acoustics, the obtained wave

numbers always have an imaginary part as the modeled system always includes losses. The relation between the angular frequency ω and the axial wave number k_z is called the dispersion relation.

- Mode Analysis in the *COMSOL Multiphysics Reference Manual*

Formulation for the Boundary Mode Interface

The Thermoviscous Acoustics, Boundary Mode Interface interface adds extended functionality to the abovementioned mode analysis available in 2D and 1D axisymmetric geometries. The interface can be applied on boundaries in 3D (and 2D axisymmetric) geometries and can be used to compute the propagating modes and out-of-plane wave number k_n on a (flat) surface of any orientation in 3D. This makes it possible to more easily set up boundary conditions at inlets of waveguides using the mode information.

The equations solved are the same as for The Thermoviscous Acoustics, Frequency Domain Interface but using modified expressions for the gradients. The implementation assumes propagation in the out-of-plane direction (that is, in the normal direction $\mathbf{n}$), and the fields can then be written as

$$p = p(\mathbf{x}_\mathrm{ip})e^{-ik_\mathrm{n}\mathbf{x}_\mathrm{op}}$$
$$\mathbf{u} = \mathbf{u}(\mathbf{x}_\mathrm{ip})e^{-ik_\mathrm{n}\mathbf{x}_\mathrm{op}}$$
$$T = T(\mathbf{x}_\mathrm{ip})e^{-ik_\mathrm{n}\mathbf{x}_\mathrm{op}}$$

where $\mathbf{x}_\mathrm{ip}$ is the in-plane coordinate, $\mathbf{x}_\mathrm{op}$ is the out-of-plane coordinate, and k_n is the wave number that this formulation computes. Because of the above assumption about the acoustic fields, the gradient operator can be split into a tangential in-plane component ($\|$) and a normal out-of-plane component ($\perp$), such that

$$\nabla p = \nabla_{||} p - ik_\mathrm{n} p\mathbf{n}$$
$$\nabla \mathbf{u} = \nabla_{||} \mathbf{u} - ik_\mathrm{n}(\mathbf{u}\cdot\mathbf{n}^\mathrm{T})$$
$$\nabla T = \nabla_{||} T - ik_\mathrm{n} T\mathbf{n}$$

The in-plane gradient is defined in COMSOL Multiphysics by the tangential derivative operator `dtang()`. The divergence of the velocity field is defined as the trace of the Jacobian, that is

$$\nabla \cdot \mathbf{u} = \text{trace}(\nabla \mathbf{u})$$

With the above modifications and redefinition of the gradient operators, solving for the propagating modes reduces to an eigenvalue problem solving for the acoustic field and the out-of-plane wave number for a given frequency.

The Nonlinear Thermoviscous Acoustics Equations

The governing equations solved in a transient thermoviscous acoustics model that uses the Nonlinear Thermoviscous Acoustics Contributions feature are also derived from Equation 6-5 using a first order perturbation approach. However nonlinear terms in the perturbations are retained. This yields the set of nonlinear thermoviscous acoustics equations

$$\frac{\partial \rho_t}{\partial t} + \nabla \cdot (\rho_0 \mathbf{u}_t) + \nabla \cdot (\rho_t \mathbf{u}_t) = 0$$

$$\rho_0 \frac{\partial \mathbf{u}_t}{\partial t} + \rho_t \frac{\partial \mathbf{u}_t}{\partial t} + \rho_0 (\mathbf{u}_t \cdot \nabla \mathbf{u}_t) = \nabla \cdot \sigma + \mathbf{F}$$

$$\rho_0 C_p \left(\frac{\partial T_t}{\partial t} + \mathbf{u}_t \cdot \nabla T_0 \right) - T_0 \alpha_p \left(\frac{\partial p_t}{\partial t} + \mathbf{u}_t \cdot \nabla p_0 \right)$$

$$\ldots + C_p \left(\rho_t \frac{\partial T_t}{\partial t} + \rho_0 (\mathbf{u}_t \cdot \nabla T_t) + \rho_t (\mathbf{u}_t \cdot \nabla T_0) \right)$$

$$\ldots - \alpha_p \left(T_t \frac{\partial p_t}{\partial t} + T_0 (\mathbf{u}_t \cdot \nabla p_t) + T_t (\mathbf{u}_t \cdot \nabla p_0) \right) = -\nabla \cdot (-k \nabla T_t) + Q$$

$$\rho_t = \rho_0 (\beta_T p_t - \alpha_p T_t)$$

$$\sigma = -p_t \mathbf{I} + \mu (\nabla \mathbf{u}_t + (\nabla \mathbf{u}_t)^T) - \left(\frac{2\mu}{3} - \mu_B \right) (\nabla \cdot \mathbf{u}_t) \mathbf{I}$$

In the default formulation only the linear equation of state is retained. This is a good approximation in most cases as cumulative nonlinear effects typically can be neglected in microacoustics. The nonlinear equations will capture local nonlinear effects like vortex shedding. These can occur when the linearity condition $|\mathbf{u}_t| << c$ is no longer fulfilled. Note that the vortices are generated by the acoustic field itself.

In models with high local sound pressure levels, the linear equation of state may no longer be valid. This happens when the linearity condition on the density $\rho_t << \rho_0$ is no longer fulfilled. If necessary, the feature has an option to change the default **Density expansion** from **First order** to **Second order**. This corresponds to the following second order equation of state

$$\rho_t = \rho_0(\beta_T p_t - \alpha_p T_t) + \frac{1}{2}p_t^2\frac{\partial^2\rho_0}{\partial p_0^2} + p_t T_t\frac{\partial^2\rho_0}{\partial p_0 \partial T_0} + \frac{1}{2}T_t^2\frac{\partial^2\rho_0}{\partial T_0^2}$$

Additional inputs to the model are necessary. For the general case, the second order derivatives of the equilibrium density $\rho_0 = \rho_0(p_0, T_0)$ with respect to pressure p_0 and temperature T_0 are needed. They contribute to the second-order Taylor expansion of the density. Per default, they are taken **From equilibrium density**; this implies that the dependency of the density on pressure and temperature should be correct and included. Note that this dependency is included if a material is defined using the functionality of the Liquid & Gas Properties Module (see the Material Properties section in the Acoustic Properties of Fluids chapter for details).

Theory for the SLNS (Sequential Linearized Navier–Stokes) Formulation

The SLNS formulation is based on a Helmholtz decomposition of the full set of governing equations of the The Thermoviscous Acoustics, Frequency Domain Interface as derived in the Acoustic Perturbation and Linearization section. The resulting three scalar Helmholtz equations are solved sequentially. These are for the viscous scaling function Ψ_v, the thermal scaling function Ψ_{th}, and the acoustic pressure p. The main assumption for the decomposition is that the acoustical wavelength is much larger than the thickness of the viscous and the thermal boundary layers. See also Ref. 11 and Ref. 12.

The three equations solved are for the viscous scaling function Ψ_v, the thermal scaling function Ψ_{th}, and the total acoustic pressure p_t. The equations are

$$\nabla \cdot (\nabla \Psi_v) + k_v^2(\Psi_v - 1) = 0 \qquad k_v^2 = -\frac{i\omega\rho}{\mu}$$

$$\nabla \cdot (\nabla \Psi_{th}) + k_{th}^2(\Psi_{th} - 1) = 0 \qquad k_{th}^2 = -\frac{i\omega\rho C_p}{k_{cond}}$$

$$\nabla \cdot \left(-\frac{1}{\rho_c}(\Psi_v \nabla p_t - \mathbf{q}_d)\right) - \frac{\Psi'_{th} k^2 p_t}{\rho_c} = Q_m \qquad k^2 = \left(\frac{\omega}{c_c}\right)^2$$

$$\Psi'_{th} = \gamma - (\gamma - 1)\Psi_{th}$$

where k_v is the viscous wave number, k_{th} is the thermal wave number, and k is the wave number of the compressional acoustic wave. Ψ'_{th} is the modified thermal scaling function. The condition, for the validity of the Helmholtz decomposition, reads

$$\left|\frac{\nabla^2 p_t}{p_t}\right| \ll \frac{1}{\delta_v^2} \qquad \delta_v^2 = \frac{2\mu}{\rho_0\omega}$$

$$\left|\frac{\nabla^2 p_t}{p_t}\right| \ll \frac{1}{\delta_{th}^2} \qquad \delta_{th}^2 = \frac{2k_{cond}}{C_p\rho_0\omega}$$

where δ_v is the viscous boundary layer thickness and δ_{th} is the thermal boundary layer thickness. This means that the wavelength of the compressible pressure wave has to be much larger than the thermal and viscous boundary layers.

Theory for the Slip Wall Boundary Condition

The Slip Wall boundary condition is a linearized version of the **Slip Velocity** option in the **Wall** boundary in the CFD interface. See the section Theory for the Wall Boundary Condition in Theory for the Single-Phase Flow Interfaces in the *CFD Module User's Guide*.

The Slip Wall boundary is useful in microscale systems where the boundary condition is rarely strictly no slip or perfect slip. There are two main effects due to noncontinuum effects: a slip velocity either due to a viscous stress or a thermal gradient along the boundary and a temperature jump due to a thermal gradient perpendicular to the boundary.

The linearized slip velocity is given as:

$$\mathbf{u}_{\text{slip}} = -\frac{\sigma_s \lambda_0}{\mu}\tau_{n,t} + \mathbf{u}_{\text{therm}}$$

$$\tau_{n,t} = \tau \cdot \mathbf{n} - (\mathbf{n}^{\mathrm{T}}\tau\mathbf{n})\mathbf{n}$$

$$\mathbf{u}_{\text{therm}} = \sigma_T \frac{\mu}{\rho_0 T_0}\left(\nabla T_t - (\mathbf{n}\cdot\nabla T_t)\mathbf{n} - \frac{\rho_0 T_t + \rho_t T_0}{\rho_0 T_0}(\nabla T_0 - (\mathbf{n}\cdot\nabla T_0)\mathbf{n})\right)$$

where σ_s is the viscous slip coefficient, λ_0 is the mean free path, σ_T is the thermal slip coefficient, $\mathbf{u}_{\text{therm}}$ is the thermal creep. There are two contributions to the slip velocity $\mathbf{u}_{\text{slip}}$: the contribution from the acoustics stress tensor τ and the thermal creep $\mathbf{u}_{\text{therm}}$ induced by the thermal gradient tangential to the boundary. The linearized temperature jump between the fluid and the boundary is given as

$$\Delta T_t = -\zeta_T(\lambda_0 \mathbf{n}\cdot\nabla T_t + \lambda_1 \mathbf{n}\cdot\nabla T_0)$$

where ζ_T is the temperature jump coefficient and λ_1 is the acoustic perturbation of the mean free path.

References for the Thermoviscous Acoustics, Frequency Domain Interface

1. W.M. Beltman, P.J.M. van der Hoogt, R.M.E.J. Spiering, and H. Tijdeman, "Implementation and Experimental Validation of a New Viscothermal Acoustic Finite Element for Acousto-Elastic Problems," *J. Sound Vib.*, vol. 216, pp. 159–185, 1998.

2. M. Malinen, M. Lyly, and others, "A Finite Element Method for the Modeling of Thermo-Viscous Effects in Acoustics," *Proc. ECCOMAS 2004*, Jyväskylä, 2004.

3. D.T. Blackstock, *Fundamentals of Physical Acoustics,* John Wiley & Sons, 2000.

4. H. Bruus, *Theoretical Microfluidics*, Oxford University Press, 2010.

5. G.K. Bachelor, *An Introduction to Fluid Dynamics*, Cambridge University Press, 2000.

6. M.K. Myers, "Transport of energy by disturbances in arbitrary steady flows," *J. Fluid Mech.*, vol. 226, pp. 383–400, 1991.

7. B. Lautrup, *Physics of Continuous Matter, Exotic and Every Day Phenomena in the Macroscopic World,* 2nd ed., CRC Press, 2011.

8. A.S. Dukhin and P.J. Goetz, "Bulk viscosity and compressibility measurements using acoustic spectroscopy," *J. Chem. Phys.*, vol. 130, p. 124519-1, 2009.

9. M.S. Cramer, "Numerical estimate for the bulk viscosity of ideal gases," *Phys. Fluids*, vol. 30, p. 066102, 2012.

10. F. Sharipov, "Data on the Velocity Slip and Temperature Jump on a Gas-Solid Interface," *J. Phys. Chem. Ref. Dat*a, vol. 40, no. 2, p. 023101, 2011.

11. W.R. Kampinga, Y.H. Wijnant and A. de Boer, "An Efficient Finite Element Model for Viscothermal Acoustics," *Acta Acustica united with Acustica*, vol. 97, p. 618-631, 2011.

12. W.R. Kampinga, *Viscothermal acoustics using finite elements – analysis tools for engineers*, Ph.D. thesis, University of Twente, 2010.

7

Ultrasound Interfaces

This chapter describes the physics interfaces found under the **Acoustics** > **Ultrasound** branch ( )

- The Convected Wave Equation, Time Explicit Interface
- The Nonlinear Pressure Acoustics, Time Explicit Interface
- Modeling with the Convected Wave Equation Interface
- Modeling with the Nonlinear Pressure Acoustics, Time Explicit Interface
- Theory for the Convected Wave Equation Interface
- Theory for the Nonlinear Pressure Acoustics, Time Explicit Interface
- References for the Ultrasound Interface

The Convected Wave Equation, Time Explicit Interface

The **Convected Wave Equation, Time Explicit (cwe)** interface ( ), found under the **Acoustics** > **Ultrasound** subbranch ( ) when adding a physics interface, is used to solve large transient linear acoustic problems containing many wavelengths in a stationary background flow. It is suited for time-dependent simulations with arbitrary time-dependent sources and fields. In general, the interface is suited for modeling the propagation of acoustic signals over large distances relative to the wavelength, for example, linear ultrasound problems. Application areas include ultrasound flowmeters and other ultrasound sensors where time of flight is an important parameter. The applications are not restricted to ultrasound. The interface includes absorbing layers that are used to set up effective nonreflecting-like boundary conditions. The interface exists in 2D, 2D axisymmetric, and 3D.

The interface is based on the discontinuous Galerkin (dG or dG-FEM) method and uses a time-explicit solver. The method is very memory efficient and can solve problems with many million degrees of freedom (DOFs). The method is also well suited for distributed computing on clusters.

In the no-flow case it can be advantageous to use The Pressure Acoustics, Time Explicit Interface instead. The equations are the same but the latter interface has more options to model, for example, scattering phenomena using a scattered field formulation or for handling sharp material discontinuities.

For modeling acoustic-structure interaction (ASI) or vibroacoustic problems the interface is fully multiphysics enabled and can be coupled to The Elastic Waves, Time Explicit Interface, using either the Convected Acoustic–Structure Boundary, Time Explicit or the Pair Convected Acoustic–Structure Boundary, Time Explicit multiphysics couplings.

The interface solves the linearized Euler equations assuming an adiabatic equation of state. The dependent variables are the acoustic pressure p and the acoustic velocity perturbation $\mathbf{u}$. The background mean flow can be any stationary flow with small to

moderate velocity gradients. General bulk dissipation (volumetric damping) can be added to model real fluids.

When solving models that are based on the dG method, optimizing the mesh is important. For further details see Optimizing the Mesh for DG in the Meshing, Discretization, and Solvers section.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Convected Wave Equation Model**, **Sound Hard Wall**, **Initial Values**, and **Axial Symmetry** (in case of 2D axisymmetric). Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Convected Wave Equation, Time Explicit** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `cwe`.

FILTER PARAMETERS FOR ABSORBING LAYERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. In the **Filter Parameters for Absorbing Layers** section you can change and control the values set for the filter used in the Absorbing Layers. The values of the filter parameters defined here are used in all absorbing layers added to the model and they override the value of filter parameters enabled in the material model (Convected Wave Equation Model). The default values of the filter parameters α, η_c, and s are set to 0.1, 0.01, and 2, respectively. Inside the absorbing layer it is important to use a filter that is not too aggressive since this will result in spurious reflections.

For general information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

EQUATION SETTINGS (2D AXISYMMETRIC ONLY)

This section is only available in **2D Axisymmetric**. Per default the acoustic velocity perturbation **u** is assumed to have only two components: the radial component u_r and the axial component u_z. You can select the **Include out-of-plane components** checkbox to enable the third — the circumferential component u_φ, which physically can only be nonzero if the background mean flow has the corresponding component.

DISCRETIZATION

In this section you can select the discretization for the **Acoustic pressure** and **Acoustic velocity**. Per default both are set to **Quartic** (4th order). Using quartic elements together with a mesh size equal to approximately half the wavelength to be resolved, leads to the best performance when using the dG method. For further details see the Meshing, Discretization, and Solvers section.

When coupling a *Fluid Flow* interface to the **Convected Wave Equation, Time Explicit** interface it is important to have consistent settings for the Geometry Shape Function and the discretization of the physics. The **Automatic** setting for the **Geometry shape function** (in the Curved Mesh Elements section on the Components node's settings) results in a linear geometry representation to fit the *Fluid Flow* interface. This can lead to numerical errors when solving the **Convected Wave Equation, Time Explicit** physics as the default is to use fourth-order (**Quartic**) spatial discretization of the dependent variables. Errors typically occur when curved boundaries are present in the model or when using multiphysics pair couplings. To remedy this change the **Geometry shape function** to **Quadratic Lagrange**.

If the settings in a model are inconsistent, a warning is given in the solver with the text:

```
The geometry shape function order is not consistent with
the discretization of the dependent variables.
Consider increasing the geometry shape function order to
Quadratic Lagrange in the Curved Mesh Element section in
the Component node's settings.
```

DEPENDENT VARIABLES

The dependent variables (field variables) are the **Acoustic pressure**, **Acoustic velocity**, and **Acoustic velocity components**. The names can be changed, but the names of fields and dependent variables must be unique within a model.

- For information about modeling strategies, meshing, the absorbing layer and more, see Modeling with the Convected Wave Equation Interface.
- For the theoretical background of the model solved, see Theory for the Convected Wave Equation Interface.

- *Gaussian Pulse in 2D Uniform Flow: Convected Wave Equation and Absorbing Layers*. Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/gaussian_pulse_absorbing_layers**
- *Ultrasound Flowmeter with Generic Time-of-Flight Configuration*. Application Library path **Acoustics_Module/Ultrasound/ultrasound_flow_meter_generic**

Domain, Boundary, Edge, Point, and Pair Nodes for the Convected Wave Equation Interface

The Convected Wave Equation, Time Explicit Interface has these domain, boundary, edge, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users). Continuity in pressure and velocity is the default condition on interior boundaries.

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Convected Wave Equation Model
- Domain Sources
- General Flux/Source
- Initial Values
- Interior Normal Velocity
- Interior Wall
- Normal Velocity
- Pressure
- Sound Hard Wall
- Specific Acoustic Impedance (Isentropic)
- Symmetry

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Convected Wave Equation Model

The **Convected Wave Equations** node adds the equations for modeling the propagation of acoustic waves in a stationary background flow. Adiabatic behavior is assumed, meaning that this in some sense represents pressure acoustics in the presence of flow. The convected wave equation model solves the linearized Euler equations also referred to as linear acoustic equations for moving media. The equations are valid for any stationary background mean flow as long as there are not too large gradients in the background properties.

The linear continuity equation, momentum equation, and equation of state solved are given by:

$$\frac{\partial \rho}{\partial t} + (\mathbf{u}_0 \cdot \nabla)\rho + (\mathbf{u} \cdot \nabla)\rho_0 + \rho(\nabla \cdot \mathbf{u}_0) + \rho_0(\nabla \cdot \mathbf{u}) = f_\mathrm{p}$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}_0 \cdot \nabla)\mathbf{u} + (\mathbf{u} \cdot \nabla)\mathbf{u}_0 + \frac{1}{\rho_0}\nabla p - \frac{\rho}{\rho_0^2}\nabla p_0 = \mathbf{f}_\mathrm{v}$$

$$\rho = \frac{p}{c_0^2}$$

where $\mathbf{u}_0$ is the background mean flow velocity, p_0 is the background mean flow pressure, ρ_0 is the background mean flow density, and c_0 is the speed of sound. All background properties can vary in space (the variation has to be smooth), but cannot be discontinuous. The right-hand side sources f_p and $\mathbf{f}_\mathrm{v}$ can be defined by the Domain

Sources. More information about the governing equations is given in the Theory for the Convected Wave Equation Interface section.

In the **Settings** window, define the properties for the acoustics model and model inputs including the background mean flow, pressure, and velocity.

MODEL INPUTS

In order to model the influence the background mean flow has on the propagation of the acoustic waves in the fluid, the **Background mean flow pressure** p_0 and **Background mean flow velocity** $\mathbf{u}_0$ need to be defined. If a material that is temperature dependent is used the **Background mean flow temperature** T_0 field is also present.

All the background mean flow parameters can be functions of space. They can be either analytical expressions (user defined) or they can be picked up from a fluid flow simulation performed using the CFD Module. In this case use the Background Fluid Flow Coupling multiphysics feature to ensure correct mapping of the flow data. By default they are set to the quiescent constant background conditions of air.

In transient models, when space dependent material data is given by an interpolation function, significant performance gains can be achieved with the following setup of the **Interpolation** function:

- Select **Use spatial coordinates as arguments**
- Set the **Frame** to **Mesh**
- Select **Reinterpolate interpolation data on computational mesh**

No arguments should be used when calling the interpolation function, for example, use `int` and not `int(x,y,z)`.

See the Interpolation section in the *COMSOL Multiphysics Reference Manual*.

FLUID PROPERTIES

Select a **Fluid model** either as **Linear elastic** (the default) or **General dissipation**. For both options define the background mean flow density ρ_0 (SI unit: kg/m^3) and the background mean flow speed of sound c_0 (SI unit: m/s). The default is to use the property values from the material (**From material**). Select **User defined** from the list to enter a user-defined value in the text field that appears. For the **General dissipation** option also define the **Sound diffusivity** δ (SI unit: m^2/s). For numerical stability reasons it is recommended to use physical values of attenuation properties.

The density can also be picked up from a fluid flow simulation when the Background Fluid Flow Coupling multiphysics feature is used.

The relation between the sound diffusivity δ and the equivalent (plane wave) attenuation coefficient α, at a given frequency f, is given by the expression $\alpha = \delta\omega^2/(2c^3)$, where $\omega = 2\pi f$ and c is the speed of sound.

LAX–FRIEDRICHS FLUX PARAMETER

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog. In this section, you specify the value of the *Lax–Friedrichs flux parameter* τ_{LF} (default value: 0.2). This value controls the numerical flux between the elements (nodal discontinuous Lagrange elements) used with the discontinuous Galerkin (dG) method. The numerical flux defines how adjacent elements are connected and how continuous p and $\mathbf{u}$ are. Different definitions of the numerical flux lead to different variants of the dG method. The flux implemented here is the so-called global Lax–Friedrichs numerical flux. The value of the parameter τ_{LF} should be between 0 and 0.5. For $\tau_{LF} = 0$ a so-called central flux is obtained. Setting $\tau_{LF} = 0.5$ gives a maximally dissipative global Lax–Friedrichs flux.

For general information about the numerical flux see the Numerical Flux section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

FILTER PARAMETERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. By default, the filter parameters α, η_c, and s are not active. Select the **Activate** checkbox to activate the filter. The filter provides higher-order smoothing for the dG formulation and can be used to stabilize the solution, for example, when a large background flow is present or large gradients are present. Inside absorbing layers the settings given here are overridden by the Filter Parameters for Absorbing Layers.

Enter values for the filter parameters in the corresponding text fields (default values: 36, 0.6, and 3). α must be positive and lie between 0 and 36. $\alpha = 0$ means no dissipation and $\alpha = 36$ means maximal dissipation. η_c should be between 0 and 1, where 0 means maximal filtering and 1 means no filtering (even if filtering is active). The s parameter should be larger than 0 and controls the order of the filtering (a

dissipation operator of order $2s$). For $s = 1$, you get a filter that is related to the classical 2nd-order Laplacian. A larger s gives a more pronounced low-pass filter.

For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

Domain Sources

Use a **Domain Sources** node to add various sources to a domain. The feature adds right-hand side sources to either the continuity or the momentum equation.

The **Domain mass source** q_{source} and the **Domain pressure source** p_{source} both contribute to the source term f_{p} of the continuity equation. This is a monopole like source that acts uniformly in all directions. The two represent different formulations of the same source type.

$$f_{\text{p}} = q_{\text{source}} + \frac{1}{c_0^2}\frac{\partial p_{\text{source}}}{\partial t}$$

The **Domain velocity source** $\mathbf{u}_{\text{source}}$ and **Domain force source** $\mathbf{f}_{\text{source}}$ both contribute to the source term $\mathbf{f}_{\text{v}}$ of the momentum equation. This is a dipole like source that acts in the direction of its vector. The two represent different formulations of the same source type.

$$\mathbf{f}_{\text{v}} = \rho_0\frac{\partial \mathbf{u}_{\text{source}}}{\partial t} + \mathbf{f}_{\text{source}}$$

All source terms can be any analytical or interpolation function of the time. The variable `t` is reserved to represent time in expressions.

DOMAIN SOURCES

In this section, add one or more of the following sources:

- Add a **Domain mass source** q_{source} (SI unit: $\text{kg}/(\text{m}^3\cdot\text{s})$)
- Add a **Domain pressure source** p_{source} (SI unit: Pa)
- Add a **Domain velocity source** $\mathbf{u}_{\text{source}}$ (SI unit: m/s).
- Add a **Domain force source** $\mathbf{f}_{\text{source}}$ (SI unit: N/m^3).

Sound Hard Wall

The **Sound Hard Wall** adds a boundary condition for a sound hard boundary or wall, which is a boundary at which the normal component of the velocity is zero (a slip condition). This is assumed true for both the acoustic velocity and the background velocity

$$-\mathbf{n} \cdot \mathbf{u} = 0 \qquad -\mathbf{n} \cdot \mathbf{u}_0 = 0$$

Note that this condition is not identical to the Symmetry condition where only the normal component of the acoustic velocity is assumed to be zero.

Initial Values

The **Initial Values** node adds initial values for the acoustic pressure and the acoustic velocity that can serve as an initial state for the solver. If more than one initial value is needed, from the **Physics** toolbar click to add more **Initial Values** nodes.

INITIAL VALUES

Enter a value or expression for the initial values for the **Pressure** p (SI unit: Pa) and the **Acoustic velocity**, $\mathbf{u}$ (SI unit: m/s).

Pressure

The **Pressure** node creates a boundary condition that acts as a pressure source at the boundary. The acoustic pressure is given at the boundary $p = p_b(t)$.

PRESSURE

Enter an expression for the **Boundary pressure** p_b (SI unit: Pa).

Symmetry

The **Symmetry** node adds a boundary condition imposing symmetry in the pressure and velocity field. Use this condition to reduce the size of a model by cutting it in half where there are known symmetries. The condition implies symmetry in the acoustic fields

$$-\mathbf{n} \cdot \mathbf{u} = 0$$

Normal Velocity

The **Normal Velocity** adds an inward normal velocity either given as a scalar v_n or as a velocity vector $\mathbf{v}_b$, the condition given is

$$-\mathbf{n} \cdot \mathbf{u} = v_n(t) = -\mathbf{n} \cdot \mathbf{v}_b(t)$$

where $\mathbf{n}$ is the surface normal. Both expressions should be functions of time. This feature represents an external source term. It can also be used to model a vibrating transducer surface or other vibrating surfaces.

NORMAL VELOCITY

Select a **Type** — **Inward Velocity** (the default) or **Velocity**.

- For **Inward Velocity** enter an expression for the **Inward velocity** $v_n(t)$ (SI unit: m/s). Use a positive value for inward velocity or a negative value for outward velocity.
- For **Velocity** enter expressions for the components of the **Velocity** $\mathbf{v}_b(t)$ (SI unit: m/s).

Specific Acoustic Impedance (Isentropic)

The **Specific Acoustic Impedance (Isentropic)** node adds a boundary condition defining a relation between the local acoustic pressure p and the normal acoustic velocity $\mathbf{n} \cdot \mathbf{u}$, that is, the acoustic impedance Z. The condition specifies the inward normal velocity

$$-\mathbf{n} \cdot \mathbf{u} = -\frac{p}{Z}$$

This condition can be used to model the properties of artificial boundaries. When the impedance is set equal to the characteristic specific impedance of a propagating plane wave $\rho_0 c_0$, the condition represent the simplest nonreflecting boundary condition. This is also the default value of the impedance when the impedance condition is added.

Note that this condition is not equivalent to the general impedance condition, as given by Myers (see Ref. 4), when a flow is present. This means that, for example, it cannot be used at a reacting surface which has a tangential flow. For this kind of detailed conditions a frequency domain representation is needed.

The specific acoustic impedance condition (with the default value $\rho_0 c_0$) should be used at the outer boundary when Absorbing Layers are used in a model.

IMPEDANCE

Enter a value or expression for the **Specific impedance** Z_n (SI unit: Pa·s/m). The default expression is `cwe.rho0*cwe.c0` which is the characteristic specific impedance $\rho_0 c_0$ of a propagating plane wave.

Interior Wall

The **Interior Wall** node adds the boundary condition for a sound hard boundary or wall on interior boundaries. Add this node from the **Interior Conditions** submenu. The condition reads

$$\mathbf{u}_{\text{up}} \cdot \mathbf{n} = \mathbf{u}_{\text{down}} \cdot \mathbf{n} = 0$$
$$\mathbf{u}_{0,\,\text{up}} \cdot \mathbf{n} = \mathbf{u}_{0,\,\text{down}} \cdot \mathbf{n} = 0$$

The condition ensures that the normal velocity is zero on both the up and down side of the boundary, while applying a slit on the pressure (the pressure is discontinuous across the boundary). The condition also assumes that the background flow has a similar condition applied at the boundary.

Interior Normal Velocity

The **Interior Normal Velocity** node adds the boundary condition for a moving interior boundary. Add this node from the **Interior Conditions** submenu. This can, for example, represent a thin vibrating structure or s speaker membrane. The condition reads

$$\mathbf{u}_{\text{up}} \cdot \mathbf{n} = \mathbf{v}_b(t) \cdot \mathbf{n} = v_n(t)$$
$$\mathbf{u}_{\text{down}} \cdot \mathbf{n} = \mathbf{v}_b(t) \cdot \mathbf{n} = v_n(t)$$

The condition ensures that the normal velocity is defined on both the up and down side of the boundary, while applying a slit on the pressure (the pressure is discontinuous across the boundary).

NORMAL VELOCITY

Select the **Type**: **Velocity** (default) or **Normal velocity** and specify either the **Velocity** (vector) $\mathbf{v}_b(t)$ or the normal component through the **Normal velocity** $v_n(t)$.

General Flux/Source

The **General Flux/Source** node, found under the **More** submenu, adds the most general boundary condition for the convected wave equation, as it is formulated (in the

conservative form) for the discontinuous Galerkin method (dG). The condition defines the normal flux $\mathbf{g}$ at an exterior boundary, given by

$$-\mathbf{n}\cdot\Gamma = \begin{bmatrix} \rho_0(\mathbf{n}\cdot\mathbf{u}) + \frac{p}{c_0^2}(\mathbf{n}\cdot\mathbf{u}_0) \\ \frac{p}{c_0^2}u_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(u(\mathbf{n}\cdot\mathbf{u}_0) + u_0(\mathbf{n}\cdot\mathbf{u})) + n_1 p \\ \frac{p}{c_0^2}v_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(v(\mathbf{n}\cdot\mathbf{u}_0) + v_0(\mathbf{n}\cdot\mathbf{u})) + n_2 p \\ \frac{p}{c_0^2}w_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(w(\mathbf{n}\cdot\mathbf{u}_0) + w_0(\mathbf{n}\cdot\mathbf{u})) + n_3 p \end{bmatrix} = \mathbf{g}$$

where $\mathbf{u} = (u, v, w)$, $\mathbf{u}_0 = (u_0, v_0, w_0)$, and $\mathbf{n} = (n_x, n_y, n_z)$ ($\mathbf{n} = (n_r, n_\varphi, n_z)$ in 2D axisymmetric) is the surface normal. Care should be taken when using this advanced condition as the method is sensitive to proper formulation of boundary conditions. Only one characteristic can enter the domain at any boundary at the time, meaning that it is easy to over-specify the problem. Use the mesh normals (`nxmesh`, `nymesh`, and `nzmeshor`; or `nrmesh`, `nphimesh`, and `nzmesh`) in the expression you define.

GENERAL FLUX/SOURCE

Enter the expression for the components of the **Flux vector g**.

General Interior Flux

The **General Interior Flux** node, found under the **More** submenu, adds a the most general interior boundary condition for the convected wave equation, as it is formulated (in the conservative form) for the discontinuous Galerkin method (dG). The condition defines the normal flux $\mathbf{g}$ on an interior boundary by

$$-\mathbf{n}\cdot\Gamma^* = \mathbf{g}$$

here Γ^* represents the total numerical flux. This means that if this condition is added on an interior boundary it overrides the existing Lax–Friedrichs numerical flux. So care should be taken when specifying this condition as it needs to include the necessary numerical flux contribution to ensure stability of the method.

On an interior boundary you can use the `up()` and `down()` operators to access values from both sides of the boundary. If a dependent variable is used in this expression (without `up()` or `down()` operators) an implicit `mean()` operation is invoked taking

the average of the up and down side values. Use the mesh normals (`nxmesh`, `nymesh`, and `nzmesh`; or `nrmesh`, `nphimesh`, and `nzmesh`) in the expression you define.

GENERAL INTERIOR FLUX

Enter the expression for the components of the **Flux vector g**.

The Nonlinear Pressure Acoustics, Time Explicit Interface

The **Nonlinear Pressure Acoustics, Time Explicit (nate)** interface ( ), found under the **Acoustics** > **Ultrasound** subbranch ( ) when adding a physics interface, is used to model the propagation of nonlinear finite amplitude acoustic waves in computational domains that contain many wavelengths. It is suited for time-dependent simulations with arbitrary time-dependent sources and fields. Absorbing layers are used to set up effective nonreflecting-like boundary conditions. Application areas include biomedical applications, for instance, ultrasonic imaging and high-intensity focused ultrasound (HIFU). The applications are not restricted to ultrasound. The interface exists in 2D, 2D axisymmetric, and 3D.

The interface is based on the discontinuous Galerkin (dG or dG-FEM) method and uses a time explicit solver. The method is very memory efficient and can solve problems with many million degrees of freedom (DOFs). The method is also well suited for distributed computing on clusters.

In the linear case it can be advantageous to use The Pressure Acoustics, Time Explicit Interface instead, for example, when modeling scattering phenomena using a scattered field formulation and a background acoustic field. This split is not possible for nonlinear problems. In the presence of a stationary background flow and linear propagation, The Convected Wave Equation, Time Explicit Interface should be used, for example, when modeling ultrasonic flow meters.

For modeling acoustic-structure interaction (ASI) or vibroacoustic problems the interface is fully multiphysics enabled and can be coupled to The Elastic Waves, Time Explicit Interface, using either the Acoustic–Structure Boundary, Time Explicit or the Pair Acoustic–Structure Boundary, Time Explicit multiphysics couplings.

The interface solves the second-order nonlinear governing equations for the acoustic pressure p and the acoustic velocity perturbations $\mathbf{u}$. The interface is suited for modeling progressive wave propagation phenomena when the cumulative nonlinear effects surpass the local nonlinear effects. Thus the model is consistent with the second-order Westervelt equation for the acoustic pressure (see also the Nonlinear Acoustics (Westervelt) Contributions feature available with The Pressure Acoustics, Transient Interface). General bulk dissipation (volumetric damping) can be added to model real fluids.

Several features are available to help solve the nonlinear and highly nonlinear problems including the use of a numerical Limiter (to capture shocks) but also the use of Adaptive Mesh Refinement.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `nate`.

FILTER PARAMETERS FOR ABSORBING LAYERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. In the **Filter Parameters for Absorbing Layers** section you can change and control the values set for the filter used in the Absorbing Layers. The values of the filter parameters defined here are used in all absorbing layers added to the model and they override the value of filter parameters enabled in the material model (Nonlinear Pressure Acoustics, Time Explicit Model). The default values of the filter parameters α, η_c, and s are set to 0.1, 0.01, and 2, respectively. Inside the absorbing layer it is important to use a filter that is not too aggressive since this will result in spurious reflections.

For general information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

NUMERICAL FLUX

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog. Only one option exists for the **Numerical flux** formulation used in the numerical scheme when solving the dG problem. For **Lax–Friedrichs** (the only option) enter the **Lax–Friedrichs flux parameter** τ_{LF} (default is 0.2). The value of the parameter τ_{LF} should be between 0 and 0.5. A parameter value of 0 represents a central flux, which is the least dissipative but also the least stable numerical flux. A parameter value of 0.5 gives a maximally dissipative global Lax–Friedrichs flux (see The Lax–Friedrichs Flux).

LIMITER

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog. The limiter is used to control and stabilize highly nonlinear problems with shock formation. The limiter can only be used with linear discretization. For details see the Solving Highly Nonlinear Problems section in Modeling with the Nonlinear Pressure Acoustics, Time Explicit Interface.

- For an example of a highly nonlinear problem that uses the WENO limiter see the *Nonlinear Propagation of a Cylindrical Wave — Verification Model* tutorial model. Application Library path **Acoustics_Module/Nonlinear_Acoustics/ nonlinear_cylindrical_wave**
- For an example of a nonlinear problem with pulse propagation that uses mesh adaptation see the *High-Intensity Focused Ultrasound (HIFU) Propagation Through a Tissue Phantom* tutorial model. Application Library path **Acoustics_Module/ Nonlinear_Acoustics/ hifu_tissue_sample**

TRANSIENT MESH SETTINGS

Enter a value for the **Maximum frequency to resolve** $f_{\max}$ (the default is `1000[Hz]`). This value is used to set up the physics-controlled mesh for the Nonlinear Pressure Acoustics, Time Explicit interface. Remember that this value should be set in order to resolve harmonics generated due to nonlinear effects.

DISCRETIZATION

In this section you can select the **Element order** for the acoustic pressure and acoustic velocity (the same is used for both). Per default the order is set to **Quartic** (4th order). Using quartic elements together with a mesh size equal to approximately one and a half of the wavelength, leads to the best performance when using the dG method. For

highly nonlinear problems it can be necessary to use a Limiter in which case it is necessary to change the **Element order** to **Linear**.

When solving **Nonlinear Pressure Acoustics, Time Explicit**, it is important to have consistent settings for the Geometry Shape Function and the Discretization of the physics. The **Automatic** setting for the **Geometry shape function** (in the Curved Mesh Elements section on the Components node's settings) may results in a linear geometry representation, if other physics are present in the model. This can lead to numerical errors when solving as the default is to use fourth-order spatial discretization of the dependent variables. To remedy this change the **Geometry shape function** to **Quadratic Lagrange**.

If the settings in a model are inconsistent, a warning is given in the solver with the text:

```
The geometry shape function order is not consistent with
the discretization of the dependent variables.
Consider increasing the geometry shape function order to
Quadratic Lagrange in the Curved Mesh Element section in
the Component node's settings.
```

DEPENDENT VARIABLES

The dependent variables are the **Acoustic pressure** and the **Acoustic velocity**. The names can be changed, but the names of fields and dependent variables must be unique within a model. The name for the **Acoustic velocity components** can also be selected individually.

- For information specific to modeling nonlinear problems using the time explicit interface, see Modeling with the Nonlinear Pressure Acoustics, Time Explicit Interface.
- For information about modeling strategies, meshing, the absorbing layer and more, see the relevant information in the Modeling with the Convected Wave Equation Interface.

Domain, Boundary, Edge, and Point Nodes for the Nonlinear Pressure Acoustics, Time Explicit Interface

The Nonlinear Pressure Acoustics, Time Explicit Interface has the following domain and boundary conditions listed in alphabetic order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users). Continuity in velocity and pressure is the default condition on interior boundaries. If a boundary separates two different materials the Material Discontinuity or Continuity condition (the pair version of material discontinuity condition) has to be applied to ensure the continuity in velocity and pressure.

- Continuity
- General Flux/Source
- General Interior Flux
- Heat Source
- Impedance
- Initial Values
- Interior Normal Velocity
- Interior Sound Hard Boundary (Wall)
- Mass Source
- Material Discontinuity
- Nonlinear Pressure Acoustics, Time Explicit Model
- Normal Velocity
- Pressure
- Sound Hard Boundary (Wall)
- Sound Soft Boundary
- Symmetry
- Volume Force Source

Nonlinear Pressure Acoustics, Time Explicit Model

The **Nonlinear Pressure Acoustics, Time Explicit Model** node adds the equations for modeling the propagation of nonlinear acoustic waves. The nonlinear continuity equation and momentum equation solved are given by:

$$\begin{aligned} \frac{1}{\rho c^2}\frac{\partial p_t}{\partial t} + \nabla \cdot \left[\left(1 + \frac{\beta p_t}{\rho c^2}\right)\mathbf{u}_t\right] &= Q_m \\ \rho\frac{\partial \mathbf{u}_t}{\partial t} + \nabla \cdot (p_t\mathbf{I}) &= \mathbf{q}_d \\ p_t = p \qquad \mathbf{u}_t &= \mathbf{u} \end{aligned} \tag{7-1}$$

where p_t is the total acoustic pressure and $\mathbf{u}_t$ is the total acoustic velocity fluctuations. Because the problem solved is nonlinear the superposition principle does not apply which means that the "total" fields are always equal to the dependent variables.

In the **Settings** window, define the properties for the acoustics model, model inputs, and material properties.

MODEL INPUTS

Depending on the material selected enter values for the Model Inputs, typically the **Temperature** T, the **Pressure** p_A, and possibly the **Relative Humidity** ϕ.

PRESSURE ACOUSTICS MODEL

Select the **Fluid model** — **Linear elastic**, **Viscous**, **Thermally conducting**, **Thermally conducting and viscous**, **General dissipation**, or **Ideal gas**. The settings are the same as for the Pressure Acoustics, Time Explicit Model.

In transient models, when space dependent material data is given by an interpolation function, significant performance gains can be achieved with the following setup of the **Interpolation** function:

- Select **Use spatial coordinates as arguments**
- Set the **Frame** to **Mesh**
- Select **Reinterpolate interpolation data on computational mesh**

No arguments should be used when calling the interpolation function, for example, use `int` and not `int(x,y,z)`.

See the Interpolation section in the *COMSOL Multiphysics Reference Manual*.

Note that for the time explicit formulation, the material data has to change smoothly. For abrupt changes the Material Discontinuity condition has to be used on an interior boundary.

COEFFICIENT OF NONLINEARITY

Select how to define the **Coefficient of nonlinearity** β — **From parameter of nonlinearity** (default) or **User defined**. The **Parameter of nonlinearity** B/A can be taken **From material** or entered as a **User defined** expression. For the **Ideal gas** fluid model, it is also possible to compute the **Coefficient of nonlinearity** β **From ratio of specific heats**.

FILTER PARAMETERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. By default, the filter parameters α, η_c, and s are not active. Select the **Activate** checkbox to activate the filter. The filter provides higher-order smoothing for the DG formulation. Inside absorbing layers the settings given here are overridden by the Filter Parameters for Absorbing Layers.

For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

Compute Minimum and Maximum Pressure

The **Compute Minimum and Maximum Pressure** subfeature can be added to the Nonlinear Pressure Acoustics, Time Explicit Model as either a domain or a boundary feature. The minimum and maximum value of the pressure, over time and space, will be computed on the selected entity. It will create two variables `nate.p_min` and `nate.p_max`, these can be used in results to, for example, evaluate the size of a focal zone.

For an example, see the *High-Intensity Focused Ultrasound (HIFU) Propagation Through a Tissue Phantom* tutorial model. Application Library path **Acoustics_Module/ Nonlinear_Acoustics/hifu_tissue_sample**

Initial Values

The **Initial Values** node adds initial values for the acoustic pressure and the acoustic velocity that can serve as an initial state for the solver. If more than one initial value is needed, from the **Physics** toolbar click to add more **Initial Values** nodes.

INITIAL VALUES

Enter a value or expression for the initial values for the **Pressure** p (SI unit: Pa) and the **Acoustic velocity**, $\mathbf{u}$ (SI unit: m/s).

Mass Source

The **Mass Source** sets up the right-hand side of the continuity equation, in Equation 7-1, in terms of a mass rate of change. This is a monopole-like source that acts uniformly in all directions.

$$Q_m = \frac{1}{\rho} M$$

MASS SOURCE

Enter the **Mass source** M (SI unit: kg/(m^3·s)).

Heat Source

The **Heat Source** sets up the right-hand side of the continuity equation, in Equation 7-1, in terms of a heat density source. This is a monopole-like source that acts uniformly in all directions:

$$Q_m = \frac{\alpha_p}{\rho C_p} Q_{heat}$$

where α_p is the (isobaric) coefficient of thermal expansion, ρ is the density, C_p is the heat capacity at constant pressure, and Q_{heat} is the heat source.

MODEL INPUT

Enter the ambient **Temperature** T, the default is `293.15[K]`.

HEAT SOURCE

Enter the **Heat Source** Q_{heat} (SI unit: W/m^3).

FLUID PROPERTIES

Enter the **Heat capacity at constant pressure** C_p and the **Ratio of specific heats** γ either selected **From material** or **User defined**. The ratio of specific heats is used to calculate the coefficient of thermal expansion through the thermodynamic relation

$$\alpha_p = \frac{1}{c}\sqrt{\frac{C_p(\gamma - 1)}{T}}$$

Volume Force Source

The **Volume Force Source** sets up the right-hand side of the momentum equation, in Equation 7-1, in terms of a volumetric force. This is a dipole-like source that acts differently in two directions.

$$\mathbf{q}_d = \mathbf{F}$$

VOLUME FORCE SOURCE

Enter the **Volume force source** $\mathbf{F}$ (SI unit: N/m^3).

Sound Hard Boundary (Wall)

The **Sound Hard Boundary (Wall)** adds a boundary condition for a sound hard boundary or wall, which is a boundary at which the normal component of the velocity is zero (a slip condition)

$$\mathbf{n} \cdot \mathbf{u}_t = 0$$

The condition is also available from the **Pairs** submenu, as an option at interfaces between parts in an assembly. This effectively introduces a condition similar to the Interior Sound Hard Boundary (Wall) but used on an assembly.

Sound Soft Boundary

The **Sound Soft Boundary** conditions adds a boundary condition for a sound soft boundary, where the acoustic pressure vanishes $p_t = 0$. It is an appropriate approximation, for example, for a liquid-gas interface or in some cases for external waveguide ports.

Pressure

The **Pressure** node creates a boundary condition that acts as a pressure source at the boundary. The total acoustic pressure is given at the boundary $p_t = p_0(t)$.

PRESSURE

Enter an expression for the **Pressure** $p_0(t)$ (SI unit: Pa). The input can be any analytical function or interpolation data that can be function of both time and space.

Symmetry

The **Symmetry** node adds a boundary condition imposing symmetry in the pressure and velocity field. Use this condition to reduce the size of a model by cutting it in half where there are known symmetries. The condition implies symmetry in the acoustic fields

$$\mathbf{n} \cdot \mathbf{u}_t = 0$$

Normal Velocity

The **Normal Velocity** condition adds a source specified in terms of an inward normal velocity $v_n(t)$ or velocity $\mathbf{v}_0(t)$ of the boundary. The component in the normal direction is used to define the boundary condition. This feature represents an external source term like a vibrating surface. The condition is also available from the **Pairs** submenu, as an option at interfaces between parts in an assembly. This effectively introduces a condition similar to the Interior Normal Velocity but used on an assembly.

NORMAL VELOCITY

Select a **Type**: **Inward Velocity** (the default) or **Velocity**.

- For **Inward Velocity** enter the value of the **Inward velocity** v_n (SI unit: m/s).
- For **Velocity** enter values for the components of the **Velocity** $\mathbf{v}_0$ (SI unit: m/s).

Impedance

The **Impedance** condition adds a boundary condition defining the relation between the local acoustic pressure p and the normal acoustic velocity $\mathbf{n}\cdot\mathbf{u}_t$, that is, the specific acoustic impedance Z_i. The model includes the option to define the nonlinear or second order pressure-velocity relation to extend the validity of the impedance condition to higher pressure amplitudes.

The second order impedance expression specifies how the normal velocity relates to the pressure through the impedance, including a correction factor, according to

$$\mathbf{n} \cdot \mathbf{u}_t = \frac{p_t}{Z_i}\left(1 - \frac{\beta p_t}{2\rho c^2}\right)$$

This condition can be used to model the properties of artificial boundaries by adding resistive losses (no reactive components can be added). When the impedance is set

equal to the characteristic specific impedance of a propagating plane wave ρc, the condition represent the simplest nonreflecting boundary condition. This is also the default value of the impedance when the condition is added.

The acoustic impedance condition (with the default value ρc) should be used at the outer boundary when *Absorbing Layers* are used in a model.

IMPEDANCE

Select the **Pressure-particle velocity relation** used for the impedance formulation **First order** (default) or **Second order**. Enter a value or expression for the **Impedance** Z_i (SI unit: Pa·s/m). The default expression is `nate.c*nate.rho` which is the characteristic specific impedance ρc of a propagating plane wave (linear theory).

Interior Sound Hard Boundary (Wall)

The **Interior Sound Hard Boundary (Wall)** node, found under the **Interior Conditions** submenu, adds the boundary condition for a sound hard boundary or wall on interior boundaries. The condition reads

$$\mathbf{u}_{\text{t, up}} \cdot \mathbf{n} = 0$$
$$\mathbf{u}_{\text{t, down}} \cdot \mathbf{n} = 0$$

The condition ensures that the total normal velocity is zero on both the up and down side of the boundary, while applying a slit on the pressure (the pressure is discontinuous across the boundary).

Interior Normal Velocity

The **Interior Normal Velocity** node, found under the **Interior Conditions** submenu, adds the boundary condition for a moving interior boundary. This can, for example, represent a thin vibrating structure or s speaker membrane. The condition reads

$$\mathbf{u}_{\text{t, up}} \cdot \mathbf{n} = \mathbf{v}_0(t) \cdot \mathbf{n} = -v_{\text{n}}(t)$$
$$\mathbf{u}_{\text{t, down}} \cdot \mathbf{n} = \mathbf{v}_0(t) \cdot \mathbf{n} = -v_{\text{n}}(t)$$

The condition ensures that the total normal velocity is defined on both the up and down side of the boundary, while applying a slit on the pressure (the pressure is discontinuous across the boundary).

NORMAL VELOCITY

Select the **Type**: **Velocity** (default) or **Normal velocity** and specify either the **Velocity** (vector) $\mathbf{v}_0$ or the normal component through the **Normal velocity** v_n.

Material Discontinuity

The **Material Discontinuity** node, found under the **Interior Conditions** submenu, adds an interior boundary condition to handle jumps in material properties. As the interface is based on the discontinuous Galerkin method special handling is necessary when a jump exists in the specific characteristic impedance $Z_0 = \rho c$ of the medium.

The condition implements continuity in total pressure and in the normal velocity over the interior boundary:

$$p_{\text{up}} = p_{\text{down}}$$
$$\mathbf{n} \cdot \mathbf{u}_{\text{up}} = \mathbf{n} \cdot \mathbf{u}_{\text{down}}$$

Subscripts “up” and “down” in the equation refer to the up and down sides of the interior boundary, relative to the normal $\mathbf{n}$.

Continuity

The **Continuity** node, exists as an option at interfaces between parts in an assembly, from the **Pairs** submenu. The condition is similar to the Material Discontinuity condition that applies to interior boundaries. The difference is that the **Continuity** condition should also be applied when the same material exists on both sides of a pair. The advantage of using a pair feature is that the mesh does not need to be conforming on the two sides of the interface (the two parts pf the assembly). This is especially advantageous for the time explicit discontinuous Galerkin method as the time step depends on mesh size and local speed of sound.

The condition implements continuity in total pressure and in the normal velocity over the pair:

$$p_{\text{t, src}} = p_{\text{t, dst}}$$
$$\mathbf{n}_{\text{src}} \cdot \mathbf{u}_{\text{t, src}} = -\mathbf{n}_{\text{dst}} \cdot \mathbf{u}_{\text{t, dst}}$$

Subscripts “src” and “dst” in the equation refer to the source and destination in the pair; that is, the two sides in the pair.

General Flux/Source

The **General Flux/Source** node, found under the **More** submenu, adds the most general boundary condition for the Nonlinear Pressure Acoustics, Time Explicit model, as it is formulated (in the conservative form) for the discontinuous Galerkin method (dG). The condition defines the normal pressure and velocity fluxes g_p and $\mathbf{g}_u$ at an exterior boundary, given by

$$-\mathbf{n}\cdot\Gamma_p = -\left(1+\frac{\beta p}{\rho c^2}\right)\mathbf{n}\cdot\mathbf{u} = g_p$$
$$-\mathbf{n}\cdot\Gamma_u = -\mathbf{n}p = \mathbf{g}_u$$

where $\mathbf{u} = (u, v, w)$ and $\mathbf{n} = (n_x, n_y, n_z)$ in 3D and 2D, or $\mathbf{n} = (n_r, n_\varphi, n_z)$ in 2D axisymmetric. Note that this condition is expressed in terms of the dependent variables and not the total fields. Care should be taken when using this advanced condition as the method is sensitive to proper formulation of boundary conditions. Only one characteristic can enter the domain at any boundary at the time, meaning that it is easy to overspecify the problem. Use the mesh normals (`nxmesh`, `nymesh`, and `nzmesh`; or `nrmesh`, `nphimesh`, and `nzmesh`) in the expression you define.

GENERAL FLUX/SOURCE

Enter the expression for the components of the **Flux vector** g_p and $\mathbf{g}_u$.

General Interior Flux

The **General Interior Flux** node, found under the **More** submenu, adds the most general interior boundary condition for the Nonlinear Pressure Acoustics, Time Explicit model, as it is formulated (in the conservative form) for the discontinuous Galerkin method (dG). The condition defines the normal flux $\mathbf{g} = [g_p\ \mathbf{g}_u]$ at an interior boundary. It should include any necessary numerical flux (as it overrides the Numerical Flux). Care should be taken when using this advanced condition as the method is sensitive to proper formulation of boundary conditions.

GENERAL INTERIOR FLUX

Enter the expression for the components of the **Flux vector** g_p and $\mathbf{g}_u$.

Modeling with the Convected Wave Equation Interface

This section contains information about:

- Meshing, Discretization, and Solvers
- Results: Variables and Quality
- Absorbing Layers
- Stabilizing Physical Instabilities (Filtering)
- Storing Solution on Selections for Large Models
- Assemblies and Pair Conditions

Meshing, Discretization, and Solvers

The Convected Wave Equation, Time Explicit Interface (CWE) is based on the discontinuous Galerkin method also known as dG-FEM or simply dG. The method is very memory efficient and is based on a time explicit formulation. This means that it is not necessary to invert a full system matrix when stepping forward in time. Inversion of this matrix is necessary in time implicit methods and is very memory consuming for large problems. Because the CWE interface is not based on the classical FEM formulation, used in most of the other acoustics interfaces, other strategies apply for meshing and discretization.

The internal time stepping size of a time explicit method is strictly controlled by the CFL condition and thus the mesh size. Meaning that the smallest mesh elements will restrict the time steps (see Optimizing the Mesh for DG below). It turns out that the dG formulation has a sweet spot for speed and efficiency for wave problems. This is achieved by using fourth-order (quartic) shape functions (the default in the interface) and a mesh with the element size of about half the wavelength of the highest frequency component that needs to be resolved. In practice a mesh with size set to $\lambda_{min}/2$ to $\lambda_{min}/1.5$ can usually be used.

The interface is supported on all mesh types and for all combinations of mesh. The internal time step, used by the solver, is automatically calculated based on mesh size (inscribed circle metric), the wave speed and background mean flow speed, and the specifics of the solver selected. By default, the time explicit solver uses the Runge–

Kutta 4th order the method (RK4). This method is good as long as the mesh has a uniform mesh element size. If a mesh includes a large distribution of mesh element sizes (in particular in cases with only a few small elements) it can be advantageous to switch to the Adam-Bashforth 3 (local) method (AB3loc). This method uses intermediate local time steps in the region where there are small mesh elements. It is not expected that AB3loc will outperform RK4 for problems with a relatively uniform distribution of elements. For these cases RK4 is be the best choice.

For general information about the two methods see The Time-Explicit Algorithms in the *COMSOL Multiphysics Reference Manual*.

OPTIMIZING THE MESH FOR DG

As mentioned above, the internal time step, used by the solver, is controlled by the smallest mesh element in the model (relative to the speed of sound in the domain). So when meshing care should be taken to avoid small elements.

To visualize a metric for the time step used in elements, plot the variable `cwe.wtc`. The variable is known as the *cell wave time scale* and is proportional to the solver time step. The actual time step depends on the solver chosen. The global minimal value exists as the variable `cwe.wtcMin`. When plotting this variable set the **Resolution** to **No refinement** and the **Smoothing** to **None**, both settings in the **Quality** section of the plot. This can help identify problematic mesh element in the model. To filter out elements with the large add a **Filter** subnode to the plot and add a logic expression of the type: `cwe.wtc<1.1*cwe.wtcMin`.

Some important meshing and geometry strategies apply:

1. In 3D mesh, always use the **Avoid too small elements** option in the **Element Quality Optimization** settings on the **Free Tetrahedral** nodes in the mesh. The **Optimization Level** can be set to different degrees: **Basic**, **Medium**, or **High**. Using this option can greatly improve the mesh for dG applications and thus speed up the computation significantly. It is recommended to avoid mesh elements with short edges, since these are bad for the dG method.
2. In 2D mesh, a possible strategy is to create a mapped mesh. This will give a uniform mesh distribution. This however only works if a mapped mesh can be constructed. In general the mesh can be optimized by adding an **Adapt** node after meshing (one for each region with a given mesh size). In the **Adaptation** section set **Solution** to **None**, **Type of expression** to **Absolute size**, and then in the **Size expression** enter the mesh size for the domain. Then set the **Maximum number of refinements** to 0. This

operation will move mesh nodes around to try to maximize the mesh size and try to remove small mesh elements.

3 Avoid small edges and surfaces in the geometry as these control the mesh. Several tools exist to remedy this by using the **Virtual Operations** in the geometry.

4 Curved surfaces and boundaries need to be resolved adequately to ensure numerical stability. Typically using a **Curvature factor** of 0.3 to 0.4 will work well. If a curved boundary is not adequately resolved it may result in too large internal time-steps. At the same time too small elements should of course be avoided. Note that, the curvature factor will automatically be set to 0.3 if the **Calibrate for** option on the size nodes is set to **Time-explicit wave propagation**.

For general information about optimizing quality see Element Quality Optimization in the *COMSOL Multiphysics Reference Manual.*

For information about virtual operations see Virtual Geometry and Mesh Control Operations in the *COMSOL Multiphysics Reference Manual.*

Results: Variables and Quality

RESULT VARIABLES

Several variables are defined to be used when analyzing the results. They include the pressure, velocity components, and the norm of the velocity, as well as material parameters. The instantaneous intensity also exists as a variable (`cwe.Ii`). It is defined as

$$\mathbf{i} = (\rho_0\mathbf{u} + \rho\mathbf{u}_0)\left(\frac{p}{\rho_0} + \mathbf{u}\cdot\mathbf{u}_0\right)$$

Notice that the instantaneous intensity differs from the usual intensity which is an averaged value.

QUALITY

When analyzing the results from a simulation with the CWE interface, be aware of the fact that fourth order elements are used to discretize the dependent variables (as discussed in the section above). This, in some sense, means that within a mesh element, the shape function has a lot of freedom and can contain a lot of details. These details are revealed by setting a high **Resolution** in the **Quality** section in the plots. The default

plots generated already have that option set. If you add more user defined plots, remember to set the resolution.

Absorbing Layers

The Convected Wave Equation, Time Explicit Interface includes so-called *absorbing layers,* also often referred to as *sponge layers*. The layers work by combining three techniques: a scaling system, filtering, and simple nonreflecting conditions. For a review of the method see, for example, Ref. 5.

The layers are set up by adding the **Absorbing Layer** under the **Definitions** node. This adds a special scaled system. The scaling effectively slows down the propagating waves and ensures that they "align up" (normal) toward the outer boundary. This means that the waves will hit the outer boundary in a closer to normal direction. Filtering attenuates and filters out high-frequency components of the wave. The filter parameters are controlled under the Filter Parameters for Absorbing Layers section at the main physics level. Finally, at the outer boundary of the layer add a simple Specific Acoustic Impedance (Isentropic) condition, which will work well to remove all remaining waves, as normal incidence has been ensured.

For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual.*

The physical thickness of the layer should be such that it contains 2 to 3 mesh elements. This in practice means that it should roughly have a thickness equal to the largest wavelength resolved in the model. This is due to the meshing requirements described in the Meshing, Discretization, and Solvers section. You want to avoid to create small mesh elements in the absorbing layers. These can potentially slow down the solution procedure since the internal time stepping is controlled by the mesh size.

For the **Absorbing Layers** select the **Type** (Cartesian, cylindrical, spherical, or user defined) under the **Geometry** section. Enter values for the **Physical Width** and **Pole Distance** under the **Scaling** section.

For more details on the **Geometry** and **Scaling** see the Infinite Elements, Perfectly Matched Layers, and Absorbing Layers section in the *COMSOL Multiphysics Reference Manual.*

For the layers to work optimally the filter should not be too aggressive. Moreover, the scaled coordinates in the layer domain should also vary smoothly. To inspect the scaled system you can, for example, plot the coordinate variables `x_absorb_ab1`, `y_absorb_ab1`, and `z_absorb_ab1`. In 2D axisymmetric they are `r_absorb_ab1` and `z_absorb_ab1`. Note that the variables are not in the plot menu under the **Replace Expression**.

Using the absorbing layers with the three combined techniques will enable the reduction of spurious reflections by a factor between 100 and 1000 compared to the incident amplitude.

See the *Gaussian Pulse in 2D Uniform Flow: Convected Wave Equation and Absorbing Layers* tutorial model for an example. Application Library path **Acoustics_Module/Tutorials/gaussian_pulse_absorbing_layers**

Stabilizing Physical Instabilities (Filtering)

In the Convected Wave Equation Model (at the domain level), it is possible to activate a filter similar to the one used in the absorbing layer. This filter can be used to stabilize and suppress physical instabilities that can occur in the solution. These are well known in linearize-Euler-like equations, see Ref. 6. In the presence of a background flow, vorticity waves can occur; these are propagated with the background flow and are not acoustic waves.

Storing Solution on Selections for Large Models

Since the CWE interface is suited for solving large acoustic problems (measured in the number of wavelengths per geometry unit it can handle), the model solved can easily contain many million degrees of freedom (DOFs). Storing the solution with a desired time resolution can thus results in huge data files. To circumvent this, a good practice is to use the **Store in Output** option found in the main **Time Dependent** solver (for example, in the **Study 1** > **Step 1: Time Dependent** node). Using the **Selection** option in **Output**, it is possible to only store data on predefined selections. This can, for example, be on a symmetry plane, along an edge, or at the location of a receiver in a model. When postprocessing the data, best results are obtained by adding the same selections to the dataset used.

Assemblies and Pair Conditions

When setting up model based on the discontinuous Galerkin time explicit interfaces it can be advantageous to use assemblies in the geometry and pair conditions in the physics interface. This allows to use nonconforming meshes at the interface between, for example, two materials. Different materials are used in the different parts of the assembly. To couple the parts, at the physics level, the pair boundary conditions should be used. They are in general available from the **Pairs** submenu.

For more detailed information about assemblies, see The Form Union/Assembly Node — Uniting the Geometry section in the *COMSOL Multiphysics Reference Manual*.

For more detailed information about the pair features, see the Continuity on Interior Boundaries and Identity, Contact, and Sector Symmetry Pairs sections in the *COMSOL Multiphysics Reference Manual*.

Modeling with the Nonlinear Pressure Acoustics, Time Explicit Interface

This section contains information about:

- Solving Highly Nonlinear Problems
- Meshing
- Adaptive Mesh Refinement

Solving Highly Nonlinear Problems

When solving nonlinear problems the degree of nonlinearity, the number of harmonics generated and resolved, as well as the possibility of shock formation needs to be taken into account.

- For nonlinear problems where only a few harmonics are generated the default physics and solver settings will work well.
- For highly nonlinear problems (with no shock formation), where more than a few harmonics are generated, it is recommended to make some changes to the Time-Explicit Solver. Change the **Update time step** to **Manual**, this will ensure that the local speed of sound is reevaluated and the internal time step is updated to ensure numerical stability.
- For models that are highly nonlinear and exhibit shock formation, the use of the WENO limiter is necessary. This functionality is only available for linear discretization. Change the **Element order** to **Linear** in the **Discretization** section on the physics interface level. Then, in the **Limiter** section select **WENO**. When limiters are used and the problem is highly nonlinear changes in the Time-Explicit Solver are also necessary. The computation of discontinuous solutions requires that a Strong Stability Preserving (SSP) Runge–Kutta method be used. The third order SSP Runge–Kutta method is achievable by changing the **Order** of the Runge–Kutta method from the default 4 to 3. Moreover change the **Update time step** to **Manual**,

this will ensure that the local speed of sound is reevaluated and the internal time step is update to ensure numerical stability.

For an example of a highly nonlinear problem with shocks see the *Nonlinear Propagation of a Cylindrical Wave — Verification Model* tutorial model. Application Library path **Acoustics_Module/Nonlinear_Acoustics/nonlinear_cylindrical_wave**

Meshing

The same considerations for the mesh apply for the Nonlinear Pressure Acoustics, Time Explicit interface as for The Convected Wave Equation, Time Explicit Interface. Details are found in Meshing, Discretization, and Solvers under the modeling section for that physics. The interface is supported on all mesh types and for all combinations of mesh.

PHYSICS-CONTROLLED MESH

The Nonlinear Pressure Acoustics, Time Explicit interface supports physics-controlled mesh. To get an appropriate mesh, remember to set the value of the **Maximum frequency to resolve** in the Transient Mesh Settings section. The automatically generated mesh will use this value and the speed of sound defined in the model to set up an appropriate mesh that follows the best practices. This is often a good mesh to use as a starting suggestion.

Adaptive Mesh Refinement

One challenge when solving nonlinear problems is that the mesh should potentially resolve many harmonics to get accurate solutions. To remedy this, adaptive mesh refinement technology can be used. The method will automatically refine the mesh to resolve large gradients, that is, sharp signal details that include several harmonics. The approach is useful for modeling the propagation of spatially localized signals like tone bursts or Gaussian pulses.

To use the adaptive mesh refinement in a model follow these steps:

- Mesh to resolve only the fundamental frequency in the model.
- In the *Time Dependent* study step, under the **Adaptation** section, select **Adaptive mesh refinement**. Then generate the default solver sequence.

- Expand the solver tree and go to the **Adaptive Mesh Refinement** node. Some changes need to be done here. For the **Adaptation method** select **General modifications**. Do not enable **Allow coarsening** as the mesh needs to resolve the fundamental frequency. Finally, update the **Error indicator** expression to use the pressure gradient norm. In a 2D axisymmetric model the expression is `sqrt(comp1.pr^2+comp1.pz^2)` and in a 3D model the expression is `sqrt(comp1.px^2+comp1.py^2+comp1.pz^2)`.
- For highly localized signals, where the mesh adaptation generates a localized region with small elements, it can also be advantageous to switch from the default **Runge-Kutta** (RK4) solver method to the **Adams-Bashforth 3 (local)** method.

For an example of a nonlinear problem with pulse propagation that uses mesh adaptation see the *High-Intensity Focused Ultrasound (HIFU) Propagation Through a Tissue Phantom* tutorial model. Application Library path **Acoustics_Module/Nonlinear_Acoustics/hifu_tissue_sample**

Theory for the Convected Wave Equation Interface

The Convected Wave Equation, Time Explicit Interface theory is described in this section:

- Governing Equations of the Convected Wave Equation
- Boundary Conditions
- The Lax–Friedrichs Flux

Governing Equations of the Convected Wave Equation

The governing equations solved in the Convected Wave Equation (CWE) interface describe the propagation of linear acoustic waves (assuming an adiabatic equation of state) in the presence of a background flow. These equation are derived by Pierce in Ref. 1 (section 8.6) and Ref. 2 and are the equations solved by this interface,

$$\begin{aligned}
&\frac{\partial \rho}{\partial t} + (\mathbf{u}_0 \cdot \nabla)\rho + (\mathbf{u} \cdot \nabla)\rho_0 + \rho(\nabla \cdot \mathbf{u}_0) + \rho_0(\nabla \cdot \mathbf{u}) = f_\mathrm{p} \\
&\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}_0 \cdot \nabla)\mathbf{u} + (\mathbf{u} \cdot \nabla)\mathbf{u}_0 + \frac{1}{\rho_0}\nabla p - \frac{\rho}{\rho_0^2}\nabla p_0 = \mathbf{f}_\mathrm{v} \\
&\rho = \frac{p}{c_0^2}
\end{aligned} \tag{7-2}$$

Pierce argues for the use of the adiabatic assumption for the acoustic processes (perturbations in the entropy, $s = 0$) but also for not retaining the 0th order entropy variable s_0 (background mean flow entropy). The argument is that the entropy s only varies because of variations in the background fields (it is zero in a homogeneous medium). This leads to a term in the momentum equation that is second order in gradients of the background field, for example, $(\partial \mathbf{u}_0/\partial x)^2$ and so forth. These terms are disregarded. This also means that the equations are not valid when these terms are large, meaning when gradients in the background fields are large.

In order to fit into the discontinuous Galerkin (dG) formulation, the governing equations need to be put on a general conservative form of the type

$$\mathbf{d}_a \frac{\partial}{\partial t}\mathbf{U} + \nabla \cdot \Gamma(\mathbf{U}) = \mathbf{S} \tag{7-3}$$

where $\mathbf{U}$ is the vector containing the dependent variables (p, $\mathbf{u}$), $\mathbf{d}_a$ is the mass matrix of the system, Γ is the flux matrix, and $\mathbf{S}$ is the right-hand-side (RHS) source vector. The conservative form of Equation 7-2 is derived as follows. Start with Euler's equations on a conservative form (omitting the RHS). For now, the dependent variables represent the full fields (not the acoustic perturbations). The continuity, momentum, and equation of state can be written

$$\begin{gathered} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \\ \frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (p\mathbf{I} + \rho \mathbf{u}\mathbf{u}^{\mathrm{T}}) = \mathbf{0} \\ \rho = \rho(p) \end{gathered} \tag{7-4}$$

The equations describe the conservation of mass ρ and momentum flux $\rho\mathbf{u}$. Linearize these equations according to the usual scheme, using

$$p = p_0(\mathbf{x}) + p_1(\mathbf{x}, t) \qquad \mathbf{u} = \mathbf{u}_0(\mathbf{x}) + \mathbf{u}_1(\mathbf{x}, t) \qquad \rho = \rho_0(\mathbf{x}) + \rho_1(\mathbf{x}, t)$$

Now, insert these into Equation 7-4 and retain only 1st order terms (the acoustic perturbations)

$$\begin{gathered} \frac{\partial \rho_1}{\partial t} + \nabla \cdot (\rho_1 \mathbf{u}_0 + \rho_0 \mathbf{u}_1) = 0 \\ \rho_0 \frac{\partial \mathbf{u}_1}{\partial t} + \mathbf{u}_0 \frac{\partial \rho_1}{\partial t} + \nabla \cdot (p_1 \mathbf{I} + \rho_1 \mathbf{u}_0 \mathbf{u}_0^{\mathrm{T}} + \rho_0 \mathbf{u}_1 \mathbf{u}_0^{\mathrm{T}} + \rho_0 \mathbf{u}_0 \mathbf{u}_1^{\mathrm{T}}) = \mathbf{0} \\ \rho_1 = \frac{p_1}{c_0^2} \end{gathered} \tag{7-5}$$

This is the conservative form of the equations implemented in the CWE interface. In the remaining of this section, the subscript 1 will be omitted from the acoustic fields. The subscript 0 is kept on the variables that represent the background mean properties.

Equation 7-5 can now be put on the form given in Equation 7-3 yielding the following components

$$\mathbf{U} = \begin{bmatrix} p \\ u \\ v \\ w \end{bmatrix} \qquad \mathbf{d}_a = \begin{bmatrix} 1/c_0^2 & 0 & 0 & 0 \\ u_0/c_0^2 & \rho_0 & 0 & 0 \\ v_0/c_0^2 & 0 & \rho_0 & 0 \\ w_0/c_0^2 & 0 & 0 & \rho_0 \end{bmatrix} \qquad \Gamma(\mathbf{U}) = \begin{bmatrix} \Gamma(\mathbf{U})_p \\ \Gamma(\mathbf{U})_u \\ \Gamma(\mathbf{U})_v \\ \Gamma(\mathbf{U})_w \end{bmatrix} \tag{7-6}$$

with $\mathbf{u} = [u, v, w]^T$ and $\mathbf{u}_0 = [u_0, v_0, w_0]^T$, and the flux components are

$$\begin{aligned} \Gamma(\mathbf{U})_p &= \rho_0 \mathbf{u} + \frac{p}{c_0^2}\mathbf{u}_0 \\ \Gamma(\mathbf{U})_u &= \frac{p}{c_0^2}u_0\mathbf{u}_0 + \rho_0(u\mathbf{u}_0 + u_0\mathbf{u}) + \begin{bmatrix} p \\ 0 \\ 0 \end{bmatrix} \\ \Gamma(\mathbf{U})_v &= \frac{p}{c_0^2}v_0\mathbf{u}_0 + \rho_0(v\mathbf{u}_0 + v_0\mathbf{u}) + \begin{bmatrix} 0 \\ p \\ 0 \end{bmatrix} \\ \Gamma(\mathbf{U})_w &= \frac{p}{c_0^2}w_0\mathbf{u}_0 + \rho_0(w\mathbf{u}_0 + w_0\mathbf{u}) + \begin{bmatrix} 0 \\ 0 \\ p \end{bmatrix} \end{aligned} \tag{7-7}$$

Boundary Conditions

Within the dG formulation all boundary conditions are given in terms of the normal flux $-\mathbf{n}\cdot\Gamma$ at a given (exterior) boundary. Thus prescribing a boundary condition means specifying the vector $\mathbf{g}$ in the expression

$$-\mathbf{n}\cdot\Gamma = \begin{bmatrix} \rho_0(\mathbf{n}\cdot\mathbf{u}) + \frac{p}{c_0^2}(\mathbf{n}\cdot\mathbf{u}_0) \\ \frac{p}{c_0^2}u_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(u(\mathbf{n}\cdot\mathbf{u}_0) + u_0(\mathbf{n}\cdot\mathbf{u})) + n_1 p \\ \frac{p}{c_0^2}v_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(v(\mathbf{n}\cdot\mathbf{u}_0) + v_0(\mathbf{n}\cdot\mathbf{u})) + n_2 p \\ \frac{p}{c_0^2}w_0(\mathbf{n}\cdot\mathbf{u}_0) + \rho_0(w(\mathbf{n}\cdot\mathbf{u}_0) + w_0(\mathbf{n}\cdot\mathbf{u})) + n_3 p \end{bmatrix} = \mathbf{g} \tag{7-8}$$

This very general condition is the one available in the General Flux/Source condition. Details about the proper formulation of the flux terms entering the vector **g** can be found in Ref. 3.

The Lax–Friedrichs Flux

The Lax–Friedrichs (LF) flux parameter used in the definition of the Lax–Friedrichs numerical flux is given, elementwise for each dependent variable, by the expression

$$\tau^* = \tau_{LF}\begin{bmatrix} \frac{c_0 + u_0}{c_0^2} \\ (c_0 + u_0)\rho_0 \\ (c_0 + u_0)\rho_0 \\ (c_0 + u_0)\rho_0 \end{bmatrix} \tag{7-9}$$

where $u_0 = |\mathbf{u}_0|$ is the norm of the background mean flow velocity. The CWE interface uses the local value of the velocity. See Ref. 3 for further details on the numerical fluxes in general.

The value of the parameter τ_{LF} (the parameter that can be edited in the user interface) controls the numerical flux between the elements (nodal discontinuous Lagrange elements) used with the dG method. The numerical flux defines how adjacent elements are connected and how continuous p and $\mathbf{u}$ are. Different definitions of the numerical flux lead to different variants of the dG method. The flux implemented here is the so-called global Lax–Friedrichs numerical flux. The value of the parameter τ_{LF} should be between 0 and 0.5. For $\tau_{LF} = 0$ a so-called central flux is obtained. Setting $\tau_{LF} = 0.5$ gives a maximally dissipative global Lax–Friedrichs flux.

For general information about the numerical flux see the Numerical Flux section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

Theory for the Nonlinear Pressure Acoustics, Time Explicit Interface

The theory for The Nonlinear Pressure Acoustics, Time Explicit Interface is described in this section.

- Governing Equations for Nonlinear Pressure Acoustics, Time Explicit

Governing Equations for Nonlinear Pressure Acoustics, Time Explicit

The *Nonlinear Pressure Acoustics, Time Explicit* interface solves the nonlinear continuity equation and momentum equation that represent the physics described by the Westervelt equation (see Ref. 7). It is derived assuming that the cumulative nonlinear effects surpass the local nonlinear effects. That is, the contribution of the Lagrangian energy density is negligible:

$$L = \frac{1}{2}\rho|\mathbf{u}_t|^2 - \frac{p_t^2}{2\rho c^2} = 0$$

The governing equations are given by:

$$\begin{aligned} \frac{1}{\rho c^2}\frac{\partial p_t}{\partial t} + \nabla\cdot\left[\left(1+\frac{\beta p_t}{\rho c^2}\right)\mathbf{u}_t\right] &= Q_m \\ \rho\frac{\partial \mathbf{u}_t}{\partial t} + \nabla\cdot(p_t\mathbf{I}) &= \mathbf{q}_d \\ p_t = p \qquad \mathbf{u}_t &= \mathbf{u} \end{aligned} \tag{7-10}$$

where p_t is the total acoustic pressure, $\mathbf{u}_t$ is the total acoustic velocity fluctuations, and β is the coefficient of nonlinearity. Because the problem solved is nonlinear the superposition principle does not apply, which means that the "total" fields are always equal to the dependent variables.

Equation 7-10 is a nonlinear hyperbolic system written in the conservative form. The system allows solution discontinuities, and the conservative formulation is the only one that gives correct discontinuous (shock) solutions; see Ref. 8.

References for the Ultrasound Interface

1. A.D. Pierce, *Acoustics and Introduction to its Physical Principles and Applications*, Acoustical society of America, 1991.

2. A.D. Pierce, "Wave equation for sound in fluids with unsteady inhomogeneous flow," *J. Acoust. Soc. Am.*, vol. 87, p. 2293, 1990.

3. J.S. Hesthaven and T. Warburton, *Nodal Discontinuous Galerkin Methods: Algorithms, Analysis, and Applications*, Springer, 2008.

4. M.K. Myers, "On the Acoustic Boundary Condition in the Presence of Flow," *J. Sound Vibration*, vol. 73, pp. 429–434, 1980.

5. P.G. Petropoulos, L. Zhao, and A.C. Cangellaris, "A Reflectionless Sponge Layer Absorbing Boundary Condition for the Solution of Maxwell's Equations with High-Order Staggered Finite Difference Schemes," *J. Comp. Phys.*, vol. 139, pp. 184–208, 1998.

6. C.W. Tam, "Computational Aeroacoustics: Issues and Methods," *AIAA Journal*, vol. 33, 1995.

7. M.D. Diaz, M.A. Solovchuk, and T.W.H. Sheu, "A conservative numerical scheme for modeling nonlinear acoustic propagation in thermoviscous homogeneous media," *J. Comp. Phys.*, vol. 363, pp. 200–230, 2018.

8. E.F. Toro, *Riemann Solvers and Numerical Methods for Fluid Dynamics. A Practical Introduction*, 3rd Ed., Springer, 2009.

8

Geometrical Acoustics Interfaces

This chapter describes the physics interfaces found under the **Acoustics** > **Geometrical Acoustics** branch ( ).

- The Ray Acoustics Interface
- Modeling with the Ray Acoustics Interface
- Theory for the Ray Acoustics Interface
- The Acoustic Diffusion Equation
- Modeling with the Acoustic Diffusion Equation Interface
- Theory for the Acoustic Diffusion Interface

The Ray Acoustics Interface

The **Ray Acoustics (rac)** interface ( ), found under the **Acoustics** > **Geometrical Acoustics** subbranch ( ) when adding a physics interface, is used to compute the trajectories, phase, and intensity of acoustic rays. Ray acoustics is valid in the high-frequency limit where the acoustic wavelength is much smaller than the characteristic geometric features. The interface can be used to model room acoustics in concert halls, theaters, class rooms, or car cabins, but also for underwater acoustics, atmospheric acoustics, and other outdoor environments.

The properties of the media in which the rays propagate can be constant within domains, change continuously within domains (graded media), or change discontinuously at boundaries. At exterior boundaries it is possible to assign a variety of wall conditions, including combinations of specular and diffuse reflection. Impedance and absorption can depend on the frequency, intensity, and direction of incident rays. Transmission and reflection are also modeled at material discontinuities. A background velocity can also be assigned to any medium to, for example, model the effects of wind in atmospheric acoustics.

- For a best practice tutorial on room acoustics and impulse response computation, see the *Small Concert Hall Acoustics*. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/small_concert_hall**
- For a model of an actual chamber music hall model located in Berlin with comparison to measurements, see the *Chamber Music Hall*. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/chamber_music_hall**
- For an underwater acoustics tutorial, see the *Underwater Ray Tracing Tutorial in a 2D Axisymmetric Geometry* tutorial in the COMSOL Application Gallery: https://www.comsol.com/model/44711

For room acoustics applications, the impulse response can be determined in the results analysis by combining the information of the simulated rays collected by the Receiver and a **Receiver** dataset. The results are analyzed using the dedicated **Impulse Response** plot. The energy and level decay curves as well as room acoustic metrics can be

computed with the **Energy Decay** subfeature. For details, see the Impulse Response Plot and Receiver Dataset section in Modeling with the Ray Acoustics Interface.

The Release from Pressure Field and the Release from Exterior Field Calculation features are used for coupling results from a wave-based *Pressure Acoustics, Frequency Domain* simulation to the ray based *Ray Acoustics* simulation. They allow users to set up realistic sources based on detailed multiphysics wave-based simulation of a loudspeaker or another sound radiating source.

When this physics interface is added, these default nodes are also added to the **Model Builder**: **Medium Properties**, **Wall**, **Material Discontinuity**, and **Ray Properties**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Ray Acoustics** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `rac`.

RAY RELEASE AND PROPAGATION

By default, the **Allow frequency distributions at release features** checkbox is cleared. Select this checkbox to model propagation of rays of different frequencies simultaneously. The ray frequency can be specified at release features by entering a value or expression, sampling the frequency from a distribution, or entering a list of values. If this checkbox is cleared, the frequency is the same for all rays and is specified in the settings for the Ray Properties node.

Regardless of whether the **Allow frequency distributions at release features** checkbox is selected, properties at walls can depend on the frequency (variable `rac.f`) of each ray released. Domain material properties, like the speed of sound, can also depend on the

frequency. Entering a frequency distribution of the released rays also enables emitting more complex source signals (decomposed in their Fourier components).

To define the speed of sound or another medium property as a function of ray properties such as the ray frequency or intensity, the ray variable must be enclosed in the `noenv()` operator. For example, to use the ray frequency `rac.f` in an expression for the speed of sound, it must be included as part of the expression `noenv(rac.f)`. See also Medium Properties.

The **Maximum number of secondary rays** prevents an inordinate number of rays from being generated by capping them at the number supplied in the text field. The default is 500. Rather than being produced directly by release features such as the Source with Directivity node, secondary rays are released when an existing ray is subjected to certain boundary conditions. For example, when a ray undergoes refraction at a Material Discontinuity between different media, the incident ray is refracted and a reflected ray is created; the degrees of freedom for this reflected ray are taken from one of the available secondary rays, which are preallocated when the study begins.

If an insufficient number of secondary rays are preallocated, a reflected ray may not be released when an existing ray undergoes refraction. Thus, the **Maximum number of secondary rays** should be large enough that all reflected rays which significantly affect the solution can be released.

Only Store Receiver Data and Accumulated Variables in the Solution

By default the **Only store receiver data accumulated variables** checkbox is cleared. If this checkbox is selected, then the degrees of freedom associated with individual rays (such as the ray position, wave vector, intensity, power, and number of reflections) will be solved for but not stored in the solution data. However, the data collected by the Receiver feature and the degrees of freedom generated by certain physics features will be stored. This may reduce the size of the stored solution file significantly. When first setting up a model, it may be good to first solve including all the data, to more easily verify the solution. The degrees of freedom generated can be in the domain or on boundaries; these quantities, called accumulated variables, are defined by the following features:

- Accumulator (Boundary)
- Sound Pressure Level Calculation (subnode to some boundary conditions)

- Accumulator (Domain)
- Nonlocal Accumulator (subnode to Release from Boundary)

MATERIAL PROPERTIES OF EXTERIOR AND UNMESHED DOMAINS

Enter a value or expression for the **Speed of sound** c_{ext} (SI unit: m/s). The default value is 343 m/s. Then enter a value or expression for the **Density** ρ_{ext} (SI unit: kg/m^3). The default value is 1.2 kg/m^3.

When one of the **Intensity Computation** options is selected (see below), select an option from the **Attenuation type** list:

- For **Attenuation coefficient, Np per unit length** (the default), enter a value or expression for the **Attenuation coefficient** α_{ext} (SI unit: Np/m). The default is 0.
- For **Intensity attenuation coefficient, Np per unit length**, enter a value or expression for the **Intensity attenuation coefficient** m_{ext} (SI unit: Np/m). The default is 0.
- For **Attenuation coefficient, dB per unit length**, enter a value or expression for the **Attenuation coefficient** α'_{ext} (SI unit: dB/m). The default is 0.
- For **Attenuation coefficient, dB per wavelength**, enter a value or expression for the **Attenuation coefficient** $\alpha_{ext}^{(\lambda)}$ (SI unit: dB). The default is 0.

A more detailed explanation of each definition of the attenuation coefficient is given in the Medium Properties section.

These material properties are used when tracing rays outside the geometry or when tracing rays through domains that are not included in the selection of the Ray Acoustics physics interface. That is, meshless ray tracing. The medium properties in the deselected and exterior void domains must be constant and cannot be functions of any field variables, such as space or temperature. The properties like the attenuation can depend on a parameter like the band center frequency when modeling room acoustics applications, see the Impulse Response Plot and Receiver Dataset section.

INTENSITY COMPUTATION

Select an option from the **Intensity computation** list: **None**, **Compute intensity**, **Compute power** (the default), **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media**. For **None** the ray intensity and power are not computed.

- For **Compute intensity**, auxiliary dependent variables are used to compute the intensity of each ray. For a complete list of the auxiliary dependent variables that are defined, see Intensity and Wavefront Curvature in Theory for the Ray Acoustics Interface. This option is more accurate and is usually less computationally

demanding than **Compute intensity in graded media**, but is only valid for computing intensity in homogeneous (constant speed of sound) media.

- For **Compute power** (the default), the total power transmitted by each ray is defined as an auxiliary dependent variable, in addition to the auxiliary dependent variables that are declared when **Compute intensity** is selected. Like intensity, the power is affected by absorbing media and by reflection or refraction at boundaries. Unlike intensity, ray power does not change due to the convergence or divergence of a bundle of rays. When selected, the Sound Pressure Level Calculation subnode for the Wall feature, the Receiver, and the Release from Pressure Field are available. The **Compute power** option is necessary for the impulse response postprocessing, see Impulse Response Plot and Receiver Dataset.
- The option **Compute intensity and power** combines the dependent variables created by the **Compute intensity** and **Compute power** options.
- For **Compute intensity in graded media**, auxiliary dependent variables are used to compute the intensity of each ray. The intensity is affected by gradients in the speed of sound. Graded media may be present, for example, when modeling the effect of the saline climb in underwater acoustics. For a complete list of the auxiliary dependent variables that are defined, see Intensity and Wavefront Curvature in Theory for the Ray Acoustics Interface. This intensity computation method is valid for both homogeneous and graded media. If all media are homogeneous, meaning that the gradient of the speed of sound is zero everywhere except at material discontinuities, then it is recommended to select **Compute intensity** instead, since it is the more accurate intensity computation method for such cases.
- For **Compute intensity and power in graded media**, the total power transmitted by each ray is defined as an auxiliary dependent variable, in addition to the auxiliary dependent variables that are declared when **Compute intensity in graded media** is selected. The Sound Pressure Level Calculation subnode is available for the Wall feature, the Receiver, and the Release from Pressure Field are available. If all media are homogeneous then it is recommended to select **Compute intensity and power** instead.

If the ray intensity is solved for (one of the options **Compute intensity**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** is selected), the **Compute phase** checkbox is available. By default, the **Compute phase** checkbox is cleared. Select this checkbox to compute the phase along each ray. An auxiliary dependent variable for the phase `rac.Psi` is created. The phase changes along the ray trajectory based on its frequency and is also subjected to phase shifts that happen at walls with specular reflection and absorption.

When the ray intensity or power is solved for (one of the options **Compute intensity**, **Compute power**, **Compute intensity and power**, **Compute intensity in graded media**, or **Compute intensity and power in graded media** is selected), a list for the **Reference pressure for the sound pressure level** appears. Select **Use reference pressure for air** (the default) to set the reference pressure to 20 μPa, **Use reference pressure for water** to set the reference pressure to 1 μPa, or **User-defined reference pressure** to enter a value or expression in the text field. The reference pressure is used to calculate the sound pressure level (variable `rac.Lp`) and sound intensity level (variable `rac.LI`) along rays.

When **Compute intensity in graded media** or **Compute intensity and power in graded media** is selected from the **Intensity computation** list, enter a **Tolerance for curvature tensor computation** (dimensionless). This tolerance is used internally when computing the principal radii of curvature in a graded medium, and is used to define a criterion by which both principal radii of curvature are declared approximately equal. A larger tolerance makes the solution less accurate but more stable.

ADDITIONAL VARIABLES

The **Count reflections** checkbox is selected per default. It is used to create and allocate an auxiliary dependent variable for the number of reflections undergone by each ray, including reflections by the Wall and Material Discontinuity features. The auxiliary variable begins at 0 when rays are released and is incremented by 1 every time a ray is reflected at a boundary. The **Count reflections** option is necessary for the impulse response postprocessing, see Impulse Response Plot and Receiver Dataset.

By default, the **Store ray status data** checkbox is cleared. Select this checkbox to add new variables for quantities that cannot necessarily be recovered from the ray trajectory data alone. This is especially true if automatic remeshing has been used in a model. The following variables are created:

- The release time of a given ray (variable name `rac.rti`).
- The time at which a ray stopped at a boundary (variable name `rac.st`).
- The final status of the ray (variable name `rac.fs`). This indicates the status of a ray at the final time step. The value is an integer which has one of the following values:
  - 0 for unreleased rays
  - 1 for rays that are still in the modeling domain
  - 2 for frozen rays
  - 3 for stuck rays
  - 4 for rays that have disappeared.

To summarize the total number of rays having each final status, the following global variables are also defined.

TABLE 8-1: GLOBAL VARIABLES BASED ON RAY STATUS.

NAME	DESCRIPTION
`fac`	Fraction of active rays at final time step
`fds`	Fraction of disappeared rays at final time step
`ffr`	Fraction of frozen rays at final time step
`fse`	Fraction of secondary rays released
`fst`	Fraction of stuck rays at final time step
`nsr`	Number of released secondary rays
`nsrf`	Number of released secondary rays at final time step
`nsu`	Number of unreleased secondary rays
`nsuf`	Number of unreleased secondary rays at final time step

The global variable names in Table 8-1 all take the unreleased secondary rays into account. For example, suppose an instance of the Ray Acoustics interface includes 100 primary rays and 100 allocated secondary rays. At the last time step, suppose that 80 of the primary rays have disappeared at boundaries and that 40 secondary rays have been emitted, all of which are still active. Then the variable `rac.fac`, the fraction of active rays at the final time step, would have the value (20 + 40)/(100 + 100) or 0.3.

ADVANCED SETTINGS

This section is only shown when **Advanced Physics Options** are enabled (click the **Show More Options** button in the **Model Builder** toolbar).

The **Wall accuracy order** sets the accuracy order of the time stepping used for time steps during which a ray-wall interaction happens. Select an order of **1** to use a forward Euler step and compute the motion both before and after the wall collision. Select an order of **2** (the default) to use a second-order Taylor method and compute the motion before the wall collision. After the collision a second-order Runge-Kutta method is used.

Select an option from the **Arguments for random number generation** list: **Generate unique arguments**, **Generate random arguments**, or **User defined**. This setting determines how the additional argument to random functions is defined in features such as the Wall boundary condition with the **Diffuse scattering** wall condition. Typically, the random

numbers are functions of the ray index, time, a unique input argument for different variable definitions, and another argument i, defined as follows:

- For **Generate unique arguments,** the additional argument is based on the position of each node in the Model Builder. As a result, random numbers generated in different nodes are created independently of each other.
- For **Generate random arguments**, the additional argument is randomly created each time the study is run.
- For **User defined**, the additional argument is defined by a user input in the **Settings** window for each feature. Uncorrelated sets of random numbers can be obtained by running a **Parametric Sweep** for different values of i.

Note that the `random` function is a pseudorandom number generator (PRNG), not a true random number generator (RNG) in the sense that an observer with perfect knowledge of the algorithm and a history of previously generated values could predict the ensuing values. However, PRNG is sufficient for most purposes.

For any model using PRNG, it is recommended to test for statistical convergence by varying the number of rays and observing the effect on some ray statistics, such as average final ray position or transmission probability. This is analogous to the mesh refinement studies that are typically recommended to validate finite element models.

The **Allow multiple release times** checkbox, which is cleared by default, allows an array of release times for the rays to be specified in any of the ray release features. If the checkbox is cleared, all rays are released at time $t = 0$.

By default the **Exclude Jacobian contributions from ray equations** checkbox is cleared. If this checkbox is selected, then expressions for the time derivatives of the degrees of freedom on rays are not included in the Jacobian matrix. Excluding contributions to the Jacobian can significantly improve solution time. The drawback is that the Jacobian is not exact, so smaller time steps or path length intervals may be needed to obtain an accurate solution. This drawback is most noticeable when computing the ray intensity or power in attenuating media.

By default, the **Treat unmeshed releasing entities as errors** checkbox is selected. Certain ray release features, such as the **Release** and **Release from Boundary** features, compute the ray release positions based on a geometric entity, which must be meshed. While this checkbox is selected, failure to mesh the selections of such features will result in an error message when running any study in the model. If this checkbox is cleared, failure to mesh the selections of such features will result in them releasing zero rays, but will

otherwise not interrupt the computation. Other physics features will still be able to release rays normally.

Enter a value for the **Maximum number of wall interactions per time step**. The default value is 1000. If a ray undergoes more than the specified number of boundary interactions in a single time step taken by the solver, the ray will disappear. This is included as a safeguard to prevent rays from getting stuck in infinite loops if the time between successive ray-wall interactions becomes infinitesimally small.

DEPENDENT VARIABLES

The dependent variables (field variables) are the **Ray position components** and **Wave number components**. The name can be changed, but the names of fields and dependent variables must be unique within a model.

Theory for the Ray Acoustics Interface

Domain, Boundary, and Global Nodes for the Ray Acoustics Interface

The Ray Acoustics Interface has these domain, boundary, and global nodes available (listed in alphabetical order).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Accumulator (Boundary)
- Accumulator (Domain)
- Auxiliary Dependent Variable
- Axial Symmetry
- Background Velocity
- Release from Boundary
- Release from Symmetry Axis
- Material Discontinuity
- Medium Properties
- Nonlocal Accumulator
- Ray Continuity
- Ray Detector
- Ray Properties
- Ray Termination
- Receiver
- Release
- Release from Data File
- Release from Edge
- Release from Exterior Field Calculation
- Release from Grid
- Release from Grid on Axis
- Release from Point
- Release from Point on Axis
- Release from Pressure Field
- Sound Pressure Level Calculation
- Source with Directivity
- Wall

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Medium Properties

Use the **Medium Properties** node to specify the speed of sound and density of the medium. This node is created by default and includes all of the selected domains for the physics interface. When the model is solved with unmeshed domains, the material data is specified in the Material Properties of Exterior and Unmeshed Domains.

MODEL INPUTS

Select, **User defined** (the default), **Common model input**, or an input from another physics interface, if applicable.

The values of the quiescent (or background) **Temperature** T (SI unit: K) and **Absolute pressure** p_A (SI unit: Pa) are entered in the **Model Inputs** section if required by the material properties (other inputs like relative humidity or salinity can also appear). The values of the model inputs can influence the material properties like, for example, the

density and speed of sound, depending on their definition in the **Materials** node. In general, if the model includes a material property that depends on a model input, the corresponding text field will be enabled.

Details about the Model Input and the Default Model Inputs are found in the Global and Local Definitions chapter of the *COMSOL Multiphysics Reference Manual.*

PRESSURE ACOUSTICS MODEL

If the ray intensity or power is computed in the model, select an option from the **Fluid model** list: **Linear elastic** (the default), **User-defined attenuation**, **Atmosphere attenuation**, **Ocean attenuation**, or **Thermally conducting and viscous**. This section determines which additional fluid properties can be specified in this section.

The **Speed of sound** c (Si unit: m/s) is always required. By default it uses values **From material**. For **User defined** enter a value or expression. The default is 343 m/s.

If the ray intensity or power is solved for, both the density and the speed of sound are required to compute the transmittance and reflectance at boundaries between different media. The default **Density** ρ (SI unit: kg/m^3) uses values **From material**. For **User defined**, enter a value or expression. The default is 1.2 kg/m^3.

Additional inputs are shown in this section depending on the option selected from the **Fluid model** list:

- If **Linear elastic** is selected, absorption by the medium is neglected, so no other inputs are required.
- If **User-defined attenuation** is selected, select an option from the **Attenuation type** list. The default is **Attenuation coefficient, Np per unit length**.
  - For **Attenuation coefficient, Np per unit length** enter a value or expression for the **Attenuation coefficient** α (SI unit: Np/m). The default is 0. Note that this is the attenuation coefficient of the pressure amplitude rather than the intensity. The

ray intensity (neglecting wavefront curvature terms) actually scales as $I(x) = I_0\exp(-2\alpha x)$, where x is the propagation distance.

- For **Intensity attenuation coefficient, Np per unit length** enter a value or expression for the **Intensity attenuation coefficient** m (SI unit: Np/m). The default is 0. Neglecting curvature terms, the ray intensity decreases as $I(x) = I_0\exp(-mx)$.

In most room acoustics references, the attenuation coefficient given is the intensity attenuation coefficient m. When setting up models, it is important to know which definition is used and choose between the preceding two options accordingly.

- For **Attenuation coefficient, dB per unit length**, enter a value or expression for the **Attenuation coefficient** α' (SI unit: dB/m). The default is 0.
- For **Attenuation coefficient, dB per wavelength**, enter a value or expression for the **Attenuation coefficient** $\alpha^{(\lambda)}$ (SI unit: dB). The default is 0.

- For **Atmosphere attenuation** enter the following in the **Model Input** section:
  - **Temperature** T (SI unit: K, default 293.15 K)
  - **Absolute pressure** p_A (SI unit: Pa, default 1 atm)
  - **Relative humidity** ϕ_w (dimensionless, default 0)

  By default, these model inputs are all **User defined**, but they can also be taken from a **Common model input**.
- For **Ocean attenuation** enter the following in the **Model Input** section:
  - **Temperature** T (SI unit: K, default 293.15 K)
  - **Practical salinity** S_p (dimensionless, default 35)
  - **Depth** D (SI unit: m, default 0)

  By default, these model inputs are all **User defined**, but they can also be taken from a **Common model input**. Also, in the **Pressure Acoustics Model** section, enter a value or expression for the **pH value** pH (dimensionless). The default is 8.

The **Practical salinity** is defined on the Practical Salinity Scale. It represents a specific way of measuring salinity through electric conductivity. Practical salinity is a unitless quantity, although it can be thought of as given in units of g/kg. The default value for the **Practical salinity** S_p is 35. For details, see Ref. 22.

- If **Thermally conducting and viscous** is selected, specify the following material properties. By default, all fluid properties take default values **From material**. Choose **User defined** to enter a value or expression for the:
  - **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)). The default value is 1005.4 J/(kg·K).
  - **Ratio of specific heats** γ (dimensionless). The default value is 1.4.
  - **Thermal conductivity** k (SI unit: W/(m·K)). The default value is 0.0257 W/(m·K). For ray acoustics, the thermal conductivity is assumed to be isotropic.
  - **Dynamic viscosity** μ (SI unit: Pa·s). The default value is 0.0181 mPa·s.
  - **Bulk viscosity** μ_B (SI unit: Pa·s). The default is 0.

To define the attenuation coefficient or another medium property as a function of ray properties such as the ray frequency or intensity, the ray variable must be enclosed in the `noenv()` operator. For example, to use the ray frequency `rac.f` in an expression for the speed of sound, it must be included as part of the expression `noenv(rac.f)`.

Attenuation Within Domains

Release

Use the **Release** node to release rays within domains based on arbitrary expressions or based on the positions of the mesh elements.

RELEASE TIMES

This section is only available when the **Allow multiple release times** checkbox has been selected in the physics interface **Advanced Settings** section. Select a **Distribution function**: **List of values** (default), **Uniform**, **Normal**, or **Lognormal**.

List of Values

Enter **Release times** (SI unit: s) or click the **Range** button ( ) to select and define a range of specific times. At each release time, rays are released with initial position and ray direction vector as defined in the following sections.

Uniform

Enter the **Number of values**, along with the **First time value** (SI unit: s) and the **Last time value** (SI unit: s). In addition, select whether the **Sampling from distribution** should be **Deterministic** or **Random**. When **Deterministic** is selected, an array of length **Number of**

values of uniformly spaced release times is generated. This array starts with the **First time value** and ends with the **Last time value** exactly. The release times are reproducible each time the solution is computed.

When **Random** is selected, an array of random numbers of length **Number of values**, with a minimum lower limit of the **First time value** and maximum upper limit of the **Last time value** is generated. In this case, a release time of exactly the first and last time values is extremely unlikely.

Normal and Lognormal

Enter the **Number of values** along with the **Mean** (SI unit: s) and the **Standard deviation** (SI unit: s). In addition, select whether the **Sampling from distribution** should be **Deterministic** or **Random**. When **Deterministic** is selected, an ordered array of length **Number of values** of normally distributed release times is generated. The release times are more closely spaced around the value entered for the **Mean** and the spacing of the release time drops off according to the value of the **Standard deviation**. The release times are reproducible each time the solution is computed.

When **Random** is selected, the normal or lognormal distribution is generated by pseudorandom sampling, so there may be some statistical error.

INITIAL POSITION

Select an **Initial position**: **Density** (the default) or **Mesh based**.

Density

For **Density** enter a value for the **Number of rays per release** N (dimensionless). The default is 100. Then enter a value or expression for the **Density proportional to** ρ (dimensionless). The default is 1.

The **Density proportional to** ρ can be an expression rather than a number; the resulting ray distribution approximately has a number density that is proportional to this expression. The resulting distribution looks a bit random, and it depends on the order in which the mesh elements are numbered. The distribution is probably not exactly the same in different COMSOL Multiphysics versions, but the total number of rays released is always N.

The **Density proportional to** expression must be strictly positive.

Advanced Options for Density-Based Release

The following options can be adjusted to make the number density of rays more closely conform to the **Density proportional to** expression.

Select a **Release distribution accuracy order** between **1** and **5** (the default is **5**), which determines the integration order that is used when computing the number of rays to release within each mesh element. The higher the accuracy order, the more accurately rays will be distributed among the mesh elements.

The **Position refinement factor** (default 0) must be a nonnegative integer. When the refinement factor is 0, each ray is always assigned a unique position, but the density is taken as a uniform value over each mesh element. If the refinement factor is a positive integer, the distribution of rays within each mesh element is weighted according to the density, but it is possible for some rays to occupy the same initial position. Further increasing the **Position refinement factor** increases the number of evaluation points within each mesh element to reduce the probability of multiple rays occupying the same initial position.

Mesh Based

For **Mesh based** the rays are released from a set of positions determined by a selection of geometric entities (of arbitrary dimension) in the mesh. Given a **Refinement factor** between 1 and 5, the centers of the refined mesh elements are used. Thus, the number of positions per mesh element is `refine^dim`, except for pyramids, where it is `(4*refine^2-1)*refine/3`.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Expression** (the default), **Spherical**, **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **Expression** a single ray is released in the specified direction. Enter coordinates for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless) based on space dimension.
- For **Spherical** a number of rays are released at each point, sampled from a spherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50.
- For **Hemispherical** a number of rays are released at each point, sampled from a hemispherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis** $\mathbf{r}$ based on space dimension.
- For **Conical** a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w

(dimensionless). The default is 50. Then enter coordinates for the **Cone axis r** based on space dimension. Then enter the **Cone angle** α (SI unit: rad; default $\pi/3$).

- The **Lambertian** option is only available in 3D. A number of rays are released at each point, sampled from a hemisphere in wave vector space with probability density based on the cosine law. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension.

If **Conical** is selected in a 3D model, select an option from the **Conical distribution** list:

- **Uniform density** (the default): rays are released with polar angles from 0 to the specified cone angle. The rays are distributed in wave vector space so that each ray subtends approximately the same solid angle.
- **Specify polar and azimuthal distributions**: specify the **Number of polar angles** N_θ (dimensionless) and the **Number of azimuthal** angles N_φ (dimensionless). Rays are released at uniformly distributed polar angles from 0 to the specified cone angle. A single axial ray ($\theta = 0$) is also released. For each value of the polar angle, rays are released at uniformly distributed azimuthal angles from 0 to 2π. Unlike other options for specifying the conical distribution, it is not necessary to directly specify the **Number of rays in wave vector space** N_w (dimensionless), which is instead derived from the relation $N_w = N_\theta \times N_\varphi + 1$.
- **Hexapolar**: specify the **Number of polar angles** N_θ (dimensionless). In this distribution, for each release point, one ray will be released along the cone axis. Six rays are released at an angle α/N_θ from the cone axis, then 12 rays at an angle of $2\alpha/N_\theta$, and so on. The total number of directions is $N_w = 3N_\theta(N_\theta + 1) + 1$.
- **Flat**: rays are released in a flat fan shape within the specified angle.

- **Marginal rays only**: the rays are all released at an angle α with respect to the cone axis. The rays are released at uniformly distributed azimuthal angles from 0 to 2π.
- **Marginal and axial rays only**: the rays are all released at an angle α with respect to the cone axis, except for one ray which is released along the cone axis. The marginal rays are released at uniformly distributed azimuthal angles from 0 to 2π.

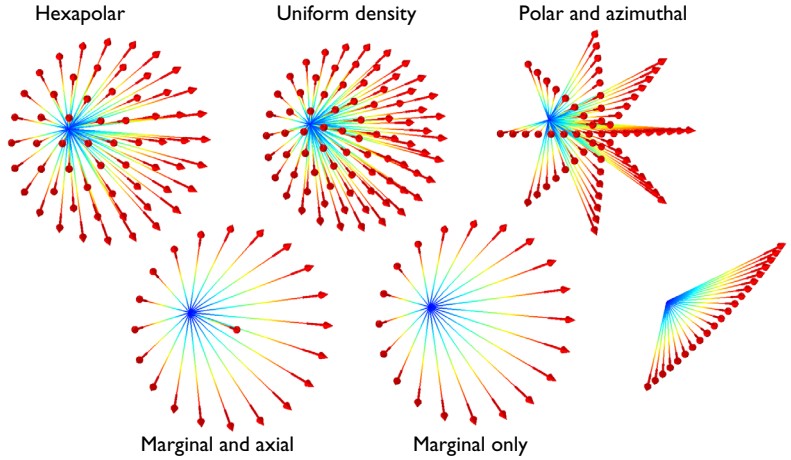


Figure 8-1: Comparison of the available cone-based release distributions.

In 3D for the **Conical distribution** you can also let the **Transverse direction** be **Automatic** (the default) or **User defined**. For **User defined** enter the components of $\mathbf{e}_t$. This controls, for example, the orientation of the ray fan when **Flat** is selected.

For **Spherical**; **Hemispherical**; **Conical** with **Flat** or **Uniform density** distribution; and **Lambertian**, select an option from the **Sampling from Distribution** list: **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray direction vectors are computed using an algorithm that seeks to distribute the rays as evenly as possible in wave vector space. This algorithm will give the same initial ray directions whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudorandom numbers. The result may be the same when rerunning the study multiple times on the same computer, but the solution is likely to be different on different architectures.

For **Expression** it is also possible to initialize the ray direction vector either in the global coordinate system or in a coordinate system that moves with the same velocity as the

background medium. Select an option from the **Initial wave vector specification** list: **With respect to fluid** (the default) or **With respect to coordinate system**.

- For **With respect to fluid** the initial wave vector is computed with respect to a coordinate system that moves at the background velocity, so the initial ray direction might not be parallel to the vector entered in the **Ray direction vector** text field if the medium is moving.
- For **With respect to coordinate system** the initial ray direction is parallel to the vector entered in the **Ray direction vector** text field as long as a ray could reasonably propagate in that direction. For example, rays cannot be released in certain directions if the background fluid is moving with a supersonic velocity.

INITIAL RAY FREQUENCY

This section is available when the **Allow frequency distributions at release features** checkbox is selected under the physics interface **Ray Release and Propagation** section.

Select a **Distribution function**: **None** (the default), **Normal**, **Lognormal**, **Uniform**, or **List of values**.

When **None** is selected, enter an initial value f_0 (SI unit: Hz). The default value is 1000 Hz.

Select **Normal** to create a normal distribution function, **Lognormal** to create a log-normal distribution function, or **Uniform** to create a uniform distribution function. For any of these distributions, select an option from the **Sampling from distribution** checkbox: **Deterministic** (the default) or **Random**. For **Random** sampling the mean and standard deviation may not be exactly equal to the specified values but will statistically converge as the number of rays is increased. The **Number of values** sets the number of values that are sampled from the distribution function at each release point.

For the **Normal** or **Lognormal** distribution enter a user-defined **Mean ray frequency** (default 1000 Hz) and **Ray frequency standard deviation** (default 100 Hz). For the **Uniform** distribution enter the **Minimum ray frequency** $f_{\min}$ (default 1000 Hz) and **Maximum ray frequency** $f_{\max}$ (default 2000 Hz). Select **List of values** to enter a list of distinct frequency values.

INITIAL PHASE

This section is available when the **Compute phase** checkbox is selected under the physics interface **Intensity Computation** section. Enter an **Initial phase** Ψ_0 (SI unit: rad). The default value is 0.

INITIAL INTENSITY

This section is available when the ray intensity is solved for in the model and **Expression** is selected as the **Ray direction vector**. Enter a value for the **Initial intensity** I_0 (SI unit: $\mathrm{W/m^2}$). The default is $1000\ \mathrm{W/m^2}$.

INITIAL RADII OF CURVATURE

This section is available when the ray intensity is solved for in the model and **Expression** is selected as the **Ray direction vector**. Select a **Wavefront shape**. In 3D the available options are **Plane wave**, **Spherical wave**, and **Ellipsoid** (the default). In 2D the available options are **Plane wave** and **Cylindrical wave** (the default). In 2D axisymmetric models the available options are **Plane wave**, **Cylindrical wave** (the default), and **Spherical wave**.

- For an idealized plane wave the radii of curvature would be infinite. However, because the algorithm used to compute intensity requires finite values, when **Plane wave** is selected the initial radii of curvature are instead given an initial value that is 10^8 times greater than the characteristic size of the geometry.
- For a **Spherical wave** or **Cylindrical wave**, enter the **Initial radius of curvature** r_0 (SI unit: m).
- For an **Ellipsoid**, enter the **Initial radius of curvature, I** $r_{1,0}$ (SI unit: m) and the **Initial radius of curvature, 2** $r_{2,0}$ (SI unit: m). Also enter the **Initial principal curvature direction, I** $\mathbf{e}_{1,0}$ (dimensionless).

For spherical and cylindrical waves the **Initial radius of curvature** must be nonzero. To release a ray such that the initial wavefront radius of curvature is zero, instead select a different option such as **Conical** from the **Ray direction vector** list.

Principal Radii of Curvature

TOTAL SOURCE POWER

This section is available:

- when the ray intensity is solved for, and
- when **Spherical**, **Hemispherical**, or **Conical** is selected as the **Ray direction vector**.
- It is also available when the ray power is solved for, and then any choice of **Ray direction vector** displays this section.

Select an option from the **Intensity initialization** list: **Uniform distribution** (the default) or **Weighted distribution**.

If **Uniform distribution** or **Weighted distribution** is selected, enter a **Total source power** P_{src} (SI unit: W). The default is 1 W. In 2D, instead enter the **Total source power per unit thickness** P_{src} (SI unit: W/m). The default is 1 W/m. For **Weighted distribution**, also enter an expression for the **Power weighting factor** P_{wt}. The weighting factor may have any unit. The released rays will have initial intensity and power proportional to the weighting factor, while still adding up to the specified source power.

For example, if you release 1000 rays in a **Spherical** distribution with an initial power of 10 W, and enter a weighting factor of `rac.niz+1`, then the sum of the power over all released rays, `rac.sum(rac.Q)`, will equal 10 W. Since `rac.niz` is the z-component of the ray direction vector, rays close to the z-axis will each have power of about 0.02 W, those around the negative z direction will have almost no power, and those in the xy-plane will each have power of about 0.01 W.

INITIAL VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

For each of the **Auxiliary Dependent Variable** nodes added to the model, select a **Distribution function** for the initial value of the auxiliary dependent variables and whether the initial value of the auxiliary dependent variables should be a scalar value or sampled from a distribution function.

The number of rays simulated can increase substantially and the following options are available for each **Auxiliary Dependent Variable** added to the model.

When **None** is selected, enter an initial value. The symbol for the initial value is the auxiliary variable name followed by a subscript 0, so for the default name `rp` the initial value has symbol rp_0.

For the initial value of the auxiliary dependent variables, select **Normal** to create a normal distribution function, **Lognormal** to create a log-normal distribution function, or **Uniform** to create a uniform distribution function. For any of these distributions, select an option from the **Sampling from distribution** checkbox: **Deterministic** (the default) or **Random**. For **Random** sampling the mean and standard deviation may not be exactly equal to the specified values but will statistically converge as the number of rays

is increased. The **Number of values** sets the number of values that are sampled from the distribution function at each release point.

For the **Normal** or **Lognormal** distribution enter the **Mean** (default 0) and **Standard deviation** (default 1). For the **Uniform** distribution enter the **Minimum** (default 0) and **Maximum** (default 1). Select **List of values** to enter a set of numerical values directly.

By default, auxiliary dependent variables are initialized after all other degrees of freedom. Select the **Initialize before wave vector** checkbox to compute the initial value of the auxiliary dependent variable immediately after computing the initial wave vectors of the rays. By selecting this checkbox, it is possible to define the initial ray direction as a function of the auxiliary dependent variables.

Accumulator (Domain)

Use the **Accumulator** node to define additional degrees of freedom on a domain. Each **Accumulator** defines a variable, called the accumulated variable, on each domain element in the selection list. The values of the accumulated variables are determined by the properties of rays in each domain element. The accumulated variable may be influenced by all rays or may be restricted to the rays released by specific release feature.

ACCUMULATOR SETTINGS

Select an **Accumulator type**: **Density** (default) or **Count**.

- For **Density**, the accumulated variable is divided by the volume of the mesh element where it is defined.
- For **Count**, the accumulated variable is unaffected by the element size.

Select an option from the **Accumulate over** list: **Elements** (the default) or **Elements and time**.

- For **Elements**, the value of the accumulated variable in an element is the sum of the source terms of all rays in that element. If the **Accumulator type** is set to **Density**, this sum is divided by the mesh element volume.
- For **Elements and time**, the time derivative of the accumulated variable in an element is the sum of the source terms of all rays in that element. If the **Accumulator type** is set to **Density**, this sum is divided by the mesh element volume. As each ray propagates through a series of mesh elements, it leaves behind a contribution to the accumulated variable that remains even after the ray has moved on.

Enter the **Accumulated variable name**. The default is `rpd`. The accumulated variable is defined as `<name>.<varname>`, where `<name>` is the physics interface name and `<varname>` is the accumulated variable name. For example, in an instance of the Ray Acoustics interface with default name `rac` and default accumulated variable name `rpd`, the variable would be named `rac.rpd`.

Enter a **Source** R. The unit of the source depends on the settings in the **Units** section. The source term is used to calculate the accumulated variable in a manner specified by the **Accumulate over** and **Accumulator type** settings.

If **Elements and time** is selected from the **Accumulate over** list, select an option from the **Source interpolation** list: **Constant**, **Linear** (the default), **Quadratic**, or **Exponential**. The **Source interpolation** determines what functional form the **Source** is assumed to follow during each time step taken by the solver. This information is used to compute the accumulated variable in mesh elements that the rays pass through during each time step.

Select an option from the **Release feature** list. If **All** (the default) is selected, the accumulated variable collects information about all rays in the selected domains or boundaries, regardless of how they were released. Alternatively, select a ray release feature from the list, and then only the rays produced by that release feature are included.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

Background Velocity

The **Background Velocity** node is used to assign a velocity to the medium in which the rays propagate. When no **Background velocity** node is present, the medium is assumed to be stationary.

BACKGROUND VELOCITY

The default **Velocity field** $\mathbf{u}$ (SI unit: m/s) is **User defined**. Enter values or expressions for the vector components.

Material Discontinuity

The **Material Discontinuity** node is the default feature on interior boundaries.

The Ray Acoustics interface always applies reflection and refraction at boundaries between different media using a deterministic ray splitting approach. The direction of the refracted ray is computed using Snell's law, based on the speed of sound on either side. If extra degrees of freedom have been allocated for secondary rays, a reflected ray is also released. If the incident ray undergoes total internal reflection, no refracted ray is produced and no secondary rays are needed to release the reflected ray.

If the ray intensity or power is solved for in the model, the **Material Discontinuity** feature computes the new values of these variables for the reflected and refracted rays.

The maximum number of reflected rays can be controlled via the **Maximum number of secondary rays** text field, which is found under the **Ray Release and Propagation** section for the physics interface.

The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

RAYS TO RELEASE

Use the inputs in this section to control whether the incident ray is split into a reflected and refracted ray at the material discontinuity. Select an option from the **Release reflected rays** list: **Always** (the default), **Never**, or **Based on logical expression**.

- If **Always** is selected, whenever a ray hits the material discontinuity, the incident ray is refracted across the boundary, and a reflected ray is produces using some of the preallocated degrees of freedom for secondary rays.
- If **Never** is selected, the incident ray is refracted, but no reflected ray is produced. This does not prevent a ray from undergoing total internal reflection at the boundary.
- If **Based on logical expression** is selected, enter a value or expression for the **Evaluation expression** e (dimensionless). The default is 1. A reflected ray is produced only if the value of this expression is nonzero for the incident ray.

If the ray intensity is solved for in the model, enter a **Threshold intensity** I_{th} (SI unit: $\mathrm{W/m^2}$). The default is $1\ \mathrm{mW/m^2}$. If the interaction of a ray with a material discontinuity would create a reflected ray of intensity less than the threshold intensity, the release of this reflected ray is suppressed.

If the ray power is solved for in the model, enter a **Threshold power** Q_{th} (SI unit: W). The default is 10^{-3} mW. If the interaction of a ray with a material discontinuity would create a reflected ray of power less than the threshold power, the release of this reflected ray is suppressed.

If both the **Evaluation expression** and the **Threshold intensity** are specified, a reflected ray is only released if the value of the expression is nonzero and the intensity of the reflected ray exceeds the threshold. In other words, a ray must satisfy all of the criteria in the **Rays to Release** section in order to be released, not just one of them.

The release of reflected rays at a material discontinuity can be suppressed to prevent an arbitrarily large number of degrees of freedom from being used to model rays of extremely low intensity. Keep in mind that this can lead to a reduction in the total energy of the system.

AUXILIARY DEPENDENT VARIABLES, REFRACTED RAY

If an Auxiliary Dependent Variable has been added to the model then there is an option to reinitialize the values of auxiliary dependent variables for the refracted ray. These settings are the same as in the New Value of Auxiliary Dependent Variables section for the Wall feature.

AUXILIARY DEPENDENT VARIABLES, REFLECTED RAY

If an Auxiliary Dependent Variable has been added to the model then there is an option to reinitialize the values of auxiliary dependent variables for the reflected ray. These settings are the same as in the New Value of Auxiliary Dependent Variables section for the Wall feature.

ADVANCED SETTINGS

Select the **Show boundary normal** checkbox to view the boundary normal in the **Graphics** window.

Material Discontinuity Theory

Wall

Use the **Wall** node to determine what happens to the rays when contact with a boundary is made. The **Wall** node is the default boundary condition on all exterior boundaries.

The Accumulator (Boundary) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

WALL CONDITION

Select a **Wall condition**: **Freeze** (the default), **Specular Reflection**, **Stick**, **Disappear**, **Pass through**, **Diffuse scattering**, **Isotropic scattering**, **Mixed diffuse and specular reflection**, or **General reflection**.

TABLE 8-2: WALL CONDITION OPTIONS.

OPTIONS	DESCRIPTION
Freeze	Select to fix the ray position and wave vector at the instant a wall is struck. So, the ray position no longer changes after contact with the wall and the wave vector of the ray remains at the same value as when the ray struck the wall. This boundary condition is typically used to recover the ray intensity or phase at the instant contact was made with the wall.
Specular Reflection	Select to specularly reflect from the wall, so that the angle of incidence and the angle of reflection are equal.
Stick	Select to fix the ray position at the instant the wall is struck. The wave vector components are set to zero.
Disappear	This option means that the ray is not displayed once it has made contact with the wall. Use it if the ray location after contact with the wall is not of interest.
Pass through	This option allows rays to cross the boundary unimpeded. It does not cause rays to be refracted at the boundary between different materials, though; to model refraction, instead use the Material Discontinuity feature.
Diffuse scattering	Select to reflect rays at a wall according to Lambert's cosine law. That is, the probability of a reflected ray propagating in a given direction within a solid angle $d\omega$ is given by $\cos(\theta)d\omega$ where θ is the angle between the direction of the ray and the wall normal.
Isotropic scattering	Select to reflect rays at a wall isotropically in random directions.
Mixed diffuse and specular reflection	Select to reflect rays at a wall either specularly or according to Lambert's cosine law, based on a user-defined probability.
General reflection	Select to allow an arbitrary direction vector to be specified after a ray makes contact with the wall. The ray direction vector components can be functions of the incident ray direction, phase, or any other quantity.

GENERAL REFLECTION SETTINGS

This section is available when **General reflection** (see Table 8-2) is selected as the **Wall condition**.

Enter values for the **Reflected ray direction vector** $\mathbf{L}_p$ (dimensionless) either in Cartesian coordinates (x, y, z) (the default) or select the **Specify tangential and normal direction vector components** checkbox to enter coordinates in the tangent-normal coordinate system (t_1, t_2, n). In this case, the normal direction is selected so that an incident ray is reflected back into the domain it previously occupied if the specified normal direction vector component is positive. The tangential directions are oriented so that they form a right-handed coordinate system, together with the normal direction.

PRIMARY RAY CONDITION

Use this section to apply two different wall conditions on the same set of boundaries based on a probability or logical expression.

Select a **Primary ray condition**: **None** (the default), **Probability**, or **Expression**. When the default, **None**, is kept, it means that the **Wall condition** is always respected by the incident rays.

Probability

If **Probability** is selected, the **Wall condition** is applied with a certain probability. Enter a value for the **Probability**, γ (dimensionless). If the **Wall condition** is not used, the ray instead behaves according to the Otherwise setting.

The value of γ should always be between 0 and 1.

For example, if the **Wall condition** is set to:

- **Freeze** and γ is set to 0.1, then for every 10 rays that strike the wall, on average one freezes and the remaining 9 rays behave according to the Otherwise setting.
- **Stick** and γ is set to 0.5 then on average half of the rays stick to the wall and the other half behave according to the Otherwise setting.

Expression

If **Expression** is selected, the **Evaluation expression** e (dimensionless) is evaluated whenever a ray strikes the wall. The default expression is 1. If the Evaluation expression is nonzero, the ray behaves according to the **Wall condition**, otherwise the ray behaves according to the Otherwise setting.

Otherwise

The options available for the **Otherwise** setting are the same as for the Wall Condition, except that **General reflection** and **Mixed diffuse and specular reflection** are not available. The **Otherwise** setting can be used to make rays interact with a wall differently with a certain probability or when a certain condition is satisfied. For example, to model reflection at a partially specular surface in which 50% of the ray intensity is absorbed and 30% is reflected specularly:

- Select **Mixed diffuse and specular reflection** as the **Wall condition**.
- Set the **Probability of specular reflection** to `0.3/(1-0.5)`.
- Set the **Primary ray condition** to **Probability**.
- Set the **Probability** to `1-0.5`.
- Select **Freeze** as the **Otherwise** option.

PHASE SHIFT

This section is available when:

- The ray intensity is solved for in the model.
- The **Compute phase** checkbox is selected under the physics interface **Advanced settings** section.
- The **Wall condition** is set to **Specular Reflection**, **Diffuse scattering**, **Isotropic scattering**, **General reflection**, or **Mixed diffuse and specular reflection**.

Select the **Apply manual phase shift** checkbox to specify the phase shift applied to reflected rays. By default this checkbox is cleared, causing the phase shift be derived from the reflection coefficient.

If the **Apply manual phase shift** checkbox is selected, enter a Phase shift $\Delta\Psi$ (SI unit: rad). The default is 0.

If the **Wall condition** is set to **Mixed diffuse and specular reflection**, this section is instead called **Phase shifts** and the phase shifts for the diffusely and specularly reflected rays are specified separately. All of the text fields are given subscripts s or d for specularly and diffusely reflected rays, respectively.

REFLECTION COEFFICIENTS MODEL

This section is available when the ray intensity or power is solved for in the model, and the **Wall condition** is set to any option that can reflect rays, including the following:

- **Specular Reflection**,

- **Diffuse scattering**,
- **Isotropic scattering**,
- **General reflection**, or
- **Mixed diffuse and specular reflection**.

Select one of the following from the **Compute reflected intensity using** list: **Absorption coefficients**, **Absorber, specified impedance**, **Reflection coefficients**, **Fluid-fluid interface**, **Fluid-solid interface**, or **Layered fluid half space**.

- For **Absorption coefficients** enter the **Absorption coefficient** α (dimensionless). The default is 0. The intensity or power of the reflected ray will be proportional to $1 - \alpha$.
- For **Absorber, specified impedance** enter the **Characteristic impedance of absorber** Z_1 (SI unit: Pa·s/m). The default value is 1 GPa·s/m, corresponding to a hard wall.

- For **Reflection coefficients** enter the **Reflection coefficient** R (dimensionless). The default value is 1. The intensity or power of the reflected ray will be proportional to R^2.
- For **Fluid-fluid interface** enter the following:
  - **Speed of sound, adjacent fluid** c_1 (SI unit: m/s). The default is 1500 m/s.
  - **Density, adjacent fluid** ρ_1 (SI unit: kg/m^3). The default is 1000 kg/m^3.
  - **Attenuation coefficient, adjacent fluid** α_1 (SI unit: 1/m). The default is 0.
- For **Fluid-solid interface** enter the following:
  - **Compressional speed of sound, adjacent solid** $c_{p,1}$ (SI unit: m/s). The default is 1500 m/s.
  - **Shear speed of sound, adjacent solid** $c_{s,1}$ (SI unit: m/s). The default is 1500 m/s.
  - **Density, adjacent solid** ρ_1 (SI unit: kg/m^3). The default is 1000 kg/m^3.
  - **Compressional attenuation coefficient, adjacent solid** $\alpha_{p,1}$ (SI unit: 1/m). The default is 0.
  - **Shear attenuation coefficient, adjacent solid** $\alpha_{s,1}$ (SI unit: 1/m). The default is 0.
- For **Layered fluid half-space** enter the following:
  - **Thickness, adjacent fluid layer** h (SI unit: m). The default is 1 μm.
  - **Speed of sound, adjacent fluid layer** c_1 (SI unit: m/s). The default is 1500 m/s.
  - **Density, adjacent fluid layer** ρ_1 (SI unit: kg/m^3). The default is 1000 kg/m^3.
  - **Attenuation coefficient, adjacent fluid layer** α_1 (SI unit: 1/m). The default is 0.
  - **Speed of sound, adjacent fluid domain** c_2 (SI unit: m/s). The default is 1500 m/s.
  - **Density, adjacent fluid domain** ρ_2 (SI unit: kg/m^3). The default is 1000 kg/m^3.
  - **Attenuation coefficient, adjacent fluid domain** α_2 (SI unit: 1/m). The default is 0.

If the **Wall condition** is set to **Mixed diffuse and specular reflection**, this section is instead called **Reflection Coefficients Model**, only the options **Absorption coefficients**, **Absorber, specified impedance**, and **Reflection coefficients** exist, and the data for the diffusely and

specularly reflected rays are specified separately. All of the text fields are given subscripts s and d for specularly and diffusely reflected rays, respectively.

To create detailed models the wall properties, for example, the surface normal impedance or the absorption coefficient, must depend on the properties of the incident ray. Doing this is possible as all wall properties may depend on:

- The frequency of the ray `rac.f`
- The direction of the ray, the acute angle of incidence `rac.phii`
- Even the intensity of the ray `rac.I`

SURFACE ROUGHNESS

This section is available when

- the ray intensity or power is solved for in the model, and
- the **Wall condition** is set to **Specular Reflection**, **General reflection**, or **Mixed diffuse and specular reflection**. For **Mixed diffuse and specular reflection** the roughness only applies to the specularly reflected rays.

Select an option from the **Surface roughness** model list: **None** (the default) or **Rayleigh roughness**. If **Rayleigh roughness** is selected, enter the **RMS roughness** σ (SI unit: m). The default is 0. Use this setting to reduce the intensity of rays that are reflected by a rough wall.

NEW VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

When a ray crosses or touches a boundary, the values of the user-defined auxiliary dependent variables can be changed. The value can be a function of any combinations of ray variables and variables defined on the boundary.

Select the **Assign new value to auxiliary variable** checkbox or boxes based on the number of auxiliary variables in the model. Then enter the new value or expression in the field. For example, if there is an auxiliary variable, psi, then enter a value for psi_{new} in the field. So, to increment the value of psi by 1 when a ray touches or crosses a boundary, enter `psi+1` in the text field for psi_{new}.

ADVANCED SETTINGS

If the **Primary ray condition** is set to **Probability**, or if the **Diffuse scattering**, **Isotropic scattering**, or **Mixed diffuse and specular reflection** wall condition is used, then the **Wall** feature generates random numbers.

If, in addition, the **Arguments for random number generation** setting is set to **User defined** in the physics interface **Advanced Settings** section, the **Advanced Settings** section is available.

Enter the **Additional input argument to random number generator** i (dimensionless). The default value is `0`.

Accumulator (Boundary)

The **Accumulator** subnode is available from the context menu (right-click the Wall or Material Discontinuity parent node) or from the **Physics** toolbar, **Attributes** menu. Each **Accumulator** subnode defines a variable, called the accumulated variable, on each boundary element in the selection of the parent node. The accumulated variables are incremented when rays interact with the boundaries where they are defined. The accumulated variable may be influenced by all rays or may be restricted to the rays released by specific release feature.

ACCUMULATOR SETTINGS

Select an option from the **Accumulator type** list: **Density** (default) or **Count**.

- For **Density** the accumulated variable is divided by the surface area of the boundary element where it is defined.
- For **Count** the accumulated variable is the sum of the source terms of all rays that hit the boundary element, and is unaffected by the boundary element size.

Select an option from the **Accumulate over** list: **Ray-wall interactions** (the default) or **Rays in boundary elements**.

- For **Ray-wall interactions** the accumulated variable is affected by all rays that hit the boundary element.
- For **Rays in boundary elements** the accumulated variable is only affected by rays that freeze or stick to the boundary element.

Enter the **Accumulated variable name**. The default is `rpb`. The accumulated variable is defined as `<scope>.<name>`, where `<scope>` includes the name of the physics

interface node, parent boundary condition, and the **Accumulator** node, and `<name>` is the accumulated variable name.

For example, if the **Accumulator** subnode is added to a Wall node in an instance of the Ray Acoustics interface using the default variable name `rpb`, the accumulated variable name might be `rac.wall1.bacc1.rpb`.

Enter a **Source** R. The unit of the source term depends on the settings in the **Units** section. Whenever a ray collides with a boundary element in the selection of the parent node, the accumulated variable in that element is incremented by the source term. If the **Accumulator type** is set to **Density**, the source term is divided by the area of the boundary element (in 3D) or the length of the boundary element (in 2D).

For example, if the source is `1` and **Count** is selected from the **Accumulator type** list, a variable is created to store the number of times each boundary element is hit by a ray.

Select an option from the **Release feature** list. If **All** (the default) is selected, the accumulated variable collects information about all rays in the selected domains or boundaries, regardless of how they were released. Alternatively, select a ray release feature from the list, and then only the rays produced by that release feature are included.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

SMOOTHING

The accumulated variables are computed using discontinuous shape functions. Select the **Compute smoothed accumulated variable** checkbox to compute a smoothed accumulated variable by computing the average value of the variable within a sphere of a user-defined radius. Then enter a **Smoothing radius** r (SI unit: m). The default is 1 m.

Accumulator Theory: Boundaries

Sound Pressure Level Calculation

When the ray power is solved for, the **Sound Pressure Level Calculation** subnode is available from the context menu (right-click the Wall, parent node) or from the **Physics**

toolbar, **Attributes** menu. The **Sound Pressure Level Calculation** subnode computes the sound pressure level resulting from the interaction of the incident and reflected acoustic waves at the wall.

SMOOTHING

Select the **Compute smoothed accumulated variable** checkbox to enter a **Smoothing radius** r (SI unit: m) The smoothing radius works like the corresponding setting for the Accumulator (Boundary) node.

When the **Compute smoothed accumulated variable** is selected, smoothed variables are defined for the intensity and sound pressure level. They can be plotted in postprocessing and are, for example, named `rac.wall1.spl1.Iw_sm` and `rac.wall1.spl1.Lp_sm`.

Sound Pressure Level Calculation Theory

Axial Symmetry

The **Axial Symmetry** node is automatically added to 2D axisymmetric models. The options available for the feature are the same as for Wall, except as follows. The **Boundary Selection** is locked and is only applicable to edges on the symmetry axis. The only wall conditions available are **Specular reflection** (the default), **Freeze**, **Stick**, and **Disappear**. In addition, the **Primary Ray Condition** section is not included.

The **Axial Symmetry** condition only applies to straight boundaries that are both on the axis of symmetry and adjacent to a domain in the selection of the physics interface. If rays are propagating in the void region outside of the selected domains, it is possible for the rays to enter the region where $r < 0$, even though no geometry is defined there.

To ensure that rays cannot propagate into the region where $r < 0$, draw line segments on the parts of the symmetry axis where no geometric entities exist, then add the **Wall** node to these edges. Keep in mind that rays can only be subjected to boundary conditions if the corresponding boundaries are meshed.

Receiver

Use the **Receiver** feature to define the boundaries of a receiver sphere in a Ray Acoustics model. The receiver collects information (arrival time and power) about the intersecting rays during the simulation. This information is then used for computing the impulse response in the Results analysis. Details are found in the Impulse Response Plot and Receiver Dataset section. The **Receiver** feature is available when **Compute power**, **Compute intensity and power**, or **Compute intensity and power in graded media** is selected in Intensity Computation.

Create parameters under **Global Definitions>Parameters** for the receiver radius and number of released rays. These two quantities are linked by the expected error in a given time interval Δt of the impulse response, see Ref. 6 for details. With the room volume V, the receiver radius R, and for an expected error of 1 dB, the number of released rays N needed is

$$N = 4.34^2 \frac{V}{\pi R^2 c \Delta t}$$

The receiver radius can be set to give a certain spatial resolution (like the width of a seat in a concert hall), and a standard time interval is $\Delta t = 0.01$ s. For small rooms or car cabins, it can be useful to set a receiver radius that represents the actual size of a microphone or slightly larger. The value obtained from this calculation should be rounded up to ensure an integer number of released rays. A higher value can also be chosen and will result in increased resolution.

The receiver should be defined as a sphere with **Object Type** selected to be **Surface** in the Geometry. The receiver should also be meshed adequately. Typically a mesh size of $R/3$, where R is the receiver radius is a good choice. The receiver is transparent to rays and is only used for detection. The receiver feature is only available when **Compute power** or **Compute intensity and power** is selected in the Intensity Computation section. **Count reflections** should also be enabled in the Additional Variables for a correct impulse response computation. Both are selected by default when adding the Ray Acoustics physics interface.

COORDINATE SYSTEM SELECTION

By default, **Global coordinate system** is selected from the **Coordinate system** list. Other user defined coordinate systems can also be selected from the list. However, boundary coordinate systems are not included in the list.

The coordinate system is used to define local built-in variables for the azimuthal and polar angles. For a physics interface with the default name `rac` and a **Receiver** node with the default name `rec1`, these variables are called `rac.rec1.phi` (azimuthal angle) and `rac.rec1.theta` (polar angle). These angles are defined with respect to the selected coordinate system; for example, if the **Global coordinate system** is used, then `rac.rec1.theta` is the angle between the ray direction vector and the positive z-axis, while `rac.rec1.phi` is the angle of the projected ray direction vector in the xy-plane, measured counterclockwise from the positive x-axis. A default expression showing the variables is set up for the **Spatial directivity** input for a dB based expression.

RECEIVER

Select the **Receiver radius** as **Automatic** (the default) or **User defined**. The automatic option will work well for a spherical receiver geometry defined in COMSOL. For an imported CAD geometry it can be useful to enter the radius r_{rec} manually. The radius is used for computing the receiver volume necessary for computing the detected signal intensity.

Select the **Directivity type** as **Omnidirectional** (the default), **User defined (dB)**, or **User defined (linear)**. For the two user defined options, enter an expression for the Spatial directivity of the receiver. Built-in polar angle variables, that are defined in relation to the **Coordinate system**, exist to simplify setting up the receiver directivity. The **Spatial directivity** $D(\varphi,\theta)$ can be an analytical expression or based on an interpolation function.

Release from Boundary

Use the **Release from Boundary** node to determine how to release rays on an interior or exterior boundary in a specific direction.

The Nonlocal Accumulator subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

See Release for information on the following sections: **Release Times**, **Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

COORDINATE SYSTEM SELECTION

It is possible to specify the initial ray direction in terms of the global coordinates or in another coordinate system defined for the model Component. Select an option from the **Coordinate system** list. By default **Global coordinate system** is selected. If other coordinate systems are defined, they can also be selected from the list. When specifying the initial ray direction (see the **Ray Direction Vector** section), direction components can be specified using the basis vectors of whichever coordinate system has been selected from the list.

When a coordinate system other than **Global coordinate system** is selected from the **Coordinate system** list, arrows will appear in the Graphics window to indicate the orientation of the basis vectors of the coordinate system on the selected boundaries.

INITIAL POSITION

Select an **Initial position**: **Uniform distribution** (the default for 2D components) **Projected plane grid** (the default for 3D components), **Mesh based**, or **Density**. Mesh based and Density have the same settings as described for the Release node.

	For 2D components, if **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The union of the selected boundaries is divided into N segments of approximately equal length, and a ray is placed in the middle of each segment.

	For 3D components, if **Projected plane grid** is selected, enter the **Number of rays per release** N (dimensionless). The rays are distributed on a plane grid in planes that are approximately tangential to the selected boundaries (for a plane boundary, you get a uniform distribution).

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Expression** (the default), **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **Expression**, a single ray is released in the specified direction. Enter coordinates for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless) based on space dimension. Select the **Specify tangential and normal vector components** checkbox to specify the initial direction using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).

- For **Hemispherical**, a number of rays are released at each point, sampled from a hemispherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension. Select the **Specify tangential and normal vector components** checkbox to specify the hemisphere axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).
- For **Conical**, a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Cone axis r** based on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is $\pi/3$ radians. Select the **Specify tangential and normal vector components** checkbox to specify the cone axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n).
- The **Lambertian** option is only available in 3D. A number of rays are released at each point, sampled from a hemisphere in wave vector space with probability density based on the cosine law. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Hemisphere axis r** based on space dimension. Select the **Specify tangential and normal vector components** checkbox to specify the hemisphere axis using a coordinate system based on the directions tangential and normal to the surface (t_1, t_2, n)

When the **Specify tangential and normal vector components** checkbox is selected, arrows indicating the normal direction on the selected boundaries will appear in the Graphics window.

Note that the normal direction may be opposite the built-in variable for the boundary normal (for example, `nx`, `ny`, and `nz`) to ensure that a positive value causes rays to be released into the simulation domain. This often occurs when the release is applied to exterior boundaries.

When the normal direction used by the **Release from Boundary** feature is opposite the normal vector defined by the geometry, the tangential directions are similarly inverted to ensure that the boundary coordinate system is right handed.

If **Conical** is selected in a 3D model, select an option from the **Conical distribution** list:

- **Uniform density** (the default): rays are released with polar angles from 0 to the specified cone angle. The rays are distributed in wave vector space so that each ray subtends approximately the same solid angle.
- **Specify polar and azimuthal distributions**: specify the **Number of polar angles** N_θ (dimensionless) and the **Number of azimuthal** angles N_φ (dimensionless). Rays are released at uniformly distributed polar angles from 0 to the specified cone angle. A single axial ray ($\theta = 0$) is also released. For each value of the polar angle, rays are released at uniformly distributed azimuthal angles from 0 to 2π. Unlike other options for specifying the conical distribution, it is not necessary to directly specify the **Number of rays in wave vector space** N_w (dimensionless), which is instead derived from the relation $N_w = N_\theta \times N_\varphi + 1$.
- **Hexapolar**: specify the **Number of polar angles** N_θ (dimensionless). In this distribution, for each release point, one ray will be released along the cone axis. Six rays are released at an angle α/N_θ from the cone axis, then 12 rays at an angle of $2\alpha/N_\theta$, and so on. The total number of ray directions in the distribution is $N_w = 3N_\theta(N_\theta + 1) + 1$.
- **Flat**: rays are released in a flat fan shape within the specified angle.
- **Marginal rays only**: the rays are all released at an angle α with respect to the cone axis. The rays are released at uniformly distributed azimuthal angles from 0 to 2π.
- **Marginal and axial rays only**: the rays are all released at an angle α with respect to the cone axis, except for one ray which is released along the cone axis. The marginal rays are released at uniformly distributed azimuthal angles from 0 to 2π.

The available **Conical distribution** options are illustrated in Figure 8-1.

In 3D for the **Conical distribution** you can also let the **Transverse direction** be **Automatic** (the default) or **User defined**. For **User defined** enter the components of $\mathbf{e}_t$. This controls, for example, the orientation of the ray fan when **Flat** is selected.

For **Hemispherical**; **Conical** with **Flat** or **Uniform density** distribution; and **Lambertian**, select an option from the **Sampling from Distribution** list: **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray direction vectors are computed using the same algorithm, which seeks to distribute the rays as evenly as possible in wave vector space, whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudorandom numbers.

INITIAL RADII OF CURVATURE

This section is available when the ray intensity is solved for in the model and **Expression** is selected as the **Ray direction vector**. Select a **Wavefront shape**. In 3D the available options are **From surface curvature** (the default), **Plane wave**, **Spherical wave**, and **Ellipsoid**. In 2D the available options are **Plane wave** (the default) and **Cylindrical wave**.

- For a **Spherical wave** or **Cylindrical wave**, enter the **Initial radius of curvature** r_0 (SI unit: m).
- For an **Ellipsoid** (3D only), enter the **Initial radius of curvature, 1** $r_{1,0}$ (SI unit: m) and the **Initial radius of curvature, 2** $r_{2,0}$ (SI unit: m). Also enter the **Initial principal curvature direction, 1** $\mathbf{e}_{1,0}$ (dimensionless).

For spherical and cylindrical waves the **Initial radius of curvature** must be nonzero. To release a ray such that the initial wavefront radius of curvature is zero, instead select a different option such as **Conical** from the **Ray direction vector** list.

Principal Radii of Curvature

Nonlocal Accumulator

Use the **Nonlocal Accumulator** subnode to communicate information from the current position of a ray to the position from which it was released.

The subnode is available from the context menu (right-click the Release from Boundary parent node) or from the **Physics** toolbar, **Attributes** menu.

Each **Nonlocal accumulator** subnode defines a variable, called the accumulated variable, that is computed using variables defined on rays released by the parent **Release from Boundary** node or on domains and boundaries encountered by such rays.

ACCUMULATOR SETTINGS

Select an **Accumulator type**: **Density** (default) or **Count**.

- For **Density**, the accumulated variable is divided by the volume of the mesh element where it is defined.
- For **Count**, the accumulated variable is unaffected by the element size.

Select an option from the Accumulate over list: **Elements** (default) or **Elements and time**.

- For **Elements**, the accumulated variable is proportional to the instantaneous value of the **Source** term R for all applicable rays.
- For **Elements and time**, the time derivative of the accumulated variable is proportional to the instantaneous value of the **Source** term R for all applicable rays, and thus the accumulated variable considers the time history of rays in the modeling domain instead of just their current values.

Enter the **Accumulated variable name**. The default is `rpi`.

Enter a **Source** R. The unit of the source depends on the settings in the **Units** section. The source term is used to calculate the accumulated variable in a manner specified by the **Accumulate over** and **Accumulator type** settings.

Select a **Source geometric entity level**: **Domains**, **Boundaries**, or **Domains and boundaries**.

- If **Domains** is selected, rays only contribute to the accumulated variable on their releasing surface if they are still active; that is, they are still propagating through a domain.
- If **Boundaries** is selected, the rays only contribute to the accumulated variable if they have become stuck or frozen to a boundary somewhere in the model.
- If **Domains and boundaries** is selected, all of the active, stuck, and frozen rays released by a feature can contribute to the accumulated variable.

UNITS

Select a **Dependent variable quantity** from the list; the default is **Dimensionless [1]**. To enter a unit, select **None** from the list and in the **Unit** field enter a value, for example, `K`, `m/s`, or `mol/m^3`.

SMOOTHING

Select the **Compute smoothed accumulated variable** checkbox to enter a **Smoothing radius** r (SI unit: m). The default is 0.1 m.

Release from Pressure Field

Use the **Release from Pressure Field** feature to create realistic sources extracted from a wave-based (near-field) simulation using The Pressure Acoustics, Frequency Domain Interface (a submodel). Here, near-field is in relation to the source, meaning that the classical point source approximation of ray tracing is not necessary. Note that setting

up classical point sources is covered by the Release from Exterior Field Calculation and the Source with Directivity features. An example, a near-field source could be a loudspeaker placed in a setup (like in the dashboard of a car) that causes local reflections and diffractions. These are wave phenomena that are not captured by the ray tracing method. The **Release from Pressure Field** is assigned to the radiating boundaries of the pressure acoustics submodel.

The **Release from Pressure Field** feature is available when **Compute power**, **Compute intensity**, or **Compute intensity and power** is selected in Intensity Computation.

When the **Release from Pressure Field** is added as a boundary source, rays are released in the direction of the acoustic intensity vector and the ray power distribution is weighted. The intensity vector is taken from the computed pressure acoustics submodel.

Because this source approach combines wave based and ray tracing methods, some restrictions exist on the use of the **Release from Pressure Field**. These are:

- When rays are released using the **Release from Pressure Field**, they are all released at the same time. Because of this, it should be assumed that the sound emanating from the source reaches and leaves every part of the release boundary at the same time. Hence the release boundary cannot be placed arbitrarily.
- Because of the restriction mentioned in the previous point, internal reflections in the source domain cannot easily be taken into account. Sound can indeed travel different distances depending on the reflection path taken, which would lead to separate time events at the release boundary and in the impulse response.
- Finally, the time delay (time of flight from the source to the release boundary) is not included in the impulse response computed in ray tracing. The release time is assumed to be 0 at the release boundary.

SOLUTION FIELD

Select the **Physics interface** that defines the full-wave near-field solution. From this list choose the Pressure Acoustics, Frequency Domain interface that is used to solve the acoustic field inside the source region. In the pressure acoustics model, the equivalent release boundary should be an open radiation condition.

RELEASE FROM PRESSURE FIELD

Enter the **Number of rays per release** for the ray acoustics simulation.

The **Initial Value of Auxiliary Dependent Variables** is documented for the Release feature.

Ray Detector

Use the **Ray Detector** feature to compute information about rays that are located in a set of selected domains or on a set of selected boundaries. The detector may detect all rays or only the rays released by a specified release feature. Computed variables are the number of rays transmitted, the transmission probability, and a logical expression which can be used to filter the rendered rays during results processing.

The **Ray Detector** feature is also capable of computing the total transmitted ray power. If the ray power is solved for, then a global variable for the total power transmitted to the detector is available.

RAY DETECTOR

Select an option from the **Release feature** list: **All** (the default), or any of the features which are capable of releasing rays. When **All** is selected, the variables computed are summed over all release features in the model.

Release from Symmetry Axis

Use the **Release from Symmetry Axis** node to release rays from specified edges along the axis of symmetry in a 2D axisymmetric geometry. It functions like the Release from Boundary node, except that it can only be applied to the axis of symmetry and that the options in the **Ray Direction Vector** section are slightly different.

See Release for information on the following sections: **Release Times, Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL POSITION

Select an **Initial position**: **Uniform distribution** (the default), **Mesh based**, or **Density**. **Mesh based** and **Density** have the same settings as described for the Release node.

If **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The default is 100. The union of the selected boundaries is divided into N segments of approximately equal length, and a ray is placed in the middle of each segment.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Expression** (the default), **Spherical**, or **Conical**.

- For **Expression**, a single ray is released in the specified direction. Enter the radial and axial components of the **Ray direction vector** $\mathbf{L}_0$ (dimensionless).
- For **Spherical**, a number of rays are released at each point, from the negative z direction to the positive z direction in the half plane with $r > 0$. These rays are sampled from an anisotropic hemispherical distribution in wave vector space, so that rays are more heavily weighted toward the r direction rather than the $\pm z$ directions. In 3D, this would correspond to a spherical distribution of rays such that each ray subtends approximately the same solid angle in wave vector space.

  Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50.
- For **Conical**, a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then enter coordinates for the **Cone axis r** based on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is $\pi/3$ radians. Unlike the **Spherical** distribution, the rays in the **Conical** distribution each subtend the same angle in 2D, not the same solid angle in 3D.

For **Spherical** and **Conical**, select an option from the **Sampling from Distribution** list: **Deterministic** (the default) or **Random**. If **Deterministic** is selected, the initial ray direction vectors are computed using the same algorithm, which seeks to distribute the rays as evenly as possible in wave vector space, whenever the study is run. If **Random** is selected, the initial direction of each ray is sampled from a probability distribution in wave vector space using pseudorandom numbers.

Ray Continuity

Use the **Ray Continuity** node to specify that rays should cross a pair boundary as if it were invisible. Pair boundaries appear when the geometry sequence ends in **Form Assembly** instead of **Form Union**. Such boundaries require special handling because the mesh elements on either side of the pair boundary are not required to match up exactly.

PAIR SELECTION

Select one or more identity pairs to allow rays to cross between the source and destination boundaries of these pairs. Such identity pairs are typically created

automatically on interior boundaries when the geometry sequence ends in a **Form Assembly** node instead of a **Form Union** node.

The **Ray Continuity** node does not cause the ray position components to change discontinuously; the source and destination boundaries for the identity pairs must be overlapping.

If the geometry sequence ends in a **Form Union** node, the **Ray Continuity** node usually is not necessary because rays can freely cross interior boundaries where no boundary condition has been applied.

Release from Edge

Use the **Release from Edge** node to release rays from positions along a set of edges in a 3D geometry.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL POSITION

Select an **Initial position**: **Mesh based**, **Uniform distribution**, or **Density** (the default). Mesh based and Density have the same settings as described for the Release node. If **Uniform distribution** is selected, enter the **Number of rays per release** N (dimensionless). The union of the selected edges is divided into N segments of approximately equal length, and a ray is released from the middle of each segment.

Release from Point

Use the **Release from Point** node to release rays from a set of points in a geometry. By default, one ray is released at every selected point, although it is possible to release multiple rays at each point if the initial direction or an auxiliary dependent variable is sampled from a distribution.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

Release from Point on Axis

The **Release from Point on Axis** feature is similar to the Release from Point feature, except that it can only release rays from points along the axis of symmetry in a 2D axisymmetric geometry.

See Release from Symmetry Axis for information on the **Ray Direction Vector** section.

For all other sections, see Release.

Ray Properties

The **Ray Properties** node is added by default. Only one instance of this node can exist for each physics interface. The **Ray Properties** node defines the equations for ray propagation and also includes the input for the ray frequency.

RAY PROPERTIES

This section is shown when the **Allow frequency distributions at release features** checkbox is cleared in the physics interface **Ray Release and Propagation** section. Otherwise the ray frequency is controlled separately by each ray release feature.

Select the **Ray frequency** as **User defined** (the default) or as announced by another feature. If the Release from Pressure Field or the Release from Exterior Field Calculation features are present in the model, the **Ray frequency** is automatically locked to the frequency of the source. The drop down becomes locked (grayed out).

For the **User defined** option enter a **Ray frequency** f (SI unit: Hz) here. The default is 1000 Hz. In typical room acoustics simulations the frequency will correspond to the center frequency of the analyzed frequency band. See Impulse Response Plot and Receiver Dataset.

Auxiliary Dependent Variable

Use the **Auxiliary Dependent Variable** node to solve additional first-order ordinary differential equations for each ray released.

AUXILIARY DEPENDENT VARIABLE

Enter a **Field variable name**. The default is `rp` and can be changed to anything provided it does not conflict with the name of the variables for the position or momentum degrees of freedom. The name should not conflict with other auxiliary dependent variables.

Enter a **Source** R. The unit of the source depends on the unit of the auxiliary dependent variable and the option selected from the **Integrate** list described below.

Select a way to **Integrate** the equation you have defined: **With respect to time** or **Along ray trajectory**. If you are interested in the residence time of a group of rays in a given system then you can set the **Source** to 1 and set **Integrate** to **With respect to time**. To compute the length of the ray trajectory you can set the **Source** to 1 and set **Integrate** to **Along ray trajectories**.

UNITS

These settings are the same as for Accumulator (Domain).

Release from Grid

Use the **Release from Grid** node to release rays from a user-defined grid of points.

See Release for information on the following sections: **Release Times, Ray Direction Vector**, **Initial Ray Frequency**, **Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, **Total Source Power**, and **Initial Value of Auxiliary Dependent Variables**.

INITIAL COORDINATES

Select an option from the **Grid type** list: **All combinations** (the default), **Specified combinations**, **Cylindrical** (3D only), or **Hexapolar** (3D only).

Linear, Rectangular, and Arbitrary Grids

For **All combinations** and **Specified combinations** enter **Initial coordinates** based on space dimension ($q_{x,0}$, $q_{y,0}$, and $q_{z,0}$ for 3D components) for the ray positions or click the **Range** button ( ) to select and define a range of specific coordinates.

If **Specified combinations** is selected, the number of initial coordinates entered for each space dimension must be equal, and the total number of rays released is equal to the length of one of the lists of initial coordinates. If **All combinations** is selected, the total number of rays released is equal to the product of the lengths of each list of initial coordinates.

For example, suppose a 2D component includes a **Release from Grid** node with the following initial coordinates:

- $q_{x,0}$ = range(0,1,3)
- $q_{y,0}$ = range(2,2,8)

If **All combinations** is selected, a total of 16 rays will be released, including every possible combination of the initial x- and y-coordinates. If **Specified combinations** is selected, 4 rays will be released with initial positions (0,2), (1,4), (2,6), and (3,8).

Cylindrical Grids

For **Cylindrical**, enter coordinates for the **Center location** $\mathbf{q}_c$ (SI unit: m). By default, the distribution is centered at the origin. Then enter the components of the **Cylinder axis direction** $\mathbf{r}_c$ (dimensionless). The rays will be released at specified radial distances and angles in the plane containing the point $\mathbf{q}_c$ and orthogonal to the direction $\mathbf{r}_c$.

Select an option from the **Radial distribution** list: **Uniform radius intervals** (the default), **Uniform number density**, or **User defined**.

For **Uniform radius intervals** or **Uniform number density**, enter a value or expression for the **Radius** R_c (SI unit: m). The default is 1 m. Then enter a positive integer for the **Number of radial positions** N_c (dimensionless). The default is 5.

For **User defined**, enter a list of **Radial coordinates** q_r (SI unit: m) directly. An arbitrary number of radial coordinates can be entered in the list. The default is 1 m.

The effect of the **Radial distribution** setting on the resulting ray placement is illustrated in Figure 8-2. For **Uniform radius intervals**, the distances between the concentric rings of rays are all equal, but the number density of rays is greater at the center of the distribution than at the outer edge. The option **Uniform number density** corrects this imbalance by defining nonuniform increments in the radial position between the concentric rings.

Enter a positive integer for the **Number of angles** N_ϕ (dimensionless). The default is 10.

The total number of rays released (before accounting for distributions of ray direction or auxiliary dependent variables at each release point) is $N_c \times N_\phi + 1$ because a single ray is also released at the center of the distribution.

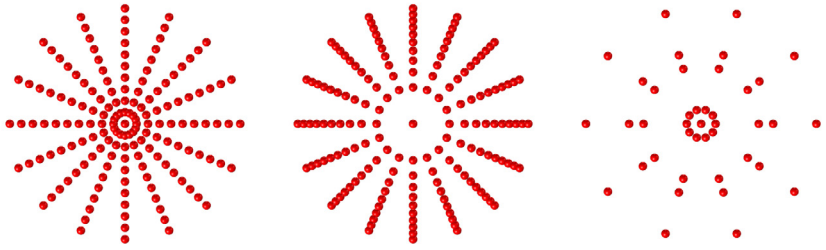


Figure 8-2: Comparison of the cylindrical release in which rays are positioned at uniform radius intervals (left), with uniform number density (center), or at user-defined radii (right).

Hexapolar Grids

For **Hexapolar**, enter coordinates for the **Center location** $\mathbf{q}_c$ (SI unit: m). By default, the distribution is centered at the origin. Then enter the components of the **Cylinder axis direction** $\mathbf{r}_c$ (dimensionless). The rays will be released at specified radial distances and angles in the plane containing the point $\mathbf{q}_c$ and orthogonal to the direction $\mathbf{r}_c$.

Enter a value or expression for the **Radius** R_c (SI unit: m). The default is 1 m. Then enter a positive integer for the **Number of radial positions** N_c (dimensionless). The default is 5.

Unlike the **Cylindrical** grid types described in the previous section, each concentric ring of the **Hexapolar** grid releases a different number of rays, as shown in Figure 8-3. One ray is always released at the center. The first ring of rays surrounding the center has 6 rays arranged in a regular hexagon. Each ring of rays beyond the first has 6 more rays than the ring preceding it, with the grid points arranged in a regular polygon. The radius increments between consecutive rings are uniform.

The total number of grid points generated is

$$N_{\text{grid}} = 1 + 3N_c(N_c + 1)$$

Thus the relationship between the number of radius intervals and the total number of grid points is quadratic.

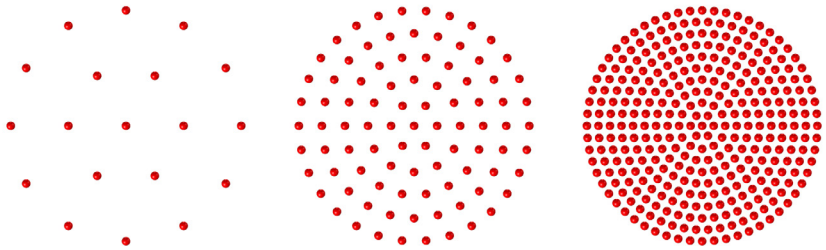


Figure 8-3: Comparison of hexapolar grids with 2 rings (left), 5 rings (middle), and 10 rings (right).

Previewing Grid Points

In the **Initial Coordinates** section, you can click the **Preview Initial Coordinates** and **Preview Initial Extents** buttons to visualize the ray release positions. Clicking **Preview Initial Coordinates** will cause a point to appear in the Graphics window for every release position. Clicking **Preview Initial Extents** will cause a bounding box to appear, indicating the spatial extents of the released rays. Examples are shown in Figure 8-4 and Figure 8-5.

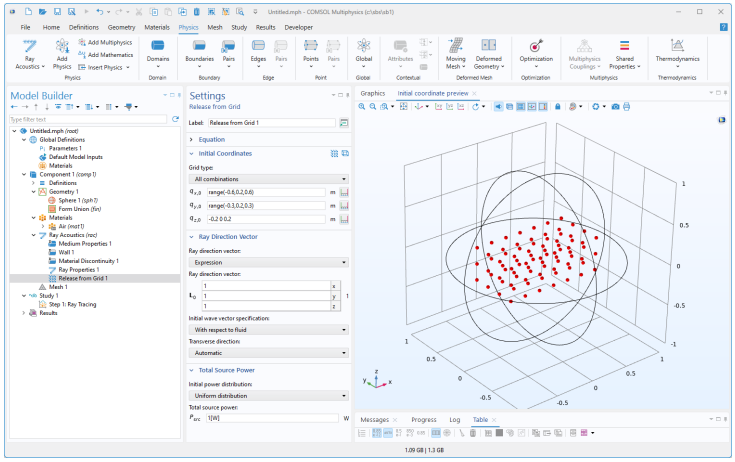


Figure 8-4: Graphics window after clicking the Preview Initial Coordinates button.

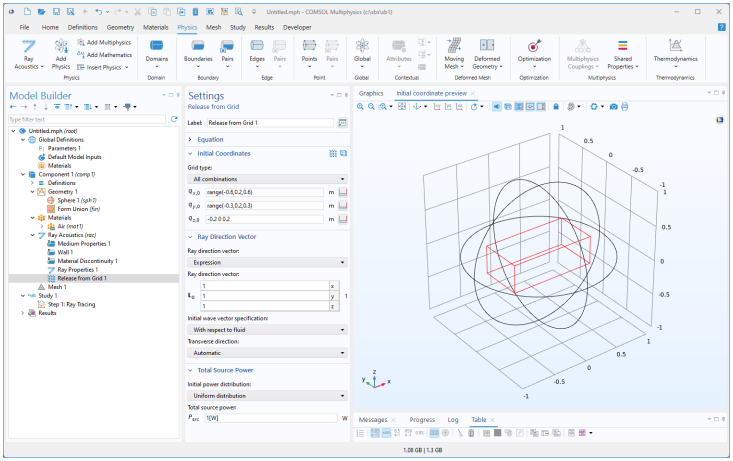


Figure 8-5: Graphics window after clicking the Preview Initial Extents button.

Release from Grid on Axis

The **Release from Grid on Axis** feature is similar to the Release from Grid feature, except that it can only release rays from a grid of points along the axis of symmetry in a 2D axisymmetric geometry; that is, $q_{r,0} = 0$ for all released rays.

See Release from Symmetry Axis for information on the **Ray Direction Vector** section.

For all other sections, see Release.

Release from Data File

Use a **Release from Data File** node to specify the initial ray position, direction, and values of auxiliary dependent variables using data from a text file.

See Release for information on the following sections: **Release Times, Initial Phase**, **Initial Intensity**, **Initial Radii of Curvature**, and **Total Source Power**.

Formatting Guidelines for Files Containing Ray Data

The imported data file should be a text file (`*.txt`) arranged in a spreadsheet format; that is, each row corresponds to a distinct ray and should have the same number of columns as all other rows. Columns can be separated by spaces, tabs, or a combination of the two. Begin a line with the percent (“`%`”) character to include comments or empty lines in the data file.

For example, a data file containing the following text would release rays at the positions (0.1, 0.2, 0.6) and (0.2, 0.4, 0.8) in a three-dimensional geometry:

```
% Ray release positions
% qx0 qy0 qz0
0.1 0.2 0.6
0.2 0.4 0.8
```

INITIAL POSITION

Browse your computer to select a text file, then click **Import** to import the data. To remove the imported data, click **Discard**. Enter the **Index of first column containing position data** i to indicate which column represents the first coordinate of the ray position vectors. The default value, 0, indicates the first column.

TRANSFORMATIONS

The distribution of loaded ray release positions may be scaled, rotated, and translated before the rays are released.

To scale the distribution of release positions, enter a value or expression for the **Scale factor** R (dimensionless). The default is 1. This scale factor can be used to correct unit discrepancies between the data file and the model geometry. For example, if the geometry length unit is in meters but the data file lists coordinates in millimeters, enter a scale factor of 0.001.

To rotate the distribution, enter the **Euler angles (Z-X-Z)** α, β, and γ (in 3D) or the **Rotation angle** α (in 2D). The default values are all 0.

In 3D, α is the rotation angle about the space-fixed z-axis, then β is the rotation angle about the transformed x-axis (or x'-axis), and finally γ is the rotation angle about the transformed z-axis (or z'-axis). Positive values indicate counterclockwise rotations.

Enter values or expressions for the components of the **Displacement** vector $\Delta\mathbf{q}$ (SI unit: m). The default is not to apply any translation.

If a translation and another type of transformation (scaling or rotation) are applied, then the translation is always applied after the other transformations.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Expression** (the default), **From file**, **Spherical**, **Hemispherical**, **Conical**, or **Lambertian** (3D only).

- For **From file**, enter the **Index of first column containing direction data** i. The default is 3. The columns are zero-indexed; that is, an index of 0 corresponds to the first

column. If you chose to rotate the loaded ray release positions in the **Initial Position** section, select the **Rotate initial directions** checkbox to apply the same rotation to the initial direction of the released rays. The checkbox is cleared by default.

- For all other settings, the settings are the same as for the Release node.

INITIAL RAY FREQUENCY

This section is available when the **Allow frequency distributions at release features** checkbox is selected under the physics interface **Ray Release and Propagation** section.

Select a **Distribution function**: **None** (the default), **Normal**, **Lognormal**, **Uniform**, **List of values**, or **From file**.

- When **From file** is selected, enter the **Index of column containing frequency data**. The default value is 3. The columns are zero-indexed; that is, an index of 0 corresponds to the first column.
- For all other options, the settings are the same as for the Release node.

INITIAL VALUE OF AUXILIARY DEPENDENT VARIABLES

This section is available if an Auxiliary Dependent Variable has been added to the model.

For each of the active **Auxiliary Dependent Variable** nodes in the model, choose an option from the **Distribution function** list: **From file**, **None** (the default), **Normal**, **Lognormal**, **Uniform**, or **List of Values**.

- When **From file** is selected, enter the **Index of column containing data**. The default value is 3. The columns are zero-indexed; that is, an index of 0 corresponds to the first column.

For all other options, the settings are the same as for the Release node.

Release from Exterior Field Calculation

This feature is only available in 3D.

Use a **Release from Exterior Field Calculation** node to launch rays with an initial intensity and phase distribution matching the exterior field solved for in another study.

The **Release from Exterior Field Calculation** feature is available when **Compute power**, **Compute intensity**, or **Compute intensity and power** is selected in Intensity Computation.

The **Release from Exterior Field Calculation** node can be used for multiscale acoustics modeling. First, the Pressure Acoustics, Frequency Domain interface should be used to compute the near-field acoustic pressure distribution, such as the first few wavelengths surrounding a speaker or another source. The **Exterior Field Calculation** node should be added to this instance of the Pressure Acoustics, Frequency Domain interface, and **Far-field integral approximation for r** $\rightarrow \infty$. should be selected from the **Type of integral** list. Because the near-field calculation is solving for the instantaneous pressure amplitude in individual waves, the *Nyquist criterion* must be fulfilled. Typically, this requires 10 linear elements or 5 second-order elements per wavelength.

Then, in the same model component or a different component, the Ray Acoustics interface can be used to extend the exterior field over an arbitrarily large number of wavelengths, because the ray tracing approach does not require a finite element mesh fine enough to resolve individual oscillations of the acoustic pressure field.

The **Release from Exterior Field Calculation** node is only available in 3D. However, the exterior field function it uses to initialize the ray intensity and phase can be defined either in a 3D model component or a 2D axisymmetric model component. If the previous model component was a 2D axisymmetric component, the intensity distribution of released rays will also be assumed to be axisymmetric.

See Release for information on the **Initial Values of Auxiliary Dependent Variables** section and the optional **Release Times** section.

INITIAL COORDINATES

Select an option from the **Grid type** list: **All combinations** (the default) or **Specified combinations**.

Linear, Rectangular, and Arbitrary Grids

Enter **Initial coordinates** based on space dimension ($q_{x,0}$, $q_{y,0}$, and $q_{z,0}$) for the ray positions or click the **Range** button ( ) to select and define a range of specific coordinates.

If **Specified combinations** is selected, the number of initial coordinates entered for each space dimension must be equal, and the total number of rays released is equal to the length of one of the lists of initial coordinates. If **All combinations** is selected, the total number of rays released is equal to the product of the lengths of each list of initial coordinates.

Previewing Grid Points

In the **Initial Coordinates** section, you can click the **Preview Initial Coordinates** and **Preview Initial Extents** buttons to visualize the ray release positions. Clicking **Preview Initial Coordinates** will cause a point to appear in the Graphics window for every release position. Clicking **Preview Initial Extents** will cause a bounding box to appear, indicating the spatial extents of the released rays.

RAY DIRECTION VECTOR

Select an option from the **Ray direction vector** list: **Spherical** (the default), **Hemispherical**, or **Conical**.

- For **Spherical** a number of rays are released at each point, sampled from a spherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50.
- For **Hemispherical** a number of rays are released at each point, sampled from a hemispherical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then select an option from the **Hemisphere axis** list: **From Euler angles** or **User Defined**. For **User Defined** enter coordinates for the **Hemisphere axis r** based on space dimension.
- For **Conical** a number of rays are released at each point, sampled from a conical distribution in wave vector space. Enter the **Number of rays in wave vector space** N_w (dimensionless). The default is 50. Then select an option from the **Cone axis** list: **From Euler angles** or **User Defined**. For **User Defined** enter coordinates for the **Cone axis r** based on space dimension. Then enter the **Cone angle** α (SI unit: rad). The default is 30 degrees.

Note that unlike the Release from Grid feature, the options **Expression** and **Lambertian** are not available here. In addition, if **Conical** is selected, the cone must be isotropic. This is because each ray is assumed to subtend approximately the same solid angle, so direction distributions that give unequal weight to some directions (like **Lambertian**) or options with no direction distribution at all (like **Expression**) cannot initialize the ray intensity and power in a consistent and physically meaningful way.

EXTERIOR FIELD VARIABLE NAME

Enter an expression for the **Exterior field variable name**. The default expression is `pext`. This expression must match the variable name in a corresponding **Exterior Field Calculation** node.

SOURCE ORIENTATION

Use the settings in this section to orient the ray source. This allows, for example, the release of rays from a speaker to be modeled with the speaker oriented in many different directions, while only solving for the exterior field once.

Enter the **Euler angles (Z-X-Z)**. The labels Z-X-Z indicate the order in which rotation about the different axes is performed. First, the local coordinate system of the exterior field is rotated about its Z-axis by the angle α. Then it is rotated about its X-axis (which is now at an angle α to the global x-axis) by the angle β. Finally, this local coordinate system is rotated about its new Z-axis by the angle γ. All three inputs are plane angles (SI unit: rad) with default values of 0.

The **Initial Value of Auxiliary Dependent Variables** is documented for the Release feature.

Source with Directivity

This feature is only available in 3D.

Use the **Source with Directivity** node to release rays with initial intensity or power following a spatial directivity function. This is useful for modeling loudspeakers or human talker sources in a room acoustics context. The rays will emanate outward from a point in either a spherical or hemispherical distribution. Predefined variables for the azimuthal and polar angles within this distribution can be used to define the spatial directivity function, either as a sound pressure level or in terms of absolute power.

COORDINATE SYSTEM SELECTION

By default, **Global coordinate system** is selected from the **Coordinate system** list. Other user defined coordinate systems can also be selected from the list. However, boundary coordinate systems are not included in the list because there is no requirement for the ray release position to be on a boundary.

There are two main reasons to choose a transformed coordinate system for this feature:

- When using a **Hemispherical** distribution of rays (see the **Ray Direction Vector** section), the first basis vector of the selected coordinate system defines the hemisphere axis. However, if the **Specify local coordinates** checkbox is selected in the **Local Coordinates** section, then the **Forward direction** $\mathbf{e}_f$ defines the hemisphere axis.

(This **Forward direction** can be defined in either global coordinates or in a transformed coordinate system.)

2 The **Source with Directivity** node defines built-in variables for the azimuthal and polar angle. For a physics interface with the default name `rac` and a **Source with Directivity** node with the default name `swd1`, these variables are called `rac.swd1.phi` (azimuthal angle) and `rac.swd1.theta` (polar angle). These angles are defined with respect to the selected coordinate system; for example, if the **Global coordinate system** is used, then `rac.swd1.theta` is the angle between the ray direction vector and the positive z-axis, while `rac.swd1.phi` is the angle of the projected ray direction vector in the xy-plane, measured counterclockwise from the positive x-axis.

No matter which coordinate system is selected, the **Ray release position** (see the **Initial Position** section) is always specified in global Cartesian coordinates. Thus the selected coordinate system controls the orientation of the source while the global Cartesian system always controls its position.

RELEASE TIMES

This section is only shown if the **Allow multiple release times** checkbox is selected in the physics interface **Advanced Settings** section. Furthermore, this **Advanced Settings** section is only shown when **Advanced Physics Options** are enabled (click the **Show More Options** button in the **Model Builder** toolbar).

Enter **Release times** (SI unit: s) or click the **Range** button ( ) to select and define a range of specific times. At each release time, rays are released with initial position and ray direction vector as defined in the following sections.

INITIAL POSITION

The rays are released in a spherical or hemispherical distribution of directions emanating outward from a single point. Here, enter the components of the **Ray release position** vector $\mathbf{q}_0$ (SI unit: m). By default the ray release position is at the origin.

RAY DIRECTION VECTOR

Select an option from the Ray direction vector list: **Spherical** (the default) or **Hemispherical**. If **Hemispherical** is selected, the hemisphere axis is not defined in this section; the axis is either the first basis vector in the selected coordinate system (see the **Coordinate System Selection** section) or the **Forward direction** $\mathbf{e}_f$ specified in the **Local Coordinates** section.

INTENSITY AND POWER

This section is shown when either ray intensity or ray power, or both, are solved for, meaning that any option except **None** has been selected from the **Intensity computation** list in the physics interface **Intensity Computation** section.

Select an option from the **Directivity** list: **Specify spatial directivity** (the default) or **Specify total source power**.

- For **Specify spatial directivity**, enter a value or expression for the **Spatial directivity** $D(\varphi, \theta)$ in dB. The default expression is `0[dB]*cos(rac.swd1.phi)*sin(rac.swd1.theta)`. The expression is a placeholder that shows the variable names for the two polar angles. The directivity will typically be defined by an interpolation function that represents a given source like a loudspeaker. Also, specify the **Reference level** L_{ref} in dB (default 65 dB) and the **Reference distance** R_{ref} (default 1 m). The ray intensity or power is initialized such that the sound pressure level of each ray equals the sum of the **Spatial directivity** and the **Reference level**, evaluated for that ray's initial direction, when the ray reaches a distance from the source equal to the **Reference distance**.
- For **Specify total source power**, enter a value or expression for the **Total source power** P_{src} (SI unit: W). The default is 1 W. Enter a **Power weighting factor** P_{wt}, which can have any unit. The default expression is 1. The initial intensity and power of each released ray will be proportional to the **Power weighting factor** evaluated for that ray, normalized such that the sum of the power over all rays equals the **Total source power**. Much like the **Spatial directivity**, the **Power weighting factor** can be a function of ray variables such as the azimuthal and polar angles. For this option, the reference distance is assumed to be 1 m. The total power divided with the surface area of the sphere/hemisphere at 1 m is used when computing the intensity along rays. Remember that the intensity is a local sound field quantity.

INITIAL RAY FREQUENCY

This section is shown when the **Allow frequency distributions at release features** checkbox is selected in the physics interface **Ray Release and Propagation** list. Unlike some other ray release features, only a single value of the **Initial ray frequency** f_0 (SI unit: Hz) may be entered, not a distribution or a list of values. The default is 1 kHz.

INITIAL PHASE

This section is shown when ray intensity is solved for, and in addition the **Compute phase** checkbox is selected in the physics interface **Intensity Computation** section. Enter a value or expression for the **Initial phase** Ψ_0 (SI unit: rad). The default is 0.

LOCAL COORDINATES

The **Specify local coordinates** checkbox is cleared by default. When it is selected, enter the components of the **Forward direction** $\mathbf{e}_f$ and the **Transverse direction** $\mathbf{e}_t$ (dimensionless). By default these directions are the first and third basis vectors, respectively, in whichever coordinate system was selected from the **Coordinate System Selection** section. These vectors are used to set the hemisphere axis (if **Hemispherical** is selected from the **Ray Direction Vector** list). They also control the definition of built-in variables for the azimuthal and polar angles, which can be used to define a spatial directivity function for the released rays.

Assigning Spatial Directivity to a Source

The **Initial Value of Auxiliary Dependent Variables** is documented for the Release feature.

Ray Termination

Use the **Ray Termination** feature to remove rays from the model as they exit the geometry, or to annihilate rays of sufficiently low intensity or power. Only one **Ray Termination** node can be added to the model, but it is possible to combine position-based and intensity-based criteria for removing rays if the ray intensity or power is computed by the model.

TERMINATION CRITERIA

Select an option from the **Spatial extents of ray propagation** list: **Bounding box, from geometry** (the default), **Bounding box, user defined**, or **None**. This list controls whether rays are terminated based on their positions.

- For **Bounding box, from geometry**, a ray is immediately terminated if it reaches the surface of the bounding box of the geometry, defined as the smallest rectangle or rectangular prism with edges parallel to the axes of the global coordinate system that encloses all of the geometric entities. This bounding box is extended by 5% in each direction so that rays can still interact with boundaries that coincide with the

perimeter of the bounding box. Therefore, rays can propagate a short distance away from the geometry before disappearing.

- For **Bounding box, user defined**, a ray is immediately terminated if it reaches the surface of a box of user-specified dimensions. To define the box, enter maximum and minimum values of each spatial coordinate.
- For **None**, the rays that propagate outward from the geometry will continue to propagate until the study is complete. If the maximum path length for the **Ray Tracing** study step is too large, this may result in trajectories that are extremely long relative to the characteristic geometry size.

If rays are released from outside the bounding box, they will not immediately disappear; instead, they will only disappear if they enter the bounding box and then exit it again.

Intensity- and Power-Based Termination Criteria

If the ray intensity or power is being computed in the model, you can then terminate rays based on either or both of these quantities.

Select an option from the **Additional termination criteria** list. **None** (the default) is always available. If ray intensity is solved for, then **Intensity** is available. If ray power is solved for, then **Power** is available. If both ray intensity and power are solved for, the **Intensity and Power** option is also shown.

- For **Intensity** or **Intensity and power**, enter the **Threshold intensity** I_{th} (SI unit: W/m^2). The default is `1e-12[W/m^2]`. Rays will be removed from the model if their intensity decreases below the specified value.
- For **Power** or **Intensity and power**, enter the **Threshold power** Q_{th} (SI unit: W). The default is `1e-3[mW]`.

The **Ray Termination** feature can terminate rays at the exact time at which their intensity or power reaches the specified threshold (to within numerical precision), if there exists a closed-form analytic expression for this time. Such a closed-form analytic solution exists if:

- The rays represent planar wavefronts in an absorbing medium, or
- The rays represent spherical or ellipsoidal wavefronts in a nonabsorbing medium.

Otherwise, the rays are stopped at the first discrete time step taken by the solver for which the intensity or power is less than the threshold value.

Ray Termination Theory

Modeling with the Ray Acoustics Interface

In this section:

- Mixed Diffuse and Specular Wall Conditions
- Assigning Directivity to a Source
- Impulse Response Plot and Receiver Dataset
- Stopping Rays for a Given Condition
- Mesh Guidelines
- Nonlocal Couplings
- Using Ray Detectors
- Other Results Plots, Datasets, and Derived Values

Mixed Diffuse and Specular Wall Conditions

At a wall where both specular reflections and diffuse reflections occur, it is common to define the scattering coefficient s. This coefficient relates the amount of energy that is specularly E_s and diffusely E_d reflected to the (total) absorption coefficient α. The normalized total reflected energy is E_t. The relations between these are

$$E_t = 1 - \alpha = E_s + E_d$$
$$E_s = (1-\alpha)(1-s) = 1 - \alpha_s$$
$$E_d = (1-\alpha)s = 1 - \alpha_d$$

where α_s is the specular absorption coefficient and α_d is the diffuse absorption coefficient.

In the **Ray Acoustics** interface, the mixed reflection condition is treated using a Monte Carlo like approach. When a ray hits a mixed wall condition, the ray is either diffusely or specularly reflected according to the **Probability of specular reflection** γ_s. This means that the probability plays the role of the scattering coefficient, by setting $\gamma_s = 1 - s$. The **Specular absorption coefficient** and the **Diffuse absorption coefficient** should both be set equal to the total absorption α.

Assigning Directivity to a Source

Acoustic sources often have a certain directivity, for example, the directivity of a loudspeaker or even the directivity of a human speaker. When setting up sources in a ray acoustics model, this directivity can be specified using the Source with Directivity feature. The spatial directivity can be imported from measured data using an interpolation function that depends on the two polar angles as well as the band center frequency.

As an alternative, the source directivity for a single frequency simulation can be automatically coupled from a pressure acoustics model using the Release from Exterior Field Calculation feature.

Impulse Response Plot and Receiver Dataset

The impulse response (IR) in a room acoustics simulation can be analyzed by collecting the ray information using the **Receiver** dataset and then analyzing the results using the **Impulse Response** plot. Room acoustic metrics like reverberation times, early energy metrics, and speech intelligibility can also be postprocessed using the **Energy Decay** subfeature. The IR can be determined for sources and boundary data (absorption coefficients, scattering parameters, source directivity, volume attenuation, and so on) given in octave, 1/3 octave, or 1/6 octave bands.

PREPARING A ROOM ACOUSTICS SIMULATION

When setting up a *Ray Acoustics* simulation and preparing it for a room acoustics simulation where the impulse response (IR) is an important output, certain considerations and steps need to be taken. They are as follows:

- Decide on the interpretation of the frequency in the ray acoustics simulation. To be able to reconstruct the IR with the dedicated plot, the frequency should represent the center frequency of octave, 1/3 octave, or 1/6 octave bands. The center frequency is also used as input for boundary conditions (absorption coefficients and scattering parameter), for sources (source strength and directivity), and for the volume absorption in the medium, typically air.
- Create parameters under **Global Definitions** > **Parameters** for the band center frequency and the source power, for example, `f0` and `P0` respectively.
- Set up interpolation functions for the properties that depend on the frequency. The data can easily be stored in a **Spreadsheet** formatted file and interpolation can be set to **Nearest neighbor**. An example of the format for a `.txt` file that specifies

absorption coefficients for some boundaries in a simulation could be (data is invented):

```
%f0, wall, entrance, window, floor, diffusers, seats
125 0.1 0.4 0.01 0.02 0.1 0.05
250 0.07 0.2 0.01 0.03 0.1 0.06
500 0.05 0.12 0.02 0.05 0.1 0.07
1000 0.04 0.07 0.02 0.1 0.1 0.15
2000 0.04 0.05 0.02 0.3 0.1 0.35
4000 0.05 0.05 0.03 0.5 0.1 0.4
8000 0.08 0.05 0.04 0.5 0.1 0.6
16000 0.1 0.05 0.05 0.5 0.1 0.6
```

- Create parameters under **Global Definitions** > **Parameters** for the receiver radius and number of released rays. These two quantities are linked by the expected error in a given time interval Δt of the impulse response, see Ref. 6 for details. With the room volume V, the receiver radius R, and for an expected error of 1 dB, the number of released rays N needed is

$$N = 4.34^2 \frac{V}{\pi R^2 c \Delta t}$$

  The receiver radius can be taken as $R = 0.3$ m to match the width of a common seat, and a standard time interval is $\Delta t = 0.01$ s. For small rooms or car cabins, it can be useful to set a receiver radius that represents the actual size of a microphone or slightly larger. The value obtained from this calculation should be rounded up to ensure an integer number of released rays. A higher value can also be chosen and will result in increased resolution.
- Add the Receiver feature and add a sphere of the desired radius in the geometry for representing the receiver.
- Make sure that either the **Compute intensity and power** or the **Compute power** option is selected in the Intensity Computation section. The **Compute power** option requires fewer degrees of freedom and is more robust. If it is selected, the ray intensity (`rac.I` and `rac.logI`) cannot be visualized in ray trajectories plots, but both the Sound Pressure Level Calculation on boundaries and the impulse response can be computed. The **Compute power** option is selected per default when adding the Ray Acoustics physics interface.
- Make sure that **Count reflections** is selected in the Additional Variables section. The **Count reflections** option is selected per default when adding the Ray Acoustics physics interface.
- Also make sure that the parameter that you created for the band center frequency (say `f0`) is used as the **Ray frequency** under the **Ray Properties** node.

- Set up the ray acoustics model with appropriate boundary conditions and sources. Generally, all **Wall Conditions** should be set to **Mixed diffuse and specular reflection**, with a default scattering coefficient $s = 0.05$ for flat surfaces.
- Create a **Ray Termination** node and choose **Bounding box, from geometry** as **Spatial extents of ray propagation**. Select **Power** as **Additional termination criteria** and enter the threshold `P0/N*1e-6`. For an omnidirectional source, this expression makes rays disappear once the power they carry has dropped by 60 dB from its initial value.
- To be able to postprocess the IR, a **Parametric Sweep** needs to be used in the study around the **Ray Tracing** study step. The sweep parameter should be the frequency defined in the parameters. To easily create a **Parameter value list** representing the center frequencies of the bands use the **ISO preferred frequencies** entry method.
- For the **Times**, specified under the **Ray Tracing** study step, only enter 0 and the final simulation time (the final time should be slightly larger than the estimated reverberation time). COMSOL uses small internal steps that accurately account for all reflections. The so-called **Extra Time Steps** are used when reconstructing the IR and other data (see below).
- Run the simulation.

RECEIVER DATASET

A **Receiver 2D** ( ) or **Receiver 3D** dataset ( ) dataset is automatically created for each Receiver feature present in the model when solving. To add it manually, it can be selected from the **More 2D Datasets** submenu and the **More 3D Datasets** submenu, respectively, to collect the data necessary to visualize the impulse response using an Impulse Response Plot. The **Impulse Response** plot uses the data from a **Receiver** dataset as input.

Data

Select the appropriate ray dataset in the **Dataset** list. Then select the frequency parameters that should be used by the receiver; select one frequency for a single band analysis or all for the broad band analysis. This is the typical behavior if a parametric sweep has been set up, as described in Preparing a Room Acoustics Simulation.

Finally, select if the **Receiver** is **Local** or pointing to a Receiver feature, for example, **Receiver 1 (rac/rec1)** referring to the first receiver feature in physics. Pointing to a given **Receiver** feature defined in the physics will ensure an efficient subsequent evaluation and analysis of the impulse response in the **Impulse Repose** plot. If the **Local** option is used, the intersection of all rays and the locally defined receiver (see below) are

recomputed. This step may be very time-consuming for models with many rays. It does however offer the flexibility of not having a fixed receiver defined in the geometry.

Receiver

The **Receiver** section is only visible if the **Local** option is selected as the **Receiver** in the **Data** section. For this option, the receiver behaves as a (transparent) sphere with a given radius and located at a center position (the microphone needs to have a certain finite size such that there is a reasonable probability of rays interacting with it). By changing the center position, the receiver can be moved around without the need to run the simulation again. However, computation of the intersection of rays with this virtual receiver can be time-consuming.

- Under **Center**, specify the **x**, **y**, and (3D only) **z** coordinates for the center of the receiver.
- Under **Radius**, specify the radius of the receiver. From the **Radius input** list, choose **Variable size (large room volume)** to determine the radius using an expression (see below), or choose **Fixed size** to enter a value for the radius in the **Radius** field (SI unit: m). Different theories exist for the appropriate size of the receiver. For room acoustic applications, it is recommended to use a fixed radius from which the number of rays can be derived, as described in Preparing a Room Acoustics Simulation.

  When selected, the following built-in expression determines the radius R of the receiver:

$$R = \log_{10}(V) d_{\mathrm{SR}} \sqrt{\frac{4}{N}}$$

  where you enter the values for the **Number of rays**, N; **Room volume**, V (SI unit: m^2); and **Source-receiver distance**, d_{SR} (SI unit: m), to determine the radius. See Ref. 19 for details on the expression. This computed receiver size is optimized for modeling of large rooms. However, it will return a different value for each source-receiver pair if there are multiple sources and receivers present in the simulation, and it will not necessarily correspond to the dimensions of a listener.

Directivity

The **Directivity** section is only visible if the **Local** option is selected as the **Receiver** in the **Data** section. From the **Directivity type** list, choose **Omnidirectional** (the default), **User**

defined (dB), or **User defined (linear)** and enter a directivity expression in the **Expression** field.

- The **User defined (dB)** option applies the given gain to the received signal (positive or negative gain).
- The **User defined (linear)** option applies a linear gain. If a negative value is used the phase of the arriving ray is inverted.

The expression can depend on the ray direction vector components (`rac.nix`, `rac.niy`, `rac.niz`) in order to set up a complex directivity pattern.

Extra Time Steps

The **Extra Time Steps** section is only visible if the **Local** option is selected as the **Receiver** in the **Data** section. Use the default **All** option. The impulse response plot has been tuned to use this option in order to get accurate and fast evaluation of arrival time and ray power.

Advanced

The **Advanced** section is only visible if the **Local** option is selected as the **Receiver** in the **Data** section. For the **Interpolation between time steps** list always use the default **Linear** interpolation option. The **Cubic** interpolation is not applicable for the impulse response computation and can lead to erroneous results.

Under **Normal variables** and **Other variables**, if desired, you can change the default names of the created variables:

- The normal direction of the receiver surface (**nx**, **ny**, and, in 3D, **nz**).
- The **Distance traveled**, that is the distance traveled by a ray inside the receiver.
- The receiver **Volume**.
- The **Directivity**, that can be used to visualize the expression for the directivity.
- The **First ray arrival time**.

Evaluation and Export

The main purpose for the **Receiver** dataset is to be used as input for the Impulse Response Plot. However, the data generated by the dataset can also be exported and used in an external analysis tool or software (ray power and arrival times).

Right-click the dataset and select **Add Data to Export**, then enter the desired data. For example, export the arrival time of the rays `t`, the frequency of the rays `rac.f`, the power of the rays `rac.Q`, the distance traveled by the rays in receiver `re1dist`, and so forth. The exported data can be stored and sorted in several formats.

When working with phase alignment of loudspeakers, it can be useful to evaluate the arrival time if the first ray. This is easily done in a **Derived Values** > **Global Evaluation** feature that points to the receiver dataset. Simply evaluate the special variable `re1first`.

See also the Receiver 2D and Receiver 3D section in the *COMSOL Multiphysics Reference Manual*.

IMPULSE RESPONSE PLOT

In room acoustics applications, the impulse response (IR) of a source-receiver configuration represents one of the most important postprocessing results. To create the plot based on the data source defined in the **Receiver** dataset, add an **Impulse Response** subnode ( ) to a 1D Plot Group to create the impulse response plot. After defining the characteristics of the impulse response, click **Plot** ( ) to create the plot. The **Energy Decay** ( ) subnode can be added to perform an analysis of the impulse response to get room acoustic metrics and the energy/level decay curves. The first rendering of the IR plot can take some time. As the data is cached, any subsequent changes to the plot will be fast.

The *Small Concert Hall Acoustics* model is a tutorial on how to compute the impulse response using the Ray Acoustics interface. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/small_concert_hall**

In the ray tracing methodology, it is, as discussed above, typical to characterize sources and boundaries in frequency bands. This also means that some of the frequency content is lost. This frequency content is reconstructed when determining or reconstructing the IR. For each ray that interacts with the receiver, a small piece of signal with appropriate amplitude, arrival time, phase, and frequency content is added to the IR (the filter bank kernel). The sum of all the contributions reconstruct the full IR.

Data

Select if the **Source** is a **Dataset** or **Function**. If **Dataset** is selected the plot will analyze data from a Ray Acoustics simulation collected by a **Receiver** dataset. If **Function** is selected the input data can be impulse response data stored in an interpolation function, for example, from importing a WAV file with measured data.

If the **Source** is set to **Dataset**, then from the **Dataset** list, choose the appropriate Receiver Dataset. Then from the **Frequency interpretation** list choose an interpretation of the frequency: **Octave** (the default), **1/3 octave**, or **1/6 octave**. This selections should coincide with the frequency interpretation used in boundary conditions and sources in the underlying ray acoustics simulation, as described in Preparing a Room Acoustics Simulation. The frequency content and resolution of the impulse response is based on this selection.

Expressions

Set up the variables that are necessary for reconstructing the impulse response (typically use the defaults). These variables are used to define and add the contribution of each ray, that intersects the receiver dataset, to the impulse response.

- Define the frequency variable for the rays (SI unit: Hz) in the **Frequency** field. The default is `rac.f`.
- Define the power variable for the rays (SI unit: $\mathrm{kg{\cdot}m^2/s^2}$) in the **Power** field. The default is `rac.Q`.
- Define the density (SI unit: $\mathrm{kg/m^3}$) in the **Density** field. The default is `rac.rho`.
- Define the speed of sound (SI unit: m/s) in the **Speed of Sound** field. The default is `rac.c`.
- Define the number of reflections undergone by the rays (SI unit: 1) in the **Number of reflections** field. The default is `rac.Nrefl`.

x-Axis Data

From the **Transformation** list, choose a transformation of the data on the x-axis: **None** (the default), for no transformation, to show the time domain signal, or **Discrete Fourier transform**. The latter will perform an FFT of the IR and depict the power spectra as function of frequency.

Advanced

The following advanced settings are available to control some aspects of the IR reconstruction as well as the underlying filter kernel used.

- Define a sampling frequency f_s (SI unit: Hz). The default is `44100`.
- Define the **Zero padding length**. The default is `11025`.

The next two options are legacy options (to achieve behavior for COMSOL Multiphysics releases older than version 5.6).

- Select **Remove noncausal signal** to remove the signal (force it to zero) prior to the arrival of the first ray. Using this option will alter the energy content of the early energy component of the IR.
- Select **Use fully randomized phase** to use the legacy method for reconstructing the IR. The fully randomized phase method (see Ref. 20) can lead to unwanted cancellations of the direct sound and early energy contributions.

The following options control the filter kernel used to reconstruct the impulse response. Advanced options allow to modify the kernel, enter a user-defined analytical kernel, or a user-defined kernel based on an interpolation function. The default is to use a Brick-wall with Kaiser window filter kernel (see Ref. 21).

- Select **Filter**, in the **Display** list, to show and inspect the filter bank in the **Graphics** window, click **Plot** ( ) to create the plot. To switch between time domain and frequency domain representation change the **Transformation** option in the **x-Axis Data** section.

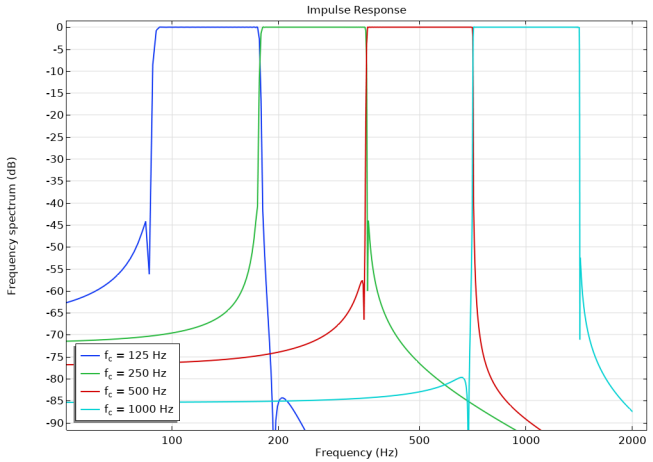


Figure 8-6: Example of octave band filters depicted using the Show the filters option.

- Select the **Filter kernel** to use **Brick-wall with Kaiser window** (the default) or **User defined**. When **User defined** is selected enter the expression for the filter kernel $h_{\mathrm{fc}}[t]$. Use and set up parameters in the **Parameters** table (as you would use variables). The parameter `fc` is reserved for the (exact) band center frequency; `fr` is reserved for the ray frequency (this is typically the nominal band center frequency selected in the

study); `fs` is reserved for the sampling frequency; `t` is for time; and `Np` is the padding length. Note that per default the **User defined** option sets up the brick-wall with Kaiser window filter kernel for octaves.

- If the default **Brick-wall with Kaiser window** is selected, you can control the ripple factor δ (the default is `0.05`). The effect on the filter can be seen using the **Show the filters** option.

Coloring and Style, and Legends

You can also make change in the **Coloring and Style** section or add and format a legend in the **Legends** section.

See also the Impulse Response section tn the *COMSOL Multiphysics Reference Manual.*

ENERGY DECAY AND OBJECTIVE ROOM ACOUSTIC METRICS

For further analysis of the impulse response, add the **Energy Decay** ( ) subnode to the **Impulse Response** plot. Once selections have been made, click **Plot** ( ) to create the plot.

Display

Select the **Band type** as **Broadband** (the default) or **Individual bands**. For the **Individual bands** option select the **Band frequency**, either an individual band or **All**. For **All**, all the level or energy decay curves are plotted.

Select the **Plot** as **Energy decay** (the default), **Level decay**, or **Modulation transfer function**, to define the type of graphical analysis of the impulse response to be plotted in the **Graphics** window.

Table

Select the room acoustic metrics to be included in the results table, per default all objective quality metrics are selected. See Ref. 8 for the definitions. The options include:

- Definition: D
- Clarity: C_{50} and C_{80}
- First ray arrival time: t_{r}
- Center time: t_{s}
- Early decay time: EDT
- Reverberation times: T_{20}, T_{30}, and T_{60}

- Speech intelligibility metrics: SNR and STI

The default name of the table is **Objective Quality Metrics**. The table displays the results in columns with the band center frequency `fc` in the first column. The results table can also be located under the **Results** > **Tables** node. A plot of the results located in the table can be created using the **Table Graph** plot.

The speech transmission index (STI) metric is defined as a single value that combines information from 14 modulation transfer function bands and 7 octave bands. To get the value specified by the standard, solve the model for octaves from 125 Hz to 8 kHz. When evaluating use the **Band type** set to the default **Broadband** option. Details about the STI computation can be found in Ref. 8 and Ref. 9.

CUSTOM PLOTS

The energy response depicted as a reflectogram or echogram can be plotted using a Ray (Plot) and depict the expression `re1dist*rac.Q/re1vol`. This will give the intensity received by the "microphone" for each ray.

Stopping Rays for a Given Condition

For some ray acoustics applications it can be advantageous to stop rays once their intensity falls below a certain threshold. This can be simply achieved at walls using the **Primary ray condition** with an **Expression**. Here you can enter a logical expression that must be satisfied for a ray to undergo a certain type of ray-wall interaction. For the intensity specify a condition of the type `rac.I<threshold`. Then the ray is either reflected according the selected condition or it can **Freeze** or **Disappear**, depending on your selection.

The number of reflections can be used in such an **Expression**. This variable is readily set up under the Additional Variables section.

The Ray Termination feature can also be used in a similar manner to remove rays for certain conditions, without the need for a wall interaction.

Mesh Guidelines

When setting up a mesh for a *Ray Acoustics* simulation the following guidelines should be followed:

- For modeling ray propagation within domains and interaction with flat surfaces, the mesh can be very coarse.
- When rays may interact with curved boundaries (either reflection or refraction), the accuracy of the reinitialized ray trajectory (that is, the reinitialized wave vector **k**) depends on the mesh. To create a very fine mesh on curved surfaces without making the mesh unnecessarily fine elsewhere, use the **Curvature factor** in the **Size** settings window. The closer this number is to zero, the finer the mesh will be on a curved surface. A value of 0.1 seems adequate for most applications. If there are small, tightly curved surfaces in the model, it might also be necessary to reduce the Minimum element size to allow the small curvature factor to do its job.
- When computing ray intensity, the accuracy of the reinitialized intensity at a curved wall or material discontinuity usually does not depend on the level of mesh refinement because it uses surface curvature variables that are based directly on the geometry, not on the mesh. However, if ALE is included in the model, the accuracy of the surface curvature does depend on the mesh.
- Domains that are not included in the selection for the Ray Acoustics interface do not need to be meshed. However, a boundary mesh must always be defined wherever any boundary condition is applied. In addition, it is assumed that the speed of sound and density in all deselected domains are the **Speed of sound in exterior domains** and **Density of exterior domains**, which are specified in the physics interface **Ray Release and Propagation** section. Thus, media with graded or temperature-dependent material properties must always be included in the selection of the Ray Acoustics interface, and they must always be meshed.
- **Accumulator** features (both on surfaces and on domains) work by defining additional dependent variables using constant shape functions. When using an **Accumulator** on a **Wall**, for example, the contribution from each ray is distributed uniformly over whatever boundary element it hits. If the mesh on the boundary is very fine, this will make the value of the accumulated variable look noisy unless the number of rays hitting the boundary is much larger than the number of boundary elements.

REPRESENTATION OF CURVED SURFACES

When rays reach the boundaries of geometric entities in a model, they do not interact with an exact parameterized representation of the geometry. Rather, they propagate

through the mesh elements that discretize the modeling domain and interact with the boundary elements that cover the surfaces of the geometric entities.

When the surfaces of the geometry are flat, the shape of the surface mesh is indistinguishable from the shape of the geometric entities themselves. Therefore, the fact that rays interact with the mesh instead of the geometry does not introduce any discretization error, and it is possible to accurately compute ray trajectories even when the mesh is extremely coarse.

Curved surfaces in the geometry, however, usually incur a significant amount of discretization error when predicting how rays will interact with them. The time and location at which the ray interacts with the boundary mesh element might be slightly different from the time at which it would have interacted with an exact representation of the surface. In addition, the tangential and normal directions on the boundary mesh element may differ from the tangential and normal directions on the surface, affecting the accuracy of boundary conditions that involve the tangential and normal directions, such as the **Specular reflection** condition.

The order of the curved mesh elements used to determine the geometry shape is controlled by the **Geometry shape order** list in the **Model Settings** section of the **Settings** window for the main **Component** node. If **Automatic**, the default, is selected, the curved mesh elements are usually represented by quadratic curves; in some cases, linear functions are used to prevent inverted mesh elements from being created.

The effect of the geometry shape order is most notable on a coarse mesh, as shown in Figure 8-7. The mesh elements are shown as pale gray lines in the background and the ray trajectories are represented as thick red arrows. The rays initially propagate downward and are specularly reflected by a parabolic surface. If **Linear** is selected from the **Geometry shape order** list, all rays that hit the same boundary element are specularly reflected in the same direction, as shown on the left. Even though the bottom surface is parabolic, the rays do not all intersect at a single focus due to the discretization error. If **Quadratic** or **Automatic** is selected, rays that hit the same boundary element can still be reflected in different directions because the tangential and normal directions can vary along the surface of the curved element. As a result, the rays reflected by the parabolic surface all intersect at a well-defined focal point as expected.

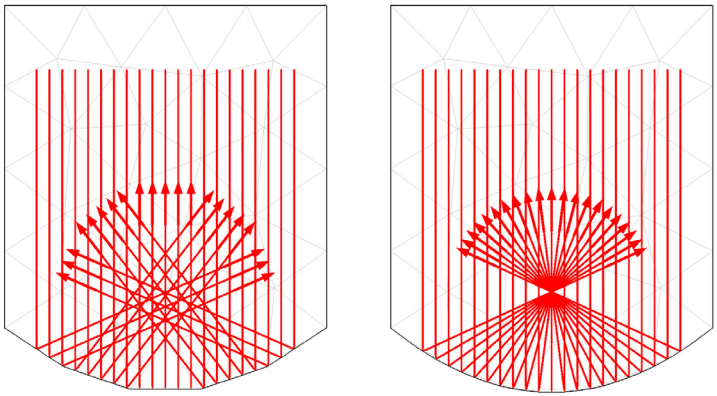


Figure 8-7: Comparison of rays being specularly reflected at a curved boundary represented using linear elements (left) and quadratic elements (right).

RAY TRACING IN AN IMPORTED MESH

It is also possible to compute ray trajectories in an imported mesh. The mesh can be imported from a COMSOL Multiphysics file (`.mphbin` for a binary file format or `.mphtxt` for a text file format) or from a NASTRAN file (`.nas`, `.bdf`, `.nastran`, or `.dat`).

If the mesh is imported from a COMSOL Multiphysics file, the imported mesh always uses linear geometry shape order for the purpose of modeling ray-boundary interactions, even if the model used to generate the mesh had a higher geometry shape order.

If the mesh is imported from a NASTRAN file, the ray-boundary interactions may be modeled using either linear or higher geometry shape order. If **Export as linear elements** is selected when generating the NASTRAN file, or if **Import as linear elements** is selected when importing the file, then linear geometry shape order will be used.

Nonlocal Couplings

The purpose of a model is often to compute the sum, average, maximum value, or minimum value of a quantity over a group of rays, such as the average intensity or the maximum path length. An instance of the Ray Acoustics interface with Name `rac` creates the following four nonlocal couplings:

The built-in nonlocal coupling names have recently changed, but the couplings with the old names are still available. In their descriptions the couplings with the old names are marked as “deprecated”. Both the old and new names are listed in Table 8-3.

- `rac.sum(expr)` evaluates the sum of the expression `expr` over the rays. The sum includes all rays that are active, frozen, or stuck to boundaries. It excludes rays that have not yet been released and those that have disappeared.
- `rac.sum_all(expr)` evaluates the sum of the expression `expr` over all rays, including those rays that are not yet released or have disappeared. Since the coordinates of unreleased and disappeared rays are not-a-number (`NaN`), the sum may return `NaN` if the model includes unreleased or disappeared rays. An expression such as `rac.sum_all(isnan(qx))` can be used to compute the total number of unreleased and disappeared rays.
- `<name>.<name>aveop1(expr)` evaluates the average of the expression `expr` over the active, frozen, and stuck rays. Unreleased and disappeared rays contribute to neither the numerator nor the denominator of the arithmetic mean.
- `<name>.<name>aveop_all1(expr)` evaluates the average of the expression `expr` over all rays. It is likely to return `NaN` if the model includes unreleased or disappeared rays.
- `rac.max(expr)` evaluates the maximum value of the expression `expr` over all active, frozen, and stuck rays.
- `rac.max_all(expr)` evaluates the maximum value of the expression `expr` over all rays.
- `rac.max(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the maximum value of the expression `expr` out of all active, frozen, and stuck rays. For example, the expression `rac.max(rac.I, qx)` would evaluate the x-coordinate `qx` of the ray with the greatest intensity `rac.I`.
- `rac.max_all(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the maximum value of the expression `expr` for all rays, including disappeared and unreleased rays.
- `rac.min(expr)` evaluates the minimum value of the expression `expr` over the active, frozen, and stuck rays.

- `rac.min_all(expr)` evaluates the minimum value of the expression `expr` over all rays.
- `rac.min(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the minimum value of the expression `expr` out of all active, frozen, and stuck rays. For example, the expression `rac.min(rac.I, qx)` would evaluate the x-coordinate `qx` of the ray with the minimum intensity `rac.I`.
- `rac.min_all(expr, evalExpr)` evaluates the expression `evalExpr` for the ray that has the minimum value of the expression `expr` for all rays, including disappeared and unreleased rays.

An instance of the Ray Acoustics interface with the default name `rac` defines the built-in nonlocal couplings shown in Table 8-3.

TABLE 8-3: BUILT-IN NONLOCAL COUPLINGS FOR THE RAY ACOUSTICS INTERFACE.

NAME	NAME	DESCRIPTION
`rac.sum(a)`	`rac.racop1(a)`	Sum over rays
`rac.sum_all(a)`	`rac.racop_all1(a)`	Sum over all rays
`rac.ave(a)`	`rac.racaveop1(a)`	Average over rays
`rac.ave_all(a)`	`rac.racaveop_all1(a)`	Average over all rays
`rac.max(a)`	`rac.racmaxop1(a)`	Maximum over rays
`rac.max_all(a)`	`rac.racmaxop_all1(a)`	Maximum over all rays
`rac.min(a)`	`rac.racminop1(a)`	Minimum over rays
`rac.min_all(a)`	`rac.racminop_all1(a)`	Minimum over all rays
`rac.max(a,b)`	`rac.racmaxop1(a,b)`	At maximum over rays
`rac.max_all(a,b)`	`rac.racmaxop_all1(a,b)`	At maximum over all rays
`rac.min(a,b)`	`rac.racminop1(a,b)`	At minimum over rays
`rac.min_all(a,b)`	`rac.racminop_all1(a,b)`	At minimum over all rays

Using Ray Detectors

A **Ray Detector** feature is a domain or boundary feature that provides information about rays arriving on a set of selected domains or surfaces from a release feature. Such quantities include the number of rays transmitted, the transmission probability, and a logical expression for ray inclusion. The feature provides convenient expressions that can be used in the Filter node of the Ray Trajectories plot, which allows only the rays

which reach the ray detector selection to be visualized. The following variables are provided by the **Ray Detector** feature, with the feature tag `<tag>`:

- `<tag>.Ntf` is number of transmitted rays from the release feature to the ray detector at the end of the simulation.
- `<tag>.alpha` is the transmission probability from the release feature to the ray detector.
- `<tag>.rL` is a logical expression for ray inclusion. This can be set in the **Filter** node of the **Ray Trajectories** plot in order to visualize the rays which connect the release feature to the detector.

The **Ray Detector** feature only creates variables, which does not affect the solution. Therefore, they can be added to a model without the need to recompute the solution, it just needs to be updated. To do this, right-click on the **Study** node and select **Update Solution**. The new variables described above will immediately be available for results processing.

Other Results Plots, Datasets, and Derived Values

A variety of dedicated postprocessing tools are available with this physics interface. They are listed below and referred to in the *COMSOL Multiphysics Reference Manual*.

SOUND PRESSURE LEVEL ON SURFACES AND IN CROSS SECTIONS

The sound pressure level on surfaces is plotted using the **Sound Pressure Level Calculation** subfeature to a **Wall** node.

The SPL can also be visualized on an interior cut-like plane by adding a plane in the geometry, meshing it, and then apply a wall with the **Wall condition** set to **Pass through**. In this way, the boundary will only be used to map the SPL and will have no influence on the ray propagation.

PLOTS

In the *COMSOL Multiphysics Reference Manual*:

- Ray (Plot)
- Ray Trajectories
- Filter for Ray and Ray Trajectories
- Particle (Plot)
- Impulse Response
- Plot Groups and Plots

RAY DATASET

In the *COMSOL Multiphysics Reference Manual*:

- Ray (Dataset)
- Receiver 2D and Receiver 3D
- Datasets

RAY EVALUATION DERIVED VALUES

In the *COMSOL Multiphysics Reference Manual*:

- Ray Evaluation
- Derived Values, Evaluation Groups, and Tables

Theory for the Ray Acoustics Interface

The Ray Acoustics Interface theory is described in this section:

- Introduction to Ray Acoustics
- Initial Conditions: Position
- Initial Conditions: Direction
- Material Discontinuity Theory
- Intensity and Wavefront Curvature
- Intensity and Phase Reinitialization
- Wavefront Curvature Calculation in Graded Media
- Attenuation Within Domains
- Ray Termination Theory
- Accumulator Theory: Domains
- Accumulator Theory: Boundaries
- Sound Pressure Level Calculation Theory
- Assigning Spatial Directivity to a Source
- Impulse Response Reconstruction
- References for the Ray Acoustics Interface

Introduction to Ray Acoustics

It is often convenient to express an acoustic wave in a stationary fluid as a plane wave of the form

$$\phi = ae^{i\Psi}$$

where ϕ (SI unit: m^2/s) is the velocity potential of the fluid, a is its amplitude, and Ψ (SI unit: rad) is the phase. When the distance from any source is many orders of magnitude larger than the wavelength of the acoustic wave, the wave may be assumed to be locally plane; that is,

$$\Psi = -\mathbf{k} \cdot \mathbf{q} + kct$$

where $\mathbf{k}$ (SI unit: rad/m) is the wave vector, $\mathbf{q}$ (SI unit: m) is the position vector, c (SI unit: m/s) is the speed of sound in the medium, and t (SI unit: s) is the time.

While the velocity potential is defined with a stationary fluid in mind, it is possible to apply the same treatment to a homogeneous fluid moving at velocity $\mathbf{u}$ (SI unit: m/s) by first formulating the equation of the acoustic wave in a coordinate system in which the fluid is stationary. This results in a more general form of the acoustic wave equation,

$$\Psi = -\mathbf{k} \cdot \mathbf{q} + (kc + \mathbf{k} \cdot \mathbf{u})t$$

By defining the angular frequency of the wave as

$$\omega = ck + \mathbf{u} \cdot \mathbf{k}$$

it follows that the equation of the acoustic wave may be expressed in the Hamiltonian form

$$\frac{d\mathbf{k}}{dt} = -\frac{\partial \omega}{\partial \mathbf{q}}$$
$$\frac{d\mathbf{q}}{dt} = \frac{\partial \omega}{\partial \mathbf{k}}$$

This is a mixed time-frequency formulation that has many advantages. See, for example, Ref. 1 and Ref. 5 for further details. It allows a ray acoustic problem to be broken down into its Fourier components, solving for the propagation of one frequency component per ray. Another benefit is the simple use of frequency dependent boundary conditions.

Initial Conditions: Position

The ray release positions may be mesh based, from a user-specified grid, uniformly distributed, loaded from a file, or based on an analytic expression.

LOADING INITIAL COORDINATES FROM FILE

If the initial ray positions are loaded from a text file using the Release from Data File node, the initial positions may be scaled, rotated, translated, or some combination of these. Assuming $\mathbf{q}_0$ to be the loaded ray release position, $\mathbf{q}_0$ can be rotated using the rotation matrix A

$$\mathbf{q}_0' = \mathrm{A}\mathbf{q}_0$$

where A in 3D is given by

$$A = \begin{bmatrix} \cos\alpha\cos\gamma - \sin\alpha\cos\beta\sin\gamma & -\cos\alpha\sin\gamma - \sin\alpha\cos\beta\cos\gamma & \sin\alpha\sin\beta \\ \sin\alpha\cos\gamma + \cos\alpha\cos\beta\sin\gamma & -\sin\alpha\sin\gamma + \cos\alpha\cos\beta\cos\gamma & -\cos\alpha\sin\beta \\ \sin\beta\sin\gamma & \sin\beta\cos\gamma & \cos\beta \end{bmatrix}$$

and in 2D by

$$A = \begin{bmatrix} \cos\alpha & -\sin\alpha \\ \sin\alpha & \cos\alpha \end{bmatrix}$$

In 3D, α, β, and γ are the Euler angles:

- First α rotates the position about the space-fixed z-axis.
- Then β rotates the position about the x'-axis, that is, the body-fixed X-axis after rotation about the z-axis.
- Finally γ rotates the position about the z''-axis, that is, the body-fixed Z-axis after rotation about the z-axis and x'-axis.

In 2D, α is the angle of counterclockwise rotation about the origin.

All components of the ray release position vector are multiplied by the **Scale factor**,

$$\mathbf{q}_0'' = R\mathbf{q}_0'$$

Finally, the initial positions may be translated by a displacement vector $\Delta\mathbf{q}$,

$$\mathbf{q}_0''' = \mathbf{q}_0'' + \Delta\mathbf{q}$$

The primes indicate that translation is applied after any rotation or dilation (scaling). However, because the dilation applies equally to all vector components, the dilation and rotation commute with each other, so their relative order does not matter.

Initial Conditions: Direction

There are several options available for specifying the initial direction of rays. These options work by assigning values to the degrees of freedom corresponding to the wave vector $\mathbf{k}$ (SI unit: rad/m) of each ray.

EXPRESSION

The default is to enter an expression for the **Ray direction vector** $\mathbf{L}_0$ (dimensionless). This vector need not have a magnitude of unity because it is always normalized automatically. The initial wave vector is

$$\mathbf{k} = \frac{\omega \mathbf{L}_0}{c|\mathbf{L}_0|}$$

where ω (SI unit: rad/s) is the angular frequency and c (SI unit: m/s) is the speed of sound in the medium.

LOADING INITIAL DIRECTION FROM FILE

If the initial ray positions are loaded from a text file, then optionally, the initial ray direction can also be loaded from the same file. If the direction is loaded from the file and a rotation has been applied to the ray release positions, then optionally the same rotation can be applied to the initial direction.

SPHERICAL

When **Spherical** is selected, the initial wave vectors are sampled from a distribution in wave vector space at each release point. The number of rays released from each point is usually equal to the specified value N_w (dimensionless), although it may be larger if the initial values of any auxiliary dependent variables are also sampled from a distribution.

In 2D, the initial wave vector components are

$$k_x = \frac{\omega}{c}\cos\theta$$

$$k_y = \frac{\omega}{c}\sin\theta$$

where θ goes from 0 to 2π in N_w steps. In 3D, the initial wave vector components are sampled according to the expressions

$$k_x = \frac{\omega}{c}\sin\theta\cos\varphi$$

$$k_y = \frac{\omega}{c}\sin\theta\sin\varphi$$

$$k_z = \frac{\omega}{c}\cos\theta$$

The azimuthal angle φ is uniformly distributed from 0 to 2π. The polar angle θ is sampled from the interval $[0, \pi]$ with probability density proportional to $\sin\theta$. The polar angle is arbitrarily chosen as the angle that the initial wave vector makes with the positive z-axis, but any direction could be chosen, because the sphere is isotropic. Therefore, each ray subtends approximately the same solid angle in wave vector space.

HEMISPHERICAL

The **Hemispherical** option is the same as the **Spherical** option, except that in 2D θ goes from 0 to π and in 3D θ goes from 0 to $\pi/2$. The angle is measured from the direction given by the **Hemisphere axis** setting.

CONICAL

The **Conical** option is the same as the **Spherical** option, except that θ goes from 0 to α. The angle is measured from the direction given by the **Cone axis** setting.

LAMBERTIAN

The **Lambertian** option releases rays within a hemisphere in 3D, but the probability distribution function is different from that of the **Hemispherical** option. Recall that for an isotropic hemispherical distribution the polar angle θ has a probability density proportional to $\sin\theta$; for the Lambertian distribution the probability density is instead proportional to $\sin\theta\cos\theta$. Because of this extra cosine term, distributions following this probability density are said to follow Lambert's cosine law.

Material Discontinuity Theory

At a Material Discontinuity between two different media, the wave vector is reinitialized using Snell's law. First, the angle of incidence θ_i is computed:

$$\theta_i = \operatorname{acos}\left(\frac{\mathbf{n}_i \cdot \mathbf{n}_s}{|\mathbf{n}_i||\mathbf{n}_s|}\right)$$

where $\mathbf{n}_i$ is a unit vector in the direction of the incident ray and $\mathbf{n}_s$ is a unit vector normal to the material discontinuity.

At a boundary between two isotropic, nonabsorbing media, the refracted ray propagates in the direction $\mathbf{n}_t$ given by the following relations:

$$\mathbf{n}_t = \eta\mathbf{n}_i + \gamma\mathbf{n}_s$$

$$\gamma = -\eta\cos\theta_i + \cos\theta_t$$

$$\eta = \frac{c_2}{c_1}$$

$$\theta_t = \operatorname{asin}(\eta\sin\theta_i)$$

where the ray propagates from the medium with speed of sound c_1 into the medium with speed of sound c_2.

If the maximum number of secondary rays has not yet been released, a reflected ray is also released at the material discontinuity with the initial direction

$$\mathbf{n}_r = \mathbf{n}_i - 2\mathbf{n}_s \cos\theta_i$$

If the ray intensity is computed, the release of secondary rays may be suppressed when the intensity of the reflected ray is less than the user-defined **Threshold intensity** I_{th} (SI unit: W/m^2). Similarly, when ray power is computed, a **Threshold power** Q_{th} (SI unit: W) can be specified.

Intensity and Wavefront Curvature

The following describes the algorithm used to compute the ray intensity when **Intensity computation** is set to **Compute intensity** or **Compute intensity and power** in the physics interface **Intensity Computation** section. In this algorithm, each ray is treated as a propagating wavefront subtending a small solid angle. The algorithm is only valid when the rays propagate in homogeneous media. The following auxiliary dependent variables are defined for each ray:

In 3D:

- The initial ray intensity I_0, which is reinitialized at material discontinuities and walls.
- The integral A of the attenuation coefficient along the ray path.
- Two initial principal radii of curvature, $r_{1,i}$ and $r_{2,i}$.
- Two principal radii of curvature, r_1 and r_2, which represent the maximum and minimum radii of curvature of the intersection of the wavefront with an arbitrary plane.
- Three components of a unit vector $\mathbf{e}_1$ in the direction corresponding to the first principal radius of curvature. This information is used to reinitialize the principal radii of curvature at curved boundaries.

In 2D:

- The initial ray intensity I_0.
- The integral A of the attenuation coefficient along the ray path.

- One initial principal radius of curvature, $r_{1,i}$.
- One radius of curvature r_1 of the wavefront. All wavefronts are assumed to be converging or diverging cylindrical waves, so it is not necessary to define a second radius of curvature.

A different algorithm is used to compute the intensity of each ray if the **Intensity computation** is instead set to **Compute intensity in graded media** or **Compute intensity and power in graded media**. This algorithm accounts for the effect of graded media on the ray intensity, but it is also slower and less accurate for homogeneous media. The following auxiliary dependent variables are defined for each ray:

In 3D:

- The intensity help variable Γ.
- The integral A of the attenuation coefficient along the ray path.
- Two principal curvature calculation help variables α_1 and α_2 and the rotation angle ϕ, which indicates the orientation of the principal curvature directions. An additional help variable is used internally to detect poles in the local coordinate system definition and to redefine the local coordinate system accordingly.

In 2D:

- The intensity help variable Γ.
- The integral A of the attenuation coefficient along the ray path.
- One principal curvature calculation help variable α_1.

For the options **Compute Power**, **Compute intensity and power**, and **Compute intensity and power in graded media**, an additional auxiliary dependent variable is defined to indicate the total power transmitted by each ray. Like intensity, power is affected by absorbing media and by reflection or refraction at boundaries. Unlike intensity, power does not change due to the focusing or divergence of a ray bundle.

PRINCIPAL RADII OF CURVATURE

In 3D models, each ray is treated as a wavefront for which two principal radii of curvature, r_1 and r_2, are defined. In addition, the values of the two principal radii of curvature are stored as $r_{1,i}$ and $r_{2,i}$ whenever the ray reaches a boundary.

Within domains, the equations

$$\frac{dr_1}{ds} = -1 \qquad \frac{dr_2}{ds} = -1$$

are solved. Negative radii of curvature indicate that the wavefront is expanding as the ray propagates, while positive radii of curvature indicate that the wavefront is converging. A continuous locus of points at which either of the principal radii of curvature equals zero is called a caustic surface. The unit vector $\mathbf{e}_1$ is defined so that r_1 is the radius of curvature of the intersection of the wavefront with the plane tangent to $\mathbf{e}_1$ and the wave vector $\mathbf{k}$. Similarly, if a vector $\mathbf{e}_2$ is defined so that

$$\mathbf{e}_2 = \frac{\mathbf{k} \times \mathbf{e}_1}{|\mathbf{k} \times \mathbf{e}_1|}$$

Then r_2 is the radius of curvature of the intersection of the wavefront with a plane tangent to $\mathbf{e}_2$ and $\mathbf{k}$. The components of $\mathbf{e}_1$ are stored as auxiliary dependent variables for each ray. The components of $\mathbf{e}_2$ can then be derived from $\mathbf{e}_1$ and $\mathbf{k}$ at any time.

The principal radii of curvature are reinitialized at material discontinuities, and the orientation of the unit vectors $\mathbf{e}_1$ and $\mathbf{e}_2$ may change. If the unit vector in the direction of the incident ray $\mathbf{n}_\mathrm{i}$ is not parallel to the surface normal $\mathbf{n}_\mathrm{s}$, then it is possible to define a unique tangent plane, called the plane of incidence, that contains the incident, reflected, and refracted rays. The unit vector normal to this plane, $\mathbf{u}_0$, is defined as

$$\mathbf{u}_0 = \frac{\mathbf{n}_\mathrm{i} \times \mathbf{n}_\mathrm{s}}{|\mathbf{n}_\mathrm{i} \times \mathbf{n}_\mathrm{s}|}$$

If the ray is normal to the surface, then the incident ray is parallel to the refracted ray and antiparallel to the reflected ray, and $\mathbf{u}_0$ can be any arbitrary unit vector orthogonal to $\mathbf{n}_\mathrm{i}$. In addition to the unit vector normal to the plane of incidence, the following unit vectors tangent to the incident wavefront, refracted wavefront, reflected wavefront, and surface normal are defined:

$$\begin{aligned}\mathbf{u}_\mathrm{i} &= \mathbf{n}_\mathrm{i} \times \mathbf{u}_0\\ \mathbf{u}_\mathrm{t} &= \mathbf{n}_\mathrm{t} \times \mathbf{u}_0\\ \mathbf{u}_\mathrm{r} &= \mathbf{n}_\mathrm{r} \times \mathbf{u}_0\\ \mathbf{u}_\mathrm{s} &= \mathbf{n}_\mathrm{s} \times \mathbf{u}_0\end{aligned}$$

where the subscripts i, t, and r denote the incident, refracted, and reflected wavefronts, respectively. For a wavefront propagating in a direction $\mathbf{n}$, with principal curvatures $k_1 = 1/r_1$ and $k_2 = 1/r_2$ defined for directions $\mathbf{e}_1$ and $\mathbf{e}_2$, respectively, the principal curvatures in two other orthogonal directions $\mathbf{e}_1'$ and $\mathbf{e}_2'$ (both orthogonal to $\mathbf{n}$) are

$$
\begin{aligned}
k_1' &= k_1\cos^2\theta + k_2\sin^2\theta \\
k_2' &= k_1\sin^2\theta + k_2\cos^2\theta \\
k_{12}' &= (k_1 - k_2)\cos\theta\sin\theta
\end{aligned}
\tag{8-1}
$$

where θ is the angle of rotation about $\mathbf{n}$ which transforms the vectors $\mathbf{e}_1$ and $\mathbf{e}_2$ to $\mathbf{e}_1'$ and $\mathbf{e}_2'$, respectively. Because k_1' and k_2' are not principal curvatures, it is necessary to include off-diagonal elements of the wavefront curvature tensor equal to k_{12}'.

The following algorithm is used to reinitialize the principal radii of curvature of the wavefront and their orientations. The reinitialization of the curvature variables follows the method of Stavroudis in Ref. 2.

1. Given $\mathbf{n}_i$ and $\mathbf{n}_s$, compute unit vectors in the directions of the reflected and refracted rays, $\mathbf{n}_r$ and $\mathbf{n}_t$.
2. Compute the vectors $\mathbf{u}_0$, $\mathbf{u}_i$, $\mathbf{u}_t$, $\mathbf{u}_r$, and $\mathbf{u}_s$.
3. Compute the angle of rotation $\theta^{(i)}$ needed to transform the local coordinate system with axes parallel to $\mathbf{e}_1$, $\mathbf{e}_2$, and $\mathbf{n}_i$ to a local coordinate system with axes parallel to $\mathbf{u}_0$, $\mathbf{u}_i$, and $\mathbf{n}_i$. (Note: the superscript $^{(i)}$ is used to distinguish this coordinate system rotation angle from the angle of incidence of the ray, often denoted θ_i.)
4. Transform the principal curvatures to the local coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_i$, and $\mathbf{n}_i$ by using Equation 8-1, substituting $\theta^{(i)}$ for θ.
5. Given the two principal curvatures of the surface, $k_{1,s}$ and $k_{2,s}$ with directions $\mathbf{e}_{1,s}$ and $\mathbf{e}_{2,s}$, transform the curvature variables to a local coordinate system with axes parallel to $\mathbf{u}_0$, $\mathbf{u}_s$, and $\mathbf{n}_s$. Let the new curvature variables be denoted by $k_{1,s}'$, $k_{2,s}'$, and $k_{12,s}'$.
6. The curvatures of the wavefront and the surface have now been defined in coordinate systems that share the axis $\mathbf{u}_0$ and only differ by a rotation by the angle of incidence θ_i about $\mathbf{u}_0$. Defining the variables η and γ as in Material Discontinuity Theory, compute the curvature variables of the refracted ray in a coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_t$, and $\mathbf{n}_t$ using the equations

$$
\begin{aligned}
k_{1,t}' &= \eta k_1' + \gamma k_{1,s}' \\
k_{12,t}' &= \frac{\eta\cos\theta_i}{\cos\theta_t}k_{12}' + \frac{\gamma}{\cos\theta_t}k_{12,s}' \\
k_{2,t}' &= \frac{\eta\cos^2\theta_i}{\cos^2\theta_t}k_2' + \frac{\gamma}{\cos^2\theta_t}k_{2,s}'
\end{aligned}
$$

7 Obtain the principal curvatures of the refracted ray by rotating the coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_t$, and $\mathbf{n}_t$ by an angle $\theta^{(t)}$ about $\mathbf{n}_t$. The angle $\theta^{(t)}$ is defined as

$$\theta^{(t)} = \frac{1}{2}\text{atan}\left(\frac{2k_{12,t}'}{k_{1,t}' - k_{2,t}'}\right)$$

8 Reinitialize the principal curvatures of the refracted ray:

$$k_{1,t} = k_{1,t}'\cos^2\theta^{(t)} + 2k_{12,t}'\cos\theta^{(t)}\sin\theta^{(t)} + k_{2,t}'\sin^2\theta^{(t)}$$
$$k_{2,t} = k_{1,t}'\sin^2\theta^{(t)} - 2k_{12,t}'\cos\theta^{(t)}\sin\theta^{(t)} + k_{2,t}'\cos^2\theta^{(t)}$$

9 Invert the principal curvatures to obtain the principal radii of curvature of the refracted ray. Rotate $\mathbf{u}_0$ about $\mathbf{n}_t$ by the angle $\theta^{(t)}$ to obtain the reinitialized principal curvature direction $\mathbf{e}_{1,t}$.

10 If a reflected ray is released, compute the curvature variables of the reflected ray in a coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_r$, and $\mathbf{n}_r$ using the equations

$$k_{1,r}' = k_1' - 2k_{1,s}'\cos\theta_i$$

$$k_{12,r}' = -k_{12}' + 2k_{12,s}'$$

$$k_{2,r}' = k_2' - \frac{2}{\cos\theta_i}k_{2,s}'$$

11 Obtain the principal curvatures of the reflected ray by rotating the coordinate system defined by $\mathbf{u}_0$, $\mathbf{u}_r$, and $\mathbf{n}_r$ by an angle $\theta^{(r)}$ about $\mathbf{n}_r$. The angle $\theta^{(r)}$ is defined as

$$\theta^{(r)} = \frac{1}{2}\text{atan}\left(\frac{2k_{12,r}'}{k_{2,r}' - k_{1,r}'}\right)$$

12 Initialize the principal curvatures of the reflected ray:

$$k_{1,r} = k_{1,r}'\cos^2\theta^{(r)} + 2k_{12,r}'\cos\theta^{(r)}\sin\theta^{(r)} + k_{2,r}'\sin^2\theta^{(r)}$$
$$k_{2,r} = k_{1,r}'\sin^2\theta^{(r)} - 2k_{12,r}'\cos\theta^{(r)}\sin\theta^{(r)} + k_{2,r}'\cos^2\theta^{(r)}$$

13 Invert the principal curvatures to obtain the principal radii of curvature of the reflected ray. Rotate $\mathbf{u}_0$ about $\mathbf{n}_r$ by the angle $\theta^{(r)}$ to obtain the initialized principal curvature direction $\mathbf{e}_{1,r}$.

In 2D components, only one principal radius of curvature is computed; each ray is treated as a cylindrical wave. In addition, no auxiliary degrees of freedom for the

principal curvature direction are required because the out-of-plane direction can always be treated as one of the axes of the local coordinate system that defines the orientation of the wavefront.

In 2D axisymmetric model components, radii of curvature are computed for the in-plane direction (that is, the *rz*-plane) as well as the out-of-plane or azimuthal directions. During ray-boundary interactions, these radii of curvature are reinitialized as if the ray interacted with a 3D surface of revolution, which may have finite radii of curvature in both the in-plane and out-of-plane directions.

INTENSITY CALCULATION

The value of the ray intensity is stored as the auxiliary dependent variable I_0 when a ray is released. At any point along the ray's trajectory, the intensity is equal to

$$I = I_0 \frac{r_{1,\mathrm{i}} r_{2,\mathrm{i}}}{r_1 r_2}$$

In 2D, the second principal radius of curvature r_2 is treated as an arbitrarily large distance that remains constant for each ray.

Intensity and Phase Reinitialization

At a material discontinuity, the intensity of the reflected and refracted rays can be computed using the material properties of the media in the adjacent domains:

$$I_\mathrm{r} = I|R|^2$$

$$I_\mathrm{t} = I(1 - |R|^2)$$

$$R = \frac{Z_2 \cos\theta_\mathrm{i} - Z_1 \cos\theta_\mathrm{t}}{Z_2 \cos\theta_\mathrm{i} + Z_1 \cos\theta_\mathrm{t}} \tag{8-2}$$

$$Z = \rho c_\mathrm{c}$$

where ρ is the density of the medium and c_c is the complex speed of sound,

$$c_\mathrm{c} = \frac{c}{1 - i\frac{c}{\omega}\alpha}$$

where

- c (SI unit: m/s) is the real speed of sound in the medium,
- ω (SI unit: rad/s) is the angular frequency of the acoustic ray, and
- α (SI unit: 1/m) is the attenuation coefficient.

The subscripts 1 and 2 denote the sides of the boundary into which the reflected and refracted ray propagate, respectively; see Ref. 4.

At a Wall, the reflection coefficient R can be defined explicitly or in terms of the absorption coefficient α or characteristic impedance of absorber Z_1. In terms of the absorption coefficient,

$$R = \sqrt{1-\alpha}$$

Where the principal square root is used for purposes of phase reinitialization. In terms of the characteristic impedance of absorber,

$$R = \frac{Z_1 \cos\theta / Z_0 - 1}{Z_1 \cos\theta / Z_0 + 1}$$

$$Z_0 = \rho c_c$$

The **Wall** feature also includes built-in options to compute the reflection coefficient for fluid-fluid interfaces, fluid-solid interfaces, and a fluid layer adjacent to a semi-infinite fluid domain. When treating the **Wall** boundary as a **Fluid-fluid interface**, the reflection coefficient is computed using Equation 8-2 as if the wall were a material discontinuity where the fluid properties of the adjacent medium are specified, except that no refracted ray is produced.

If the **Wall** boundary is modeled as a **Fluid-solid interface**, the reflection coefficient is instead defined as

$$R = \frac{Z_{\text{tot}} - \dfrac{Z_0}{\cos\theta_{\text{i}}}}{Z_{\text{tot}} + \dfrac{Z_0}{\cos\theta_{\text{i}}}}$$

where

$$Z_{\text{tot}} = \frac{Z_{\text{p, 1}}}{\cos\theta_{\text{p, t}}}\cos^2(2\theta_{\text{s, }t}) + \frac{Z_{\text{s, 1}}}{\cos\theta_{\text{s, t}}}\sin^2(2\theta_{\text{s, t}})$$

where the subscripts p and s refer to the propagation of compressional and shear waves in the adjacent solid domain, respectively. For example, $\theta_{p,t}$ is the refraction angle computed using the compressional complex speed of sound in the adjacent solid domain.

If the **Wall** boundary is modeled as a **Layered fluid half-space**, the boundary is treated as a thin layer of one fluid backed by a semi-infinite domain of a second fluid. The reflection coefficient is

$$R = \frac{\zeta_1(\zeta_2 - \zeta_0) - i(\zeta_1^2 - \zeta_0\zeta_2)\tan\phi}{\zeta_1(\zeta_2 + \zeta_0) - i(\zeta_1^2 - \zeta_0\zeta_2)\tan\phi}$$

$$\zeta_0 = \frac{Z_0}{\cos\theta_i}$$

$$\zeta_1 = \frac{Z_1}{\cos\theta_{t,1}}$$

$$\zeta_2 = \frac{Z_2}{\cos\theta_{t,2}}$$

where the subscripts 1 and 2 refer to the properties of the thin layer and the semi-infinite fluid domain, respectively. For example, $\theta_{t,1}$ is the angle of refraction in the thin layer. The angle ϕ is the phase delay in the thin layer,

$$\phi = \frac{\omega}{c_1} h_1 \cos\theta_{t,1}$$

where h_1 (SI unit: m) is the layer thickness.

If the **Rayleigh roughness** model is used, the reflected intensity is multiplied by an additional factor to account for surface roughness:

$$I_r = I\left|R\exp(-0{,}5\Gamma^2)\right|^2$$

$$\Gamma = 2k\sigma\cos\theta_i$$

where k (SI unit: rad/m) is the wave vector magnitude of the ray and σ (SI unit: m) is the RMS roughness of the surface.

If the phase of the ray is also computed, the phase shift of the reflected ray at a boundary is

$$\Delta\Psi_r = \arg(R)$$

At material discontinuities, the phase shift of the refracted ray is given by the argument of the transmission coefficient T.

Wavefront Curvature Calculation in Graded Media

It is possible to compute the ray intensity by changing **Intensity computation** to **Compute intensity** or **Compute intensity in graded media** in the Settings window for the Ray Acoustics interface. The options **Compute intensity and power** and **Compute intensity and power in graded media** can also be used to compute intensity, with the only difference being that these options define an additional auxiliary dependent variable for the total power transferred by the ray. The setting **Compute intensity** is more robust and accurate than **Compute intensity in graded media**, but is only applicable to homogeneous media. The setting **Compute intensity in graded media** can be used for both homogeneous and graded media, but it introduces more numerical error than **Compute intensity**.

ASSUMPTIONS FOR COMPUTING INTENSITY IN HOMOGENEOUS MEDIA

When **Compute intensity** is selected, the ray intensity is computed along each ray path using the following assumptions:

1. The wavefront represented by each ray subtends a constant plane angle (2D) or solid angle (3D), which can change only when rays are reflected or refracted at boundaries.
2. Given the radii of curvature at two positions within the same domain and the intensity at one of these two positions, the intensity at the other position can be computed using the relationship

$$I = I_0 \frac{r_{1,0} r_{2,0}}{r_1 r_2}$$

3. Except when rays are reflected or refracted at boundaries, the principal radii of curvature change at a constant rate:

$$\frac{dr_1}{ds} = \frac{dr_2}{ds} = -1$$

4. The principal curvature directions do not change, except when rays are reflected or refracted at boundaries.

These assumptions are not valid in graded media, in which the speed of sound changes continuously as a function of position. It is possible for the angle between two different

rays to change as they propagate through the medium, so the solid angle subtended by the wavefront is no longer constant. As a result, the ray intensity cannot be expressed using the ratio of initial and final principal radii of curvature.

CURVATURE TENSOR DEFINITION IN GRADED MEDIA

The calculation of ray intensity in graded media is based on the concept of a curvature tensor $\mathbf{K}$, defined in terms of the principal curvatures κ_1 and κ_2 and the corresponding principal curvature directions $\mathbf{e}_1$ and $\mathbf{e}_2$:

$$\mathbf{K} = \kappa_1 \mathbf{e}_1 \mathbf{e}_1^{\mathrm{T}} + \kappa_2 \mathbf{e}_2 \mathbf{e}_2^{\mathrm{T}}$$

Because $\mathbf{e}_1$ and $\mathbf{e}_2$ are orthogonal, it follows that κ_1 and κ_2 are eigenvalues of $\mathbf{K}$. It also follows that $\mathbf{K}$ is singular because there is no contribution that is orthogonal to both $\mathbf{e}_1$ and $\mathbf{e}_2$.

The signs of the principal curvatures are chosen so that positive curvature indicates that the wavefront is converging, whereas negative curvature indicates that the wavefront is diverging.

The selection of the coordinate system in which the curvature tensor is defined is crucial. It is convenient to describe the coordinate system so that two of the coordinate axes lie in the plane containing $\mathbf{e}_1$ and $\mathbf{e}_2$, since this reduces the number of nonzero terms in $\mathbf{K}$. Further reduction in the number of nonzero terms can be achieved if $\mathbf{K}$ can be defined in a coordinate system in which $\mathbf{e}_1$ and $\mathbf{e}_2$ are basis vectors. This is possible in 2D because one of the principal curvature directions is always parallel to the out-of-plane direction, but in 3D it is not feasible because the principal curvature directions can change as rays propagate through a graded medium.

In the most general 3D case, the curvature tensor is described using the following symbols, each of which corresponds to a different orthonormal basis:

- $\mathbf{K}_{\mathrm{X}}$: curvature tensor defined in the Cartesian coordinate system with basis vectors $\mathbf{x}$, $\mathbf{y}$, and $\mathbf{z}$.
- $\mathbf{K}_{\mathrm{W}}$: curvature tensor defined in the coordinate system consisting of the two principal curvature directions $\mathbf{e}_1$ and $\mathbf{e}_2$ and the direction of propagation $\mathbf{t}$.
- $\mathbf{K}_{\mathrm{S}}$: curvature tensor defined in a coordinate system in which one basis vector is the direction of propagation $\mathbf{t}$. The other two basis vectors $\mathbf{s}_1$ and $\mathbf{s}_2$ can be determined arbitrarily, as long as they are orthogonal to each other and to $\mathbf{t}$, such that $\mathbf{s}_1 \times \mathbf{s}_2 = \mathbf{t}$. It is more convenient to operate in this coordinate system if the basis vectors can be expressed strictly in terms of $\mathbf{x}$, $\mathbf{y}$, $\mathbf{z}$, and $\mathbf{t}$.

The basis vectors $\mathbf{s}_1$ and $\mathbf{s}_2$ are defined as follows:

$$\mathbf{s}_1 = \frac{\mathbf{t} \times \mathbf{z}}{|\mathbf{t} \times \mathbf{z}|}$$

$$\mathbf{s}_2 = \mathbf{t} \times \mathbf{s}_1$$

To avoid poles in the definition of the basis vectors, the following alternative definitions are used when the rays propagate nearly parallel to the *z*-axis:

$$\mathbf{s}_1 = \frac{\mathbf{t} \times \mathbf{x}}{|\mathbf{t} \times \mathbf{x}|}$$

$$\mathbf{s}_2 = \mathbf{t} \times \mathbf{s}_1$$

The relationship between $\mathbf{s}_1$, $\mathbf{s}_2$, $\mathbf{e}_1$, and $\mathbf{e}_2$ is given by

$$\mathbf{e}_1 = \mathbf{s}_1 \cos\phi - \mathbf{s}_2 \sin\phi$$

$$\mathbf{e}_2 = \mathbf{s}_1 \sin\phi + \mathbf{s}_2 \cos\phi$$

where the rotation angle ϕ is an auxiliary dependent variable that is stored for each ray.

The relationship between $\mathbf{K}_S$ and $\mathbf{K}_W$ is given by

$$\mathbf{K}_S = \mathbf{Q}_0 \mathbf{K}_W \mathbf{Q}_0^T$$

where $\mathbf{Q}_0$ is the rotation matrix:

$$\mathbf{Q}_0 = \begin{bmatrix} \mathbf{s}_1 \cdot \mathbf{e}_1 & \mathbf{s}_1 \cdot \mathbf{e}_2 & \mathbf{s}_1 \cdot \mathbf{t} \\ \mathbf{s}_2 \cdot \mathbf{e}_1 & \mathbf{s}_2 \cdot \mathbf{e}_2 & \mathbf{s}_2 \cdot \mathbf{t} \\ \mathbf{t} \cdot \mathbf{e}_1 & \mathbf{t} \cdot \mathbf{e}_2 & \mathbf{t} \cdot \mathbf{t} \end{bmatrix} = \begin{bmatrix} \cos\phi & \sin\phi & 0 \\ -\sin\phi & \cos\phi & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

The relationship between $\mathbf{K}_X$ and $\mathbf{K}_S$ is given by

$$\mathbf{K}_X = \mathbf{Q} \mathbf{K}_S \mathbf{Q}^T$$

where $\mathbf{Q}$ is the rotation matrix:

$$\mathbf{Q} = \begin{bmatrix} \mathbf{x} \cdot \mathbf{s}_1 & \mathbf{x} \cdot \mathbf{s}_2 & \mathbf{x} \cdot \mathbf{t} \\ \mathbf{y} \cdot \mathbf{s}_1 & \mathbf{y} \cdot \mathbf{s}_2 & \mathbf{y} \cdot \mathbf{t} \\ \mathbf{z} \cdot \mathbf{s}_1 & \mathbf{z} \cdot \mathbf{s}_2 & \mathbf{z} \cdot \mathbf{t} \end{bmatrix}$$

In 2D, one of the principal curvature directions is always known, so it is possible to define $\mathbf{s}_1$ and $\mathbf{s}_2$ so that they coincide with $\mathbf{e}_1$ and $\mathbf{e}_2$. The rotation matrix $\mathbf{Q}_0$ is the identity, and the auxiliary dependent variable for the angle ϕ can be omitted.

DERIVATIVES OF THE CURVATURE TENSOR

Following Ref. 3, the derivative of the curvature tensor along the ray trajectory in a graded medium is given by the expression

$$\frac{d\mathbf{K}_\mathrm{S}}{ds} = \sum_{i=1}^{5} \mathbf{M}_i$$

where the terms $\mathbf{M}_i$ correspond to the different ways in which the curvature can depend on the medium properties. They are defined using the following expressions:

$$\mathbf{M}_1 = \mathbf{K}_\mathrm{S} \cdot \mathbf{K}_\mathrm{S}$$

$$\mathbf{M}_2 = -c\left(\mathbf{t} \cdot \nabla_\mathrm{S}\left(\frac{1}{c}\right)\right)\mathbf{K}_\mathrm{S}$$

$$\mathbf{M}_3 = -c\Pi\left(\nabla_\mathrm{S} \otimes \nabla_\mathrm{S}\left(\frac{1}{c}\right)\right)$$

$$\mathbf{M}_4 = 2c^2\Pi\left(\nabla_\mathrm{S}\left(\frac{1}{c}\right) \otimes \nabla_\mathrm{S}\left(\frac{1}{c}\right)\right)\Pi$$

$$\mathbf{M}_5 = -c\left[\left(\mathbf{K}\nabla_\mathrm{S}\left(\frac{1}{c}\right)\right) \otimes \mathbf{t} + \mathbf{t} \otimes \nabla_\mathrm{S}\left(\frac{1}{c}\right)\mathbf{K}\right]$$

where Π is the projection matrix:

$$\Pi = \mathbf{I} - (\mathbf{t} \otimes \mathbf{t})$$

The gradient operator ∇_S consists of derivatives that are taken with respect to the local coordinates with basis vectors $\mathbf{s}_1$, $\mathbf{s}_2$, and $\mathbf{t}$.

DERIVATIVES OF THE PRINCIPAL CURVATURES

By application of the chain rule, the derivatives of the nonzero elements of $\mathbf{K}_S$ can be expressed in terms of the principal curvatures that occupy the diagonal elements of $\mathbf{K}_W$ and the rotation angle ϕ:

$$\frac{\partial \kappa_1}{\partial s} = \frac{\partial K_{\mathrm{S},11}}{\partial s}\cos^2\phi - 2\frac{\partial K_{\mathrm{S},12}}{\partial s}\sin\phi\cos\phi + \frac{\partial K_{\mathrm{S},22}}{\partial s}\sin^2\phi$$

$$\frac{\partial \kappa_2}{\partial s} = \frac{\partial K_{S,11}}{\partial s}\sin^2\phi + 2\frac{\partial K_{S,12}}{\partial s}\sin\phi\cos\phi + \frac{\partial K_{S,22}}{\partial s}\cos^2\phi$$

$$\frac{\partial \phi}{\partial s} = \frac{1}{\kappa_2 - \kappa_1}\left[\frac{\partial K_{S,12}}{\partial s}\cos(2\phi) + \frac{1}{2}\left(\frac{\partial K_{S,11}}{\partial s} - \frac{\partial K_{S,22}}{\partial s}\right)\sin(2\phi)\right]$$

NUMERICAL STABILIZATION

The principal curvatures κ_1 and κ_2 are not ideal choices for the auxiliary dependent variables stored by each ray because their values can become arbitrarily large as rays approach caustics. Similarly, the principal radii of curvature r_1 and r_2 are not ideal choices because their values can become arbitrarily large when a diverging wavefront begins to converge while propagating through a graded medium.

Instead, the auxiliary dependent variables stored by each ray are the help variables α_1 and α_2. In 2D, only one help variable is allocated because the out-of-plane principal radius of curvature is assumed to be infinite. The principal curvatures are expressed in terms of the help variables using the expression

$$\kappa_i = \kappa_0 \tan\alpha$$

where $\kappa_0 = 1$ 1/m and $i \in [1, 2]$. The derivatives are then related by the expression

$$\frac{d\alpha_i}{ds} = \frac{1}{\kappa_0}\frac{d\kappa_i}{ds}\cos^2\alpha_i$$

Similarly, the intensity may become infinitely large at caustics, and its reciprocal becomes infinitely large as rays undergo attenuation, so a help variable Γ is used to represent the intensity of the ray:

$$I = I_0 \tan\Gamma$$

where $I_0 = 1$ W/m^2. The relationship between the ray intensity help variable and the principal curvatures is

$$\frac{d\Gamma}{ds} = \frac{1}{2}\sin(2\Gamma)[\kappa_1 + \kappa_2 - 2\alpha]$$

where α is the attenuation coefficient of the medium.

Attenuation Within Domains

Rays can gradually lose energy as they propagate through absorbing media. For a plane wave propagating through an absorbing medium with attenuation coefficient α (SI unit: 1/m), the intensity decreases exponentially:

$$\frac{dI}{ds} = -2\alpha I \tag{8-3}$$

The Ray Acoustics interface defines an additional dependent variable A (dimensionless) for the path integral of the attenuation coefficient,

$$\frac{dI_0}{ds} = 0 \tag{8-4}$$

$$I = I_0 \exp(A)$$

$$\frac{dA}{ds} = -2\alpha$$

Despite requiring an extra degree of freedom per ray, the advantage of Equation 8-4 over Equation 8-3 is that the dependent variables in Equation 8-4 all vary linearly within a homogeneous medium. This makes Equation 8-4 much less prone to numerical stiffness; that is, it remains more accurate when the solver takes long time steps, compared to Equation 8-3, potentially reducing solution time and improving accuracy.

The attenuation coefficient is controlled by the option selected from the **Fluid model** list in the **Pressure Acoustics Model** section of the Medium Properties.

If **Linear elastic** is selected, no attenuation occurs.

If **Linear elastic with attenuation** is selected, the attenuation factor is user defined. Note that the attenuation coefficient α is a pressure amplitude attenuation coefficient in nepers per meter (Np/m). Alternatively, you can specify the amplitude attenuation coefficient in decibels per meter α' or decibels per wavelength $\alpha^{(\lambda)}$. Another option is to specify the intensity attenuation coefficient m directly. The relationships between these definitions of the attenuation coefficient may be summarized as follows:

$$\alpha = \frac{m}{2} = \frac{\alpha'}{20\log_{10}(e)} = \frac{\alpha^{(\lambda)}}{20\log_{10}(e)\lambda}$$

If **Thermally conducting and viscous** is selected, the attenuation factor is defined in terms of the medium properties:

$$\alpha = \frac{\omega^2 b}{2\rho c^3}$$

$$b \equiv \frac{4}{3}\mu + \mu_B + \frac{(\gamma - 1)k}{C_p}$$

where

- c (SI unit: m/s) is the speed of sound in the medium,
- ρ (SI unit: kg/m^3) is the density.
- C_p (SI unit: J/(kg·K)) is the heat capacity at constant pressure.
- γ (dimensionless) is the ratio of specific heats.
- k (SI unit: W/(m·K)) is the thermal conductivity.
- μ (SI unit: Pa·s) is the dynamic viscosity.
- μ_B (SI unit: Pa·s) is the bulk viscosity.

Built-in models for attenuation in the ocean and atmosphere are also available. The theory for these features can be found in the Attenuation and Loss Models section in the Acoustic Properties of Fluids chapter.

Ray Termination Theory

The Ray Termination feature can annihilate rays when their intensity or power decreases below a specified threshold.

If the ray intensity is computed, then the threshold ray intensity I_{th} (SI unit: W/m^2) can be specified. Then the termination criterion is

$$I < I_{th}$$

If the ray power is computed, then the threshold ray power Q_{th} (SI unit: W) can be specified. Then the termination criterion is

$$Q < Q_{th}$$

It is also possible to specify termination criteria based on both intensity and power; in this case, the rays terminate if either criterion is satisfied.

Whenever possible, the **Ray Termination** feature attempts to stop rays at the exact instant when the ray intensity or power reaches a specified threshold value. A closed-form analytic solution for the exact stop time is available under the following circumstances:

- The rays represent planar wavefronts in an absorbing medium, or
- The rays represent spherical or ellipsoidal wavefronts in a nonabsorbing medium.

If neither of these special conditions is met, the ray can only terminate if the termination criterion is met at the beginning of a time step taken by the solver; the ray intensity or power at the instant the ray is stopped may be less than the threshold.

PLANAR WAVEFRONTS IN AN ABSORBING MEDIUM

Let us consider waves that are not reflected at a boundary. In this special case, the principal wavefront radii of curvature are so large that the ray intensity can only change significantly in an attenuating medium. The exact time t_s (SI unit: s) at which the ray intensity reaches the threshold value is

$$t_s = t_0 + \tau \log\left(\frac{I}{I_{th}}\right)$$

where

- t_0 (SI unit: s) is the previous time step, reflection time, or refraction time,
- I (SI unit: W/m^2) is the intensity at t_0, and
- I_{th} (SI unit: W/m^2) is the threshold intensity.

If the termination criterion is based on the ray power Q (SI unit: W) instead of ray intensity, the analogous expression for the stop time is

$$t_s = t_0 + \tau \log\left(\frac{Q}{Q_{th}}\right)$$

The characteristic time for ray attenuation τ (SI unit: s) is

$$\tau = \frac{1}{2\alpha V_g}$$

where

- α (SI unit: 1/m) is the attenuation coefficient of the medium and
- V_g (SI unit: m/s) is the group velocity magnitude.

DIVERGING WAVEFRONTS IN A NONABSORBING MEDIUM

Now also consider a wave that is not reflected at a boundary. In a nonabsorbing medium in which the wavefront is nonplanar, the ray can only terminate due to the intensity decreasing below the threshold, since the power does not change. The time t_s (SI unit: s) at which the ray intensity reaches the threshold in a nonattenuating, homogeneous medium is

$$t_s = t_0 + \frac{1}{2V_g}\left(r_1 + r_2 + \sqrt{(r_1 - r_2)^2 + 4\frac{I}{I_{th}}|r_1 r_2|}\right)$$

where

- t_0 (SI unit: s) is the previous time step, reflection time, or refraction time,
- V_g (SI unit: m/s) is the group velocity magnitude,
- r_1 (SI unit: m) is the first principal radius of curvature of the wavefront at t_0,
- r_2 (SI unit: m) is the second principal radius of curvature of the wavefront at t_0,
- I (SI unit: W/m^2) is the intensity at t_0, and
- I_{th} (SI unit: W/m^2) is the threshold intensity.

In 2D models, or in 3D models where the wavefronts are cylindrical (one extremely large radius of curvature and one finite radius of curvature), the stop time is instead

$$t_s = t_0 + \frac{1}{V_g}\left(r_1 + \frac{I}{I_{th}}|r_1|\right)$$

where r_1 (SI unit: m) is the finite principal radius of curvature of the cylindrical wavefront.

Accumulator Theory: Domains

The Accumulator (Domain) node is used to transfer information from rays to the domains they occupy or pass through. Each **Accumulator** defines a variable, called the accumulated variable, in the selected domains. The accumulated variable is discretized using constant shape functions, so its value is uniform over every mesh element and may be discontinuous between adjacent mesh elements.

The name of the accumulated variable is specified in the **Accumulated variable name** field in the **Accumulator Settings** section of the Settings window. The default variable name, **rpd**, will be used in the remainder of this section when referring to the accumulated variable.

ACCUMULATOR TYPE

The options in the **Accumulator type** list are **Density** and **Count**. If **Density** is selected, the source term is divided by the area or volume of the mesh element when calculating each ray's contribution to the accumulated variable. If **Count** is selected, no division by the area or volume of the mesh element occurs.

The equations in the following section are valid for the **Density** type. The corresponding value of the accumulated variable for the **Count** type is

$$\mathrm{rpd}_{\mathrm{count}} = \mathrm{rpd}_{\mathrm{density}} \times V$$

where V is the mesh element volume (in 3D) or area (in 2D).

ACCUMULATION OVER ELEMENTS

When **Elements** is selected from the **Accumulate over** list, the value of the accumulated variable in a mesh element is the sum of the source terms R_i evaluated for all rays in that mesh element:

$$\mathrm{rpd} = \frac{1}{V}\sum_{i=1}^{N} R_i$$

where N is the total number of rays in the element and V is the area or volume of the mesh element. In other words, the contribution of each ray to the accumulated variable is distributed uniformly over the mesh element the ray is in, regardless of the ray's exact position within the element.

If **Elements and time** is selected from the **Accumulate over** list, then the sum of the source terms for rays in the mesh element is used to define the time derivative of the accumulated variable, rather than its instantaneous value:

$$\frac{\mathrm{d}(\mathrm{rpd})}{\mathrm{d}t} = \frac{1}{V}\sum_{i=1}^{N} R_i$$

Thus, the value of the accumulated variable depends on the time history of rays in the mesh element, instead of the instantaneous positions of the rays. As each ray propagates, it will leave behind a trail based on its contributions to the accumulated variables in mesh elements it has traversed. The algorithm for accumulating over time takes into account the fraction of a time step taken by the solver that the ray spends in each mesh element, even if it crosses between elements during the time step.

Accumulator Theory: Boundaries

The Accumulator (Boundary) feature transfers information from rays to the boundaries they hit or pass through. Each **Accumulator** defines a variable, called the accumulated variable, on the selected boundaries. The accumulated variable is discretized using constant shape functions, so its value if uniform over every mesh element and may be discontinuous between adjacent mesh elements.

The name of the accumulated variable is specified in the **Accumulated variable name** field in the **Accumulator Settings** section of the settings window. The default variable name, rpb, will be used in the remainder of this section when referring to the accumulated variable.

The options in the **Accumulator type** list are **Density** and **Count**. If **Density** is selected, the source term is divided by the surface area or length of the boundary mesh element when calculating each ray's contribution to the accumulated variable. If **Count** is selected, no division by the surface area or length of the boundary element occurs.

The equations in the following section are valid for the **Density** type. The corresponding value of the accumulated variable for the **Count** type is

$$\mathrm{rpb}_{\mathrm{count}} = \mathrm{rpb}_{\mathrm{density}} \times V$$

where V is the boundary element surface area (in 3D) or length (in 2D).

When **Rays in boundary elements** is selected from the **Accumulate over** list, the accumulated variable in a boundary element gets incremented by the source term R whenever a ray freezes or sticks to the boundary:

$$\mathrm{rpb}_{\mathrm{new}} = \mathrm{rpb} + \frac{R}{V}$$

where division by the mesh element area or length occurs because the accumulator is assumed to be of type **Density**. Thus the source term evaluated for an incident ray is uniformly distributed over the boundary element it freezes or sticks to.

If instead **Ray-wall interactions** is selected from the **Accumulate over** list, then the accumulated variable gets incremented regardless of what type of ray-wall interaction occurs. Thus, it is possible for the same ray to increment the accumulated variable in many different boundary elements, or even in the same element multiple times.

BUILT-IN GLOBAL VARIABLES

By default, the boundary **Accumulator** defines the following global variables:

TABLE 8-4: BUILT-IN GLOBAL VARIABLES FOR BOUNDARY ACCUMULATORS.

NAME	EXPRESSION
<scope>.<name>_ave	Average of accumulated variable
<scope>.<name>_int	Integral of accumulated variable
<scope>.<name>_max	Maximum of accumulated variable
<scope>.<name>_min	Minimum of accumulated variable
<scope>.<name>_sum	Sum of accumulated variable over elements

Here, <scope> includes the physics interface name and the names of the Accumulator and parent features. For example, the average of the accumulated variable over a boundary may be called rac.wall1.bacc1.rpb_ave, where rac is the name of the Ray Acoustics interface, wall1 is the name of the parent Wall node, bacc1 is the name of the Accumulator node, and rpb is the accumulated variable name. These variables are all available in the **Add/Replace Expression** menus during results evaluation.

These global variables are computed by defining a set of nonlocal couplings on the selection of the parent physics feature, such as the Wall feature to which the Accumulators are added. The following expressions for the global variables are used.

TABLE 8-5: BUILT-IN GLOBAL VARIABLE DEFINITIONS FOR BOUNDARY ACCUMULATORS.

NAME	EXPRESSION
<scope>.<name>_ave	<wscope>.aveop(<scope>.<name>)
<scope>.<name>_int	<wscope>.intop(<scope>.<name>)
<scope>.<name>_max	<wscope>.maxop(<scope>.<name>)
<scope>.<name>_min	<wscope>.minop(<scope>.<name>)
<scope>.<name>_sum	<wscope>.intop(<scope>.<name>/<scope>.meshVol)

Here, <wscope> is the scope of the parent boundary feature, for example goprac.wall1.

Sound Pressure Level Calculation Theory

It is possible to compute the sound pressure level due to incident and reflected rays at a surface. To do so, the ray power must be solved for. Then the Sound Pressure Level Calculation node can be added as a subnode to the Wall node.

The sound pressure level is computed by first computing the Wall intensity I_w (SI unit: W/m^2). The wall intensity on a given boundary element is

$$I_{\mathrm{w},i} = \frac{1}{A_i}\sum_j Q_j(|R|^2 + 2|R|\cos(\arg(R)) + 1)$$

where the subscript i denotes the ith boundary element, the subscript j indicates the jth ray, Q_j (SI unit: W) is the power transferred by the jth ray, and R (dimensionless) is the reflection coefficient. The sum is taken over all rays that hit the ith boundary element.

The sound pressure level $L_{\mathrm{p},i}$ in the ith element is

$$L_{\mathrm{p},i} = 10\log\left(\frac{\rho c I_{\mathrm{w},i}}{p_{\mathrm{ref,SPL}}^2}\right)$$

where ρ is the density of the fluid, c is the speed of sound, and $p_{\mathrm{ref,SPL}}$ is the reference pressure corresponding to 0 dB. Note that the base-10 logarithm is used.

Assigning Spatial Directivity to a Source

The Source with Directivity node can be used to release rays in a distribution with user defined spatial directivity. This node sets up a point source where rays emanate outward in a spherical or hemispherical distribution.

When the ray intensity or power is solved for, you may either specify the spatial directivity in terms of sound pressure level or enter the total source power. When specifying a spatial directivity function, the intensity of each released ray at a distance r (SI unit: m) from the release point is

$$I = \frac{p_{\mathrm{RMS}}^2}{\rho c}\left(\frac{R_{\mathrm{ref}}}{r}\right)^2$$

where p_{RMS} is the root mean square pressure at a distance $r = R_{\mathrm{ref}}$ from the source,

$$D(\varphi, \theta) + L_{\mathrm{ref}} = 20\log_{10}\left(\frac{p_{\mathrm{RMS}}}{p_{\mathrm{ref,SPL}}}\right)$$

- ρ (SI unit: kg/m^3) is the density of the surrounding fluid,
- c (SI unit: m/s) is the speed of sound in the medium,
- R_{ref} (SI unit: m) is the reference distance,
- $D(\varphi, \theta)$ is the spatial directivity in dB, and
- L_{ref} is the reference level in dB.

Strictly speaking, the ray acoustics approximation gives infinite ray intensity for a spherical or hemispherical wavefront as it is first released from an infinitesimally small point source. In reality, diffraction effects would make the ray acoustics approximation inaccurate within a few wavelengths of the release position.

If the ray power is solved for, then the total power of all released rays is

$$P_{\mathrm{src}} = \Omega \frac{p_{\mathrm{RMS}}^2}{\rho c} R_{\mathrm{ref}}^2$$

where Ω (SI unit: rad) is the total solid angle subtended by the released rays: 2π for a hemispherical release or 4π for a spherical release.

Alternatively, you can specify the total source power P_{src} directly, along with a power weighting factor P_{wt}. Then the intensity and power of each released ray is proportional to the value of P_{wt} evaluated on that ray, while the sum of the initial power over all rays is P_{src}.

Impulse Response Reconstruction

In a ray tracing simulation (with the Ray Acoustics interface), an impulse response (IR) can be reconstructed by recombining the information carried by rays and detected at the receiver. The information carried by the rays consists of the ray power (with a given frequency band resolution) and the travel time. The aim of the reconstructed IR is to have the correct energetic signature, energy decay characteristics, and spectral characteristics. A reconstructed IR, from a ray tracing simulation, will never match a measured IR if compared sample per sample. This is simply not possible since information like the phase content is missing when frequency bands are used; ray tracing is not a full wave method.

The ray tracing simulation consists of propagating rays (ray index n) in frequency bands (band index c and center frequency f_{c}). In each frequency band c, the ray power Q_{n} (of ray n) decreases at ray wall interactions (absorption) and due to volumetric attenuation. Wall absorption, scattering, and volumetric attenuation characteristics is defined with a given n'th octave band resolution.

The ray arrives at the receiver at time t_{n} with an intensity/pressure amplitude contribution, given by

$$I_{\mathrm{n}} = \frac{Q_{\mathrm{n}} d}{V_{\mathrm{rec}}} \qquad p_{\mathrm{rms,n}}^2 = \rho c I_{\mathrm{n}} \qquad p_{\mathrm{rms,n}} = \frac{|p_{\mathrm{n}}|}{\sqrt{2}}$$

where V_{rec} is the volume of the receiver, d is the distance traveled in the receiver by the ray, p_{rms} is the RMS pressure, ρ is the density, and c is the speed of sound. The arrival time t_{n} is computed as the time instance where the ray has traveled half the distance $d/2$ in the receiver volume (this leads to a good accuracy in time).

To reconstruct the IR the filter kernels (of the nth octave filter bank) are convolved with the impulse signal (it is assumed that rays carry an impulse or Dirac impulse when emitted from the source). The filter kernel of band c is $w_{\text{c}}(t)$. The built-in filters are so-called brick-wall with Kaiser window filters, defined in octaves, $1/3$ octaves, or $1/6$ octaves (see Impulse Response Plot). The f_{c} = 1 kHz octave filter kernel in time and frequency domain is depicted inFigure 8-8.

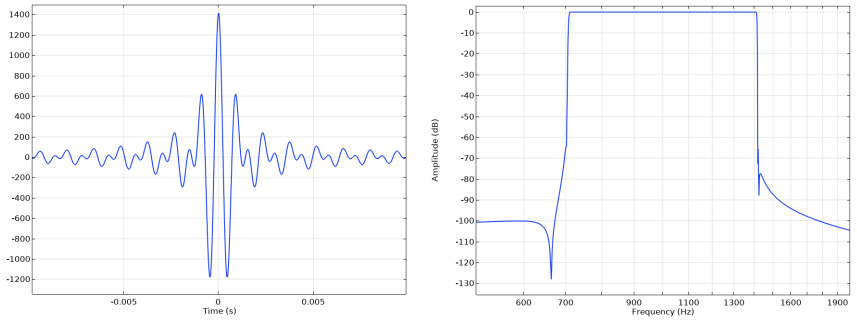


Figure 8-8: The 1 kHz brick-wall with Kaiser window filter kernel in time and frequency domain.

An often seen expression for the IR reconstruction is given using amplitude summation

$$\sum_{c}\sum_{n}(-1)^{N_{\text{ref,n}}}w_{\text{c}}(t-t_{\text{n}})p_{\text{rms,n}}$$

where $N_{\text{ref,n}}$ is the number of reflections undergone by ray n, before arriving at the receiver. The first factor in the sum ensures that rays that follow the same path (such as direct sound and early reflections) are summed up correctly (in phase), while rays that have undergone many reflections get a pseudo random phase applied (see Ref. 6 and Ref. 7).

An issue with the above summation approach is that the level of the IR is not correct in absolute terms. For example, the level of the reconstructed signals at a small and large receiver, placed at the same location, will not be identical. To remedy this, the

convolution/filtering of each ray is still done on the amplitude but the summation has to be done energetically. This gives instead

$$p_{\mathrm{n}}(t) = w_{\mathrm{c}}(t - t_{\mathrm{n}}) p_{\mathrm{rms,n}}$$

$$h_c^{\mathrm{p}}(t) = \sum_n (-1)^{N_{\mathrm{ref,n}}} \mathrm{sign}(p_{\mathrm{n}}(t)) \frac{|p_{\mathrm{n}}(t)|^2}{2}$$

$$h_c(t) = \mathrm{sign}(h_{\mathrm{c}}^{\mathrm{p}}(t)) \sqrt{|h_c^{\mathrm{p}}(t)|}$$

$$IR(t) = \sum_c h_c(t)$$

where the superscript p stands for power summed signal. This ensures a consistent energy content and energy conservation of the reconstructed IR signal.

References for the Ray Acoustics Interface

1. L.D. Landau and E.M. Lifshitz, *Fluid Mechanics, Course of Theoretical Physics, Volume 6*, Butterworth–Heinemann, 2003.

2. O.N. Stavroudis, *The Optics of Rays, Wavefronts, and Caustics*, Academic, 1972.

3. R.M. More and K. Kosaka, "Wave-front curvature in geometrical optics", Phys. Rev. E, vol. 57, no. 5, pp. 6127–6134, 1998.

4. A.D. Pierce, *Acoustics: An Introduction to Its Physical Principles and Applications*, Acoustical Society of America, 1991.

5. F.B. Jensen, W.A. Kuperman, M.B. Porter, and H. Schmidt, *Computational Ocean Acoustics*, Second Edition, Springer, 2011.

6. M. Vorländer, *Auralization: Fundamentals of Acoustics, Modelling, Simulation, Algorithms and Acoustic Virtual Reality*, Springer, 2008.

7. A. Alpkocak and M.K. Sis, "Computing Impulse Response of Room Acoustics Using the Ray-Tracing Method in Time Domain," *Archives of Acoustics*, vol. 35, pp. 505–519, 2010.

8. H. Kuttruff, *Room Acoustics*, Fifth Edition, CRC Press, 2009.

9. . Houtgast, H.J.M. Steeneken, and R. Plomp, "Predicting Speech Intelligibility in Rooms from the Modulation Transfer Function. I. General Room Acoustics," *Acustica*, vol. 46, pp. 60–72, 1980.

The Acoustic Diffusion Equation Interface

The **Acoustic Diffusion Equation (ade)** interface ( ), found under the **Acoustics** > **Geometrical Acoustics** subbranch ( ), solves a diffusion equation for the acoustic energy density. This type of analysis also known as energy finite elements or EFEM. It is applicable for high-frequency acoustics inside coupled rooms when the acoustic fields are diffuse. The interface only exists in 3D. The diffusion of the acoustic energy density depends on the mean free acoustic path and thus on the individual room geometry. Absorption may be applied at walls and a transmission loss may be applied when coupling rooms together. Increased diffusion due to room fitting can be added. Material properties and sources may be specified in frequency bands.

The interface is well suited for quick assessment of sound pressure level distribution inside buildings and other large structures. Compared to a ray acoustics simulation this interface does not include any phase information, direct sound, and early reflections. The interface supports stationary studies for modeling a steady-state sound energy or sound pressure level distribution. A time dependent study can be used to determine energy decay curves and reverberation times. The reverberation time of coupled and uncoupled rooms can also be determined using the eigenvalue study. The eigenvalue is directly related to the exponential decay time.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Acoustic Diffusion Model**, **Room**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can

also right-click **Acoustic Diffusion Equation** to select physics features from the context menu.

For tutorial models that use the Acoustic Diffusion Equation, see the Application Library model:

- *Acoustics in a Single-Family Home*. Application Library path **Acoustics_Module/Building_and_Room_Acoustics/one_family_house**

Or the Application Gallery models:

- *Acoustics of Coupled Rooms Using the Acoustic Diffusion Equation:* www.comsol.com/model/20171
- *Acoustic Mean Free Path in a Room:* www.comsol.com/model/72491

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `ade`.

SOUND PRESSURE LEVEL SETTINGS

See Sound Pressure Level Settings for the Pressure Acoustics, Frequency Domain interface. Note that only **Use reference pressure for air** or **User-defined reference pressure** are available selections.

BAND STUDIED

This section cannot be edited when the **Band Input Type** is **Flat (continuous)**. For all other options under **Band Input Type**, the default N_{study} (dimensionless) is 1, meaning that material and other properties defined in the first band are used. To sweep the bands enter a parameter (defined under **Global Definitions** > **Parameters**) and solve the model using a parametric sweep in the study, in this way solving the model for the desired number of bands.

BAND INPUT TYPE

Select a **Type** — **Flat (continuous)** (the default), **Octave bands**, **1/3 octave bands**, or **User defined bands**. The frequency content of a model solved using the acoustic diffusion equation all lies implicitly in material parameters, absorption properties, sources and so forth. These can be given in a broadband (or for a single frequency) by selecting Flat (continuous) or in bands when given in this way.

- For **Octave bands** the table that displays defaults to 11 rows under **Band number** with the associated **Lower band limit (Hz)**, **Center frequency (Hz)**, and **Upper band limit (Hz)** values.
- For **1/3 octave bands** the table that displays defaults to 32 rows under **Band number** with the associated **Lower band limit (Hz)**, **Center frequency (Hz)**, and **Upper band limit (Hz)** values.
- For **User defined bands** the table that displays defaults to 1 row under **Band number** with default associated values in the **Lower band limit (Hz)**, **Center frequency (Hz)**, and **Upper band limit (Hz)** columns.
  - Under the table click the **Add** (+) or **Delete** ( ) buttons to edit the table contents. Or right-click a table cell and select **Add** or **Delete**.
  - To save the contents of a table, click the **Save to File** button ( ) and enter a **File name** in the **Save to File** dialog, including the extension `.txt`. Click to **Save** the text file. The information is saved in space-separated columns in the same order as displayed on screen.
  - Use the **Load from File** button ( ) and **Load from File** dialog to import data in text files, generated by, for example, a spreadsheet program. Data must be separated by spaces or tabs (or be in a Microsoft Excel Workbook spreadsheet if the license includes LiveLink™ *for* Excel®).

The lower band limit, center frequency, and upper band limit in the studied band are defined as the global variables `ade.fL`, `ade.fC`, and `ade.fU`, respectively. These can, for example, be used when plotting data in postprocessing.

Note that the tables that are imported, using the **Load from File** button ( ), will replace the existing data. You can only import a table that has the expected size. The band number column is disregarded when importing. To create a data file to load, it is recommended to save the existing table (octaves or 1/ octaves) and then edit and change the values in that file.

DISCRETIZATION

From the list select the element order for the **Sound energy density**, the default is **Quadratic**.

DEPENDENT VARIABLES

This physics interface defines one dependent variable (field), the **Sound energy density** w. If required, edit the name, which changes both the field name and the dependent variable name. If the new field name coincides with the name of another pressure field in the model, the interfaces share degrees of freedom and dependent variable name. The new field name must not coincide with the name of a field of another type, or with a component name belonging to some other field.

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Domain, Boundary, and Global Nodes for the Acoustic Diffusion Equation Interface
- Theory for the Acoustic Diffusion Interface

Domain, Boundary, and Global Nodes for the Acoustic Diffusion Equation Interface

The Acoustic Diffusion Equation has these domain, boundary, and point nodes available (listed in alphabetical order).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Acoustic Diffusion Model
- Continuity[1]
- Manual Destination Selection
- Domain Source
- Fitted Domain
- Initial Values
- Inward Energy Flux
- Mapped Room Coupling
- Point Source
- Room
- Room Coupling
- Wall

[1]Described for the Pressure Acoustics, Frequency Domain interface

Acoustic Diffusion Model

The **Acoustic Diffusion Model** is added by default and is used to define the material properties and the governing diffusion equations.

MATERIAL PROPERTIES

For each of the following properties the default takes values **From material**, or select **User defined** to enter a different value or expression in the text field.

- **Density** ρ (SI unit: kg/m^3). The default is 1.2 kg/m^3.

- **Speed of sound** c (SI unit: m/s). The default is 343 m/s.
- **Volume absorption coefficient** m_a (SI unit: 1/m). The default is 0 1/m. This parameter determines the amount of absorption due to losses in the air. The effects are usually of interest in very large rooms/volumes or at very high frequencies. The built-in option **Classical viscous and thermal absorption** defines the frequency absorption coefficient due to viscous and thermal losses. For the **Flat (continuous)** band option it is evaluated at the studied frequency f_{eval} (the default is 1000 Hz) while it is averaged over the studied band for the other Band Input Type.

Room

The **Room** node is added by default and is used to define the mean free path model of each room (closed volumes) in a model. A Wall subnode is also added by default. Additional **Wall** subnodes and the Inward Energy Flux subnode are available from the context menu (right-click the parent node) or from the **Physics** toolbar.

Each room (volume) in a model needs a **Room** node with associated definitions of all wall properties. The volume and surface area of the room is calculated and used to determine the mean free path (depending on the model selected).

MEAN FREE PATH MODEL

Select a **Mean free path model** — **Ideal diffuse field** (the default) or **User defined**. For **User defined** enter a **Mean free path** λ (SI unit: m). In most cases, the ideal diffuse field is valid and applicable. For the user defined case, the mean free path can, for example, be computed using a ray acoustics model.

For a tutorial model that computes the mean free path, see the Application Gallery model: *Acoustic Mean Free Path in a Room*
www.comsol.com/model/acoustic-mean-free-path-in-a-room-724915

Classical statistical room acoustic variables that are based on the room geometry and absorption properties of the walls are defined and can be used for postprocessing. See Statistical Model of Reverberation Time for details. They are:

- The total room absorption `ade.A`
- The Sabine reverberation time `ade.T60_Sabine`
- The Eyring-Norris reverberation time `ade.T60_EN`
- The Schroeder frequency of the room `ade.f_Schroeder`

Each room feature (say `ade.room1`, `ade.room2`, and so on) also has several room variables associated that may be plotted in postprocessing. They are:

- The room volume `ade.room1.V`
- The room surface area `ade.room1.S`
- The mean free path of the room `ade.room1.lambda`

Wall

The **Wall** subnode is available from the context menu (right-click the Room parent node) or from the **Physics** toolbar, **Attributes** menu. It is used to define the wall absorption properties.

WALL ABSORPTION PROPERTIES

The absorption coefficient properties to define depend on the choice of Band Input Type for The Acoustic Diffusion Equation. The absorption is defined through the random incidence absorption coefficient α (the value is different from the normal incidence absorption typically used in, for example, pressure acoustics).

- If the **Band Input Type** is **Flat (continuous)** enter an **Absorption coefficient** α (dimensionless).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter an **Absorption coefficient** α (dimensionless) in each row corresponding to the **Band Number**.

Select an **Exchange coefficient model** — **Sabine type** (the default), **Eyring type**, **Modified Eyring type**, or **User defined**. For **User defined** enter an **Exchange coefficient** h (SI unit: m/s). The choice gives different statistical models which are applicable for different

ranges of absorption coefficients. The Sabine type is applicable for $\alpha < 0.2$ while the two others are valid up to an absorption of 1.

!	Note that the tables that are imported, using the **Load from File** button ( ), will replace the existing data. You can only import a table that has the expected size. The band number column is disregarded when importing. To create a data file to load, it is recommended to save the existing table and then edit and change the values in that file.

Inward Energy Flux

The **Inward Energy Flux** subnode is available from the context menu (right-click the Room parent node) or from the **Physics** toolbar, **Attributes** menu. It is used to define the energy flux that stems from wall sources. This could be a vibrating wall of a structure or the energy flux as measured from external sources to a building.

ENERGY FLUX

The inward energy flux properties to define depend on the choice of Band Input Type for The Acoustic Diffusion Equation.

- If the **Band Input Type** is **Flat (continuous)** enter an **Inward energy flux** J_{in} (SI unit: W/m^2).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter an **Inward energy flux** J_{in} (SI unit: W/m^2) in each row corresponding to the **Band Number**.

Initial Values

The **Initial Values** node adds initial values for the sound energy density. Add more **Initial Values** nodes from the **Physics** toolbar. When calculating the energy impulse response of a system using a time dependent study, set the initial value inside a small volume (small domain) or define it in terms of a Gaussian shape.

INITIAL VALUES

Enter a value or expression for the **Sound energy density** w (SI unit: J/m^3) initial value. The default is 0 J/m^3.

Domain Source

Use the **Domain Source** node to define a source, the right-hand side of the governing diffusion equation. The source can be a constant value defined inside a small volume, it can be a function of time, or it can a function of space. The value corresponds to the total emitted power Q of the source it is automatically transformed into a power density q by dividing with the source domain volume (calculated automatically).

DOMAIN SOURCE

How you define the domain source depends on the choice of Band Input Type for The Acoustic Diffusion Equation.

- If the **Band Input Type** is **Flat (continuous)** enter a **Domain source** Q (SI unit: W).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter a **Domain source** Q (SI unit: W) in each row corresponding to the **Band Number**.

Fitted Domain

Use the **Fitted Domain** node to define the fitting parameters. These are parameters that define a region with, for example, furniture inside a room. The domain is characterized in terms if its average statistical properties.

FITTED DOMAIN PARAMETERS

Enter values or expressions for the following:

- **Number density of fitting** n_f (SI unit: $1/\mathrm{m}^3$).
- **Average cross-section of fitting** Q_f (SI unit: m^2).

How you define the **Absorption coefficient of fitting** is based on the choice of Band Input Type for The Acoustic Diffusion Equation.

- If the **Band Input Type** is **Flat (continuous)** enter a value in the text field for the **Absorption coefficient of fitting** α_f (dimensionless). For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter an **Absorption coefficient of fitting** α_f (dimensionless) in each row corresponding to the **Band Number**.

Room Coupling

Use the **Room Coupling** node to couple rooms together allowing an acoustic energy flux back and forth. This can be used to model transmission through walls or openings defined in terms of their transmission loss. The feature is also used to couple rooms

that are connected through a large open surface. It is also available as the **Pair Room Coupling**.

TRANSMISSION LOSS

How you define the transmission loss depends on the choice of Band Input Type for The Acoustic Diffusion Equation.

- If the **Band Input Type** is **Flat (continuous)** enter a **Transmission loss** TL (SI unit: dB).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter a **Transmission loss** TL (SI unit: dB) in each row corresponding to the **Band Number**.

CONTINUITY

Select the **Force continuity** checkbox to constrain the dependent variable resulting in a zero transmission loss. Use this option when rooms are coupled through large open surfaces. Note that the absorption properties of these surfaces need to be set correctly independent of this option.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

Mapped Room Coupling

Use the **Mapped Room Coupling** node to couple two rooms when they do not share a common inner boundary. This is, for example, necessary if mid-planes have not been added in the CAD drawing. The condition has the same functionality as the Room Coupling feature except that a mapping is used between the source and destination walls.

TRANSMISSION LOSS

How you define the transmission loss depends on the choice of Band Input Type for The Acoustic Diffusion Equation.

- If the **Band Input Type** is **Flat (continuous)** enter a **Transmission loss** TL (SI unit: dB).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter a **Transmission loss** TL (SI unit: dB) in each row corresponding to the **Band Number**.

CONTINUITY

Select the **Force continuity** checkbox to ensure continuity resulting in a zero transmission loss.

To display the **Constraint Settings**, **Orientation of Source**, **Orientation of Destination**, and **Mapping Between Source and Destination** sections, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

For details see Constraint Settings, Orientation of Source and Destination, and Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

Manual Destination Selection

The **Manual Destination** option is available from the context menu (right-click the Mapped Room Coupling parent node) or from the **Physics** toolbar, **Attributes** menu. When selected a **Destination** selection section appears in the Mapped Room Coupling feature.

Point Source

Use the **Point Source** node to add a point source to model a source of small dimensions. For a time dependent model, it is possible to enter an expression that depend on time.

POINT SOURCE

How you define the **Point source power** depends on the choice of Band Input Type for The Acoustic Diffusion Equation.

- If the **Band Input Type** is **Flat (continuous)** enter a **Point source power** q_p (SI unit: W).
- For **Octave bands**, **1/3 octave bands**, or **User defined bands**, enter a **Point source power** q_p (SI unit: W) in each row corresponding to the **Band Number**.

Modeling with the Acoustic Diffusion Equation Interface

In this section:

- The Eigenvalue Study Type
- Combined Stationary and Time Dependent Study

The Eigenvalue Study Type

The *Acoustic Diffusion Equation* interface and eigenvalue study will produce the modes that decay exponentially for each room, including the effect of couplings to adjacent rooms and wall absorption. The modes are of the form

$$w(\mathbf{x}, t) = w_i(\mathbf{x})e^{-\lambda_i t}$$

where each mode w_i has an associated eigenvalue λ_i (this is the exponential time scale). The eigenvalue directly gives the reverberation time T_{60} of that subsystem

$$T_{60} = \frac{60}{10\log_{10}(e)\lambda_i} \approx \frac{55.3}{4\lambda_i}$$

Typically, there will be as many modes as there are rooms in the building.

A list relating the eigenvalues to the reverberation times is automatically shown in an evaluation group in the **Results** node. Identify the mode associated with a given reverberation time by looking at the **Sound Energy Density Mode** default plot.

Combined Stationary and Time Dependent Study

By combining a Stationary and a Time Dependent study you can readily model the so-called steady-state decay behavior of the system. Use the stationary solution as an initial condition for the transient model. Remember to disable all sources in the transient study step by enabling the **Modify physics tree and variables for study step** option. Evaluating the sound pressure level, `ade.Lp`, in different points will result in the energy decay curves.

Theory for the Acoustic Diffusion Interface

This section contains a small review of the classical models for the prediction of reverberation time in room acoustics followed by the theory background of the acoustic diffusion equation.

In this section:

- Statistical Model of Reverberation Time
- The Acoustic Diffusion Equation
- References for the Acoustic Diffusion Equation Interface

Statistical Model of Reverberation Time

In classical statistical acoustics (see Ref. 9 and Ref. 16) the reverberation time T_{60} (RT) is the time taken for the sound pressure level (created by an impulse source) to decay 60 dB. The reverberation time can be calculated using the Sabine formula

$$T_{60} = \frac{55.3V}{cA}$$

where V is the room volume, c is the speed of sound, and A is the total absorption of all surfaces. The factor 55.3 is a roundoff of $24 \cdot \ln(10)$. The total absorption is calculated by

$$A = \sum S_i \alpha_i = S\bar{\alpha}$$

where S_i and α_i denote the area and absorption of the i'th area. S is the total surface area and $\bar{\alpha}$ is the average absorption of the room. For large rooms this equation is corrected by the absorption of air

$$T_{60} = \frac{55.3V}{cA + 4m_a V}$$

where m_a is the air absorption constant (see Table 3.1 in Ref. 9). Other versions of the T_{60} reverberation time exist one is the Eyring–Norris given by

$$T_{60} = \frac{55.3V}{-cS\ln(1-\bar{\alpha})}$$

The models for the reverberation time given above are statistical models for diffuse sound field (large number of reflections). At low frequencies the modal behavior of the room becomes important. The lower bound for the application of these statistical models is the Schroeder frequency (see Ref. 10 and Ref. 16) given by

$$f > 2000(\text{m/s})^{3/2}\sqrt{T_{60}/V}$$

The classical expressions given above are available for postprocessing in the Acoustic Diffusion Equation interface.

The Acoustic Diffusion Equation

The acoustic diffusion model is based on the assumption that the volumes (rooms) studied contain scatterers that uniformly scatter the sound field and that the sound field is diffuse (large number of reflections). Using the diffusion of light in a scattering environment as an analogy one can express a diffusion equation for the sound-energy density $w = w(\mathbf{x},t)$ (SI unit: J/m^3). The diffusion equation describes the energy flow from high to low energy regions. Further details about the model equations and boundary conditions are found in papers by Xiang and others (see Ref. 3, Ref. 7, and Ref. 8) and in papers by Billon, Valeau, and more (see Ref. 5, Ref. 6, Ref. 11, and Ref. 15).

DOMAIN EQUATIONS

The domain diffusion equation for the sound-energy density $w = w(\mathbf{x},t)$ is given by

$$\frac{\partial w}{\partial t} + \nabla \cdot \mathbf{J} + cm_a w = q(\mathbf{x}, t)$$

where the local energy flux vector $\mathbf{J}$ (SI unit: $\text{J/m}^2\text{/s} = \text{W/m}^2$) is defined in the usual way, as

$$\mathbf{J} = -D_t \nabla w$$

The total diffusion coefficient is $D_t = D = \lambda c/3$ (SI unit: $\text{m}^2\text{/s}$), λ is the mean free path (SI unit: m), c is the speed of sound (SI unit: m/s), and m_a is the volumetric absorption coefficient of air (SI unit: 1/m). The volumetric absorption coefficient (or attenuation coefficient) should not be confused with the energy absorption coefficient α used in boundary conditions which is dimensionless. The source term q represents

the spatial sound source (SI unit: $J/m^3/s = W/m^3$). The term $cm_a w$ accounts for volume absorption in air (dissipation). Note that in certain models, the $cm_a w$ term accounts for the total absorption at boundaries in a "mean" sense (only using no flux boundary conditions); this approach is not used here.

In the interface, m_a can be given as a user input or it can be defined as the "classical viscous and thermal (volumetric) absorption" coefficient (as in pressure acoustics) given by

$$m_{a,\,\text{classical}} = \frac{\omega^2 \mu}{2\rho c^3}\left(\frac{4}{3} + \frac{\gamma - 1}{\text{Pr}}\right)$$

The absorption coefficient is either integrated over the band or it is given at a certain frequency. The volumetric absorption is only important for very large domains/rooms. It is furthermore assumed that $m_a \lambda << 1$ such that the diffusion coefficient remains unchanged.

The mean free path λ is the distance a sound particle on average travels between reflections. It is related to the average reflections frequency $\not{n}$ by $\lambda = c/\not{n}$. The mean free path is a property of each room and for a regular cubic like room it can be calculated by (the usual convention)

$$\lambda = \frac{4V}{S}$$

where V is the room volume and S the total room surface area (see Ref. 18). The mean free path is in principle defined for every room as it depends on the room geometry and shape (see examples in Ref. 13 and Ref. 14). Entering a user defined mean free path based on for example measurements or a ray tracing simulation is also possible (see Ref. 1, Ref. 2, and Ref. 17).

Fitted Rooms/Zones with Obstacles (Chairs, Machines, and So On)

In zones with fittings, like furniture or other absorbers, the scatterers are modeled statistically by their number density n_f (SI unit: $1/m^3$), their average cross-section Q_f (SI unit: m^2), and their absorption coefficient α_f (dimensionless). For the domain with scatterers the mean free path becomes

$$\lambda_\text{f} = \frac{1}{n_\text{f} Q_\text{f}}$$

defining the fitting diffusion constant

$$D_f = \frac{c\lambda_f}{3}$$

With these definitions in place, the governing diffusion equation can be modified to take the fitting into account (see, for example, Ref. 15)

$$\frac{\partial w}{\partial t} + \nabla \cdot (-D_t \nabla w) + c\left(m_a + \frac{\alpha_f}{\lambda_f}\right) w = q(\mathbf{x}, t) \qquad D_t = \frac{D_f D}{D_f + D}$$

VALIDITY

The time limit after which the acoustic diffusion model leads to correct results has been discussed by many authors; see, for example, Ref. 6, Ref. 12, or Ref. 15. They suggest that a limit of one mean free time λ/c could be considered for the diffusion equation to have physical meaning. Before this time, the high probability of the particles of not having hit a scatterer/surface yet leads to invalid results. Typically, this type of models only apply at frequencies above the Schroeder frequency. This is a good "rule of thumb" measure — in several publications results have been seen to match measurements also at lower frequencies. Below the Schroeder frequency the room eigenmodes are important and can be modeled using Pressure Acoustics.

SOURCES

The source term q ($\mathbf{x}$,t) (SI unit of power density $J/m^3/s = W/m^3$) can be defined as a point source or a volume source.

- Point source located at $\mathbf{x}_s$ are of the type q ($\mathbf{x}$,t) = $P(t)\delta(\mathbf{x} - \mathbf{x}_s)$ where P is the power of the source (SI unit: W).
- Volume/domain sources of volume V_s emitting the power Q (SI unit: W) yielding a power density Q/V_s.
- An impulse source in a transient model is typically modeled as an initial value for $w(\mathbf{x},0)$ in a given small domain (constant in space). The source (initial value) could also be Gaussian shaped in space, here a larger domain.

BOUNDARY CONDITIONS

Wall (with Different Absorption Properties)

At walls, a mixed boundary condition accounting for absorption losses is used (the surface normal $\mathbf{n}$ being outward to the volume Ω)

$$\mathbf{J} \cdot \mathbf{n} = -D_t \frac{\partial w}{\partial n} = J_{out} = hw$$

where h is the exchange coefficient. Different models exist for this coefficient as given by Xiang and others (see Ref. 3 and Ref. 4). The Sabine type exchange coefficient is

$$h = cA_S = c\frac{\alpha}{4} \qquad \text{for} \qquad \alpha < 0.2$$

This expression is valid for surfaces where the absorption coefficient is smaller than 0.2. The Eyring type model is defined by

$$h = cA_E = c + \frac{-\ln(1-\alpha)}{4} \qquad \text{for} \qquad \alpha < 1$$

This model is not suited for absorption coefficients close to 1. Finally, there is the (modified) model by Jing and Xiang (see Ref. 4)

$$h = cA_M = c\frac{\alpha}{2(2-\alpha)} \qquad \text{for} \qquad \alpha \leq 1$$

In this model, the boundary condition is theoretically grounded and can model high absorption for a small portion of surfaces. In addition, the diffusion-equation model inherently assumes that overall absorption in rooms under test must not be "high".

Interior BCs (Coupling Between Rooms/Volumes)

On interior boundaries between rooms (thin walls, doors, grills, thin panels, and so on) there can be a transmission loss (TL) associated. The transmission loss is related to the transmission coefficient τ of the boundary as

$$\mathrm{TL} = 10\log\left(\frac{1}{\tau}\right)$$

This results in a condition where the field w is discontinuous across the boundary (a slit). The TL for an opening is typically given for the boundary in which the opening is located. That is, the combined transmission and absorption is smeared on a boundary.

With the surface normal pointing out of the domains, the flux on the downside of the surface/wall (including absorption) is

$$\mathbf{J} \cdot \mathbf{n}_{\mathrm{down}} = \mathrm{down}\left(hw + \frac{\tau c}{4}w\right) - \mathrm{up}\left(\frac{\tau c}{4}w\right)$$

on the upside of the surface/wall (including absorption) the total flux is

$$\mathbf{J} \cdot \mathbf{n}_{\text{up}} = \text{up}\left(hw + \frac{\tau c}{4} w\right) - \text{down}\left(\frac{\tau c}{4} w\right)$$

The transmission coefficient is due to reciprocity the same from both sides. This results in the combined condition

$$\mathbf{J} \cdot \mathbf{n}_{\text{up}} - \mathbf{J} \cdot \mathbf{n}_{\text{down}} = [hw]_{\text{up}} - [hw]_{\text{down}} - \frac{\tau c}{4}(w_{\text{up}} - w_{\text{down}})$$

The "room coupling" condition adds the fluxes related to the TL. Note also that the terms of the type $\tau cw/4$ should probably be modified for small TL because τ then becomes larger than 0.2 (at around 7 dB). An option exists to force continuity by constraining

$$w_{\text{up}} = w_{\text{down}}$$

which basically corresponds to a null transmission loss.

Inflow and Outflow Boundary Conditions

Some general boundary conditions defining a total inflow or outflow of energy are added as

$$\mathbf{J} \cdot \mathbf{n} = -D_{\text{t}} \frac{\partial w}{\partial n} = J_{\text{out}}$$

$$\mathbf{J} \cdot \mathbf{n} = -D_{\text{t}} \frac{\partial w}{\partial n} = -J_{\text{in}}$$

POSTPROCESSING VARIABLES/EXPRESSIONS

The square pressure (valid for spatially averaged quantities and volumes that are large compared to the wavelength but small compared to the mean free path):

$$p^2(\mathbf{x}, t) = \rho c^2 w(\mathbf{x}, t)$$

Solution of the energy equation yields (with an impulse source) the energy room impulse response (excluding the direct sound, as the sound field is assumed diffuse). It is also known as the energy-time function (ETF) or energy-time curve (ETC). The SPL response is given by

$$L_{\text{p}}(\mathbf{x}, t) = 10\log\left(\frac{\rho c^2 w(\mathbf{x}, t)}{P_{\text{ref}}^2}\right)$$

This is in general the local SPL for any source. For a steady-state model, this is the spatial SPL distribution $L_p(\mathbf{x})$.

The steady-state sound-energy decay $d(\mathbf{x},t)$ can be calculated from an energy impulse response using so-called Schroeder integration. This corresponds to first finding the steady state (using a stationary solver) and then use this solution as initial condition (source turned off) in a time-dependent model. The same response can be calculated using Schroeder integration of an impulse response model

$$d(\mathbf{x},t) = \frac{1}{N(\mathbf{x})}\int_t^\infty w(\mathbf{x},\tau)d\tau \qquad N(\mathbf{x}) = \int_0^\infty w(\mathbf{x},\tau)d\tau$$

This integration can be set up in COMSOL Multiphysics postprocessing using the built-in `timeint()` operator.

A variable for the a posteriori assessment of whether the diffuse field assumption is met is given by (named "A posteriori diffuse field evaluation norm" in the postprocessing list)

$$L_{\text{diffuse}} = \frac{|\mathbf{J}|}{cw} = \frac{D_t|\nabla w|}{cw}$$

If this variable is small, then the acoustic diffusion assumption is met. Another variable that can be assessed is the Schroeder frequency as given above.

DIFFERENT STUDY TYPES

Transient

A transient simulation is used to model the energy impulse response (the response to a source of the type $E_0\lambda(t)$, typically approximated by a step function of a short time) of a room or the steady-state energy decay response (decay from a steady state solution). In a single room, the response will be the same as measured in all points (except for a small time lag). This will result in identical reverberation times (RT) estimates. The interest of the transient model is more evident when several rooms are coupled. Here, several time scales exist because of the interaction between the rooms and these can be seen in a transient energy impulse response.

Steady State

The steady-state model is used to find the spatial SPL distribution inside a room or collection of rooms and volumes when subject to a steady noise source. This could be an engine noise or the noise generated by other machinery.

Eigenvalue

An eigenvalue analysis of the transient equation results in all the "basis functions" of the problem. That is, modes that do not change shape in space but only decay in time with the exponential time scale given by the eigenvalue. They are of the type

$$w(x,t) = w_i(x)e^{-\lambda_i t}$$

where λ_i is the i:th eigenvalue and w_i the associated mode. The eigenvalues are a direct measures of the reverberation time (RT) of the different rooms. T_{60} is for example simply given by

$$T_{60} = \frac{60}{10 \cdot \log(e) \cdot \lambda_i} = \frac{55.3}{4\lambda_i}$$

where the last equation is the classical expression. Evaluating the eigenvalues will gives an easy measure of the slopes of the energy-time functions (ETFs) and can be combined with a transient model to give the full picture of the reverberation in the different volumes (rooms) and their coupled behavior. When performing an eigenvalue study, the variable `ade.T60` gives the reverberation time associated with the given eigenvalue. Inspect the sound energy density modes plot to determine see to which room the reverberation time is associated.

References for the Acoustic Diffusion Equation Interface

1. J. Picaut, L. Simon, and J.D. Polack, "Sound field in long rooms with diffusely reflecting boundaries", *Appl. Acoust.*, vol. 56, pp. 217–240, 1999.

2. J. Pujolle, "Les différentes définitions du libre parcours moyen du son dans une salle", *Revue d'Acoustique*, vol. 36, pp. 44–50, 1976 (in French).

3. N. Xiang, Y. Jing, and A.C. Bockman, "Investigation of acoustically coupled enclosures using a diffusion-equation model", *J. Acoust. Soc. Am.,* vol. 126, 2009.

4. Y. Jing and N. Xiang, "On boundary conditions for the diffusion equation in room-acoustic prediction", *J. Acoust. Soc. Am.* vol. 123, pp. 145–153, 2008.

5. A. Billon and others, "Modeling the sound transmission between rooms coupled through partition walls by using a diffusion model", *J. Acoust. Soc. Am.,* vol. 123, 2008.

6. V. Valeau and others, "On the use of a diffusion equation for room acoustic prediction", *J. Acoust. Soc. Am.*, vol. 119, 2006.

7. Y. Jing and N. Xiang, "On The Use of a Diffusion Equation Model for Sound Energy Flow Prediction in Acoustically Coupled Spaces", COMSOL Conference Boston 2008, www.comsol.com/paper/on-the-use-of-a-diffusion-equation-model-for-sound-energy-flow-prediction-in-acoustically-coupled-spaces-5124.

8. Y. Jing and N. Xiang, "Boundary Conditions for the Diffusion Equation Model in Room-acoustic Prediction", COMSOL Conference Boston 2007.

9. T. J. Cox and P. D'Antonio, *Acoustic, Absorbers and Diffusers*, Taylor and Francis, 2009.

10. R. Schroeder, "New method of measuring reverberation time", *J. Acoust. Soc. Am.,* vol. 37, pp. 409–412, 1965.

11. C. Foy, V. Valeau, A. Billon, J. Picaut, and M. Hodgson, "An Empirical Diffusion Model for Acoustic Prediction in Rooms with Mixed Diffuse and Specular Reflections", *Acta Acustica united with Acustica*, vol. 95, pp. 97–105, 2009.

12. J. Escolano, J.M. Navarro, and J.L. Lopes, "On the limitation of a diffusion equation model for acoustic predictions of rooms with homogeneous dimensions (L)," *J. Acoust. Soc. Am.*, vol. 128, p. 1586, 2010.

13. C. Visentin, N. Prodi, V. Valeau, and J. Picaut, "A numerical investigation of the Fick's law of diffusion in room acoustics", *J. Acoust. Soc. Am.*, vol. 132, p. 3180, 2012.

14. C. Visentin, *A numerical and experimental validation of the diffusion equation applied to room acoustics*, PhD thesis, 2012.

15. V. Valeau, M. Hodgson, and J. Picaut, "A Diffusion-Based Analogy for the Prediction of Sound Fields in Fitted Rooms", *Acta Acustica united with Acustica*, vol. 93, pp. 94–105, 2007.

16. A. D. Pierce, *Acoustics, An Introduction to Its Physical Principles and Applications*, Chapter SIX, Acoustical Society of America, 1991.

17. L. Beranek and N. Nishihara, "Mean-free-paths in concert and chamber music halls and the correct method for calibrating dodecahedral sound sources", *J. Acoust. Soc. Am.*, vol. 135, p. 223, 2014.

18. H. Kuttruff, *Room Acoustics*, Fifth Edition, Spon Press, 2009.

19. Z. Xiangyang, C. Ke'an, and S. Jincai, "On the accuracy of the ray-tracing algorithms based on various sound receiver models", *Appl. Acoust.* vol. 64, pp. 433–441, 2003.

20. F. Pind, "Broadband Hybrid Room Acoustic Simulations," Master Thesis, Technical University of Denmark, DTU, 2013.

21. J. Soares Augusto, "The Io-sinh function, calculation of Kaiser windows and design of FIR filters", 10.13140/RG.2.1.3247.0489, 2015.

22. IOC, SCOR and IAPSO, 2010: *The international thermodynamics equation of seawater - 2010: Calculation and use of thermodynamic properties.* Intergovernmental Oceanographic Commission, Manuals and Guides No. 56, UNESCO (English), 196 pp.

9

Pipe Acoustics Interfaces

This chapter describes the physics interfaces found under the **Pipe Acoustics** branch ( ) and background theory.

- The Pipe Acoustics Frequency Domain and Transient Interfaces
- Theory for the Pipe Acoustics Interfaces

The Pipe Acoustics Frequency Domain and Transient Interfaces

In this section:

- The Pipe Acoustics, Frequency Domain Interface
- The Pipe Acoustics, Transient Interface
- Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces

The Pipe Acoustics, Frequency Domain Interface

The **Pipe Acoustics, Frequency Domain (pafd)** interface ( ), found under the **Acoustics** > **Pipe Acoustics** branch ( ) when adding a physics interface, is used to compute the acoustic pressure and velocity variations when modeling the propagation of sound waves in flexible pipe systems. The governing equations are formulated in a general way to include the possibility of a stationary background flow. The physics interface can for example be used to compute the propagation of sound waves in HVAC systems, other large piping systems, or simply in an organ pipe.

In the frequency domain all sources and variations are assumed to be harmonic. The solved equations assume that the propagating waves are plane. The propagation of higher-order modes that exist above their cutoff frequency, dictated by the pipe cross section, is not modeled.

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduce the equations to a 1D component with scalar dependent variables. The physics interface is available in 3D on edges and points, and in 2D on boundaries and points.

In the no-background flow case, the 1D pipe domain can be coupled to a 3D pressure acoustics model using the Acoustic–Pipe Acoustic Connection multiphysics coupling.

When this physics interface is added, these default nodes are also added to the **Model Builder**: **Fluid Properties**, **Pipe Properties**, **Closed**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and point conditions. You can also right-click **Pipe Acoustics, Frequency Domain** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `pafd`.

SOUND PRESSURE LEVEL SETTINGS

The zero level on the dB scale varies with the type of fluid. That value is a reference pressure that corresponds to 0 dB. This variable occurs in calculations of the sound pressure level L_p based on the root mean square (rms) pressure p_rms, such that

$$L_\mathrm{p} = 20\log\left(\frac{p_\mathrm{rms}}{p_\mathrm{ref}}\right) \qquad \text{with} \qquad p_\mathrm{rms} = \sqrt{\frac{1}{2}pp^*}$$

where p_ref is the reference pressure and the star (*) represents the complex conjugate. This is an expression valid for the case of harmonically time-varying acoustic pressure p.

Select a **Reference pressure for the sound pressure level** based on the fluid type:

- **Use reference pressure for air** to use a reference pressure of 20 µPa ($20{\cdot}10^{-6}$ Pa).
- **Use reference pressure for water** to use a reference pressure of 1 µPa ($1{\cdot}10^{-6}$ Pa).
- **User-defined reference pressure** to enter a reference pressure $p_{\mathrm{ref,\,SPL}}$ (SI unit: Pa). The default value is the same as for air, 20 µPa.

For postprocessing, plot the sound pressure level `pafd.Lp`, which depends on the selected reference pressure. You can also plot the intensity magnitude `pafd.I_mag` (this is an RMS value) in the frequency domain or the instantaneous intensity magnitude `patd.Ii_mag`.in the time domain.

DISCRETIZATION

Select the discretization of the element types used in the finite element formulation. Select the **Pressure** and **Tangential velocity** discretization orders. The default is to use a **Quadratic/Linear** formulation which also ensures numerical stability of the problem.

DEPENDENT VARIABLES

This section is used to define the dependent variables (fields) for **Pressure** p (SI unit: Pa) and **Tangential velocity** u (SI unit: m/s). If required, edit the name, but dependent variables must be unique within a model.

- Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces
- Theory for the Pipe Acoustics Interfaces

The Pipe Acoustics, Transient Interface

The **Pipe Acoustics, Transient (patd)** interface ( ), found under the **Acoustics** > **Pipe Acoustics** branch ( ) when adding a physics interface, is used to compute the acoustic pressure and velocity variations when modeling the propagation of sound waves in flexible pipe systems. The governing equations are formulated in a general way to include the possibility of a stationary background flow. The physics interface can for example be used to compute the propagation of sound waves in HVAC systems, other large piping systems, or simply in an organ pipe.

The solved equations assume that the propagating waves are plane. The propagation of higher-order modes that exist above their cutoff frequency, dictated by the pipe cross section, is not modeled.

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduce the equations to a 1D component with scalar dependent variables. The physics interface is available in 3D on edges and points, and in 2D on boundaries and points.

In the no-background flow case, the 1D pipe domain can be coupled to a 3D pressure acoustics model using the Acoustic–Pipe Acoustic Connection multiphysics coupling.

When this physics interface is added, these default nodes are also added to the **Model Builder**: **Fluid Properties**, **Pipe Properties**, **Closed**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and point conditions. You can also right-click **Pipe Acoustics, Transient** to select physics features from the context menu.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `patd`.

TRANSIENT SOLVER SETTINGS

Select the **Time stepping** (method) as **Manual** (default and recommended) or **Automatic/free** and then enter the **Maximum frequency to resolve** in the model. The default frequency is set to `1000[Hz]` but should be changed to reflect the frequency content of the sources used in the model. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

The rest of the settings are the same as for The Pipe Acoustics, Frequency Domain Interface.

- Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces
- Theory for the Pipe Acoustics Interfaces

Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces

The Pipe Acoustics, Frequency Domain Interface and The Pipe Acoustics, Transient Interface have these edge, boundary, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Closed
- End Impedance
- Fluid Properties
- Initial Values
- Pipe Properties
- Pressure
- Velocity
- Volume Force

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Initial Values

The **Initial Values** node adds initial values for the pressure and tangential velocity that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter values or expressions for the initial value of the **Pressure** p (SI unit: Pa) and the **Tangential Velocity** u (SI unit: m/s).

Fluid Properties

The **Fluid Properties** node adds the momentum and continuity equations solved by the physics interface, except for volume forces which are added by the Volume Force node. The node also provides an interface for defining the material properties of the fluid.

MODEL INPUTS

Enter a value for the **Background mean flow pressure** p_0 (SI unit: Pa) and the **Background mean flow temperature** T_0 (SI unit: K). These will also serve as input the material parameters that depend on the pressure and temperature.

BACKGROUND VELOCITY

Enter a value or expression for the **Background mean flow velocity** u_0 (SI unit: m/s). Values for the background variables for the pressure p_0 and velocity u_0 can be obtained by solving a *Pipe Flow* model on the same geometry. The Pipe Flow interface requires the Pipe Flow Module.

PHYSICAL PROPERTIES

Select a **Fluid model** as **Linear elastic** (the default). Then select the **Fluid Material** as **Edge Material** (the default) or select another material if added to the **Materials** node.

The default **Density** ρ (SI unit: kg/m^3) and **Speed of sound** c_s (SI unit: m/s) use the values **From material**. For **User defined** enter different values or expressions.

Pipe Properties

The **Pipe Properties** node is used to define the pipe shape, pipe model, wall drag force, and flow profile correction factor.

PIPE SHAPE

Select a pipe shape from the list — **Not set** (the default), **Circular**, **Square**, **Rectangular**, or **User defined**.

- For **Circular** enter a value or expression for the **Inner diameter** d_i (SI unit: m). The default is 10 cm (0.1 m).
- For **Square** enter a value or expression for the **Inner width** w_i (SI unit: m). The default is 5 cm (0.05 m).
- For **Rectangular** enter a value or expression for the **Inner width** w_i (SI unit: m; the default is 5 cm) and **Inner height** h_i (SI unit: m; the default is 10 cm).
- For **User defined** enter a value or expression for the **Cross-sectional area** A (SI unit: m^2; the default is 0.01 m^2) and **Wetted perimeter** Z (SI unit: m; the default is 0.4 m).

PIPE MODEL

Select a **Pipe model** — **Incompressible cross section** (the default), **Zero axial stress**, **Anchored at one end**, or **Anchored at both ends**.

When **Zero axial stress**, **Anchored at one end**, or **Anchored at both ends** is chosen, select an option from the **Young's modulus** E (SI unit: Pa) and **Wall thickness** Δw lists — **Not set** (the default) or **User defined**. For **User defined** in either case, enter different values or expressions.

For **Anchored at one end** or **Anchored at both ends** also select an option from the **Poisson's ratio** υ (dimensionless) list — **Not set** (the default) or **User defined**. For **User defined** enter a value or expression for Poisson's ratio.

WALL DRAG FORCE

Enter a value or expression for τ_w (SI unit: N/m^2). The default is 0 N/m^2.

FLOW PROFILE CORRECTION FACTOR

Enter a value or expression for β (dimensionless). The default is 1. For most practical applications this correction factor is 1 as the propagating waves are assumed plane and uniform. This value should typically be changed if a wall drag force is introduces or if a nonplug-flow background flow field is used. The flow profile correction factor is defined as

$$\beta = \left(\int(\tilde{\mathbf{u}} + \tilde{\mathbf{u}}_0)^2 dA\right) / \left(\int(\tilde{\mathbf{u}} + \tilde{\mathbf{u}}_0) dA\right)^2$$

where the velocity field is the actual one that exists in the pipe cross section. The factor thus measures the deviation from a flat background flow profile (plug flow) and a plane propagating acoustic wave.

Flow Profile Correction Factor

Volume Force

Use the **Volume Force** node to add a volumetric force acting on the fluid in the pipe. The feature can be used to model sources that generate propagating waves in a pipe system. The force is projected in the tangential direction of the pipe,

VOLUME FORCE

Enter the three components of the **Volume Force** $\mathbf{F}$ (SI unit: N/m^3).

Closed

Use the **Closed** node to impose zero velocity. This is the default condition added on all end points.

Theory for the Pipe Acoustics Boundary Conditions

Pressure

Use the **Pressure** node to define the boundary pressure at the pipe ends.

PRESSURE

Enter a value or expression for the **Pressure** p (SI unit: Pa). The default is 0 Pa.

In the frequency domain p represents the amplitude and phase (if it is complex valued) of a harmonic pressure source.

In the time domain enter an expression for the pressure p, for example, a forward moving sinusoidal wave of amplitude 1 Pa can be written as `1[Pa]*sin(omega*t-k*x)`, where `omega` and `k` are parameters defining the angular frequency and wave number, respectively.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

Theory for the Pipe Acoustics Boundary Conditions

Velocity

Use the **Velocity** node to prescribe a velocity at the pipe ends.

VELOCITY

Enter a value or expression for the velocity u_{in} (SI unit: m/s) at the inlet and/or outlet of a pipe. The default is 0 m/s. The velocity u_{in} is defined relative to background flow u_0 and thus in the tangential coordinate system. Enable the **Show physics symbols** from the **Graphics and Plot Windows** menu in the **Preferences** window in order to visualize the boundary or edge tangent direction. Click the **Fluid Properties** node to see the tangents as a red arrows.

- Theory for the Pipe Acoustics Boundary Conditions
- The Preferences Window in the *COMSOL Multiphysics Reference Manual*

End Impedance

Use the **End Impedance** node to model conditions at the end of a pipe. The condition can either model an infinite pipe and thus represent the characteristic impedance of the pipe system at that point. This results in a zero reflection condition. Alternatively, the condition can represent the radiation impedance of an open pipe in either a flanged (in an infinite baffle) or unflanged (a pipe ending in free open space). The end impedance can also be user defined and could then represent modeled or experimental values for a specific pipe configuration.

The wave speed c in the pipe can be different from the speed of sound c_s in an open space. It then depends on the elastic properties of the pipe structure. It is defined in Equation 9-6 in the Governing Equations section.

The wave speed can be evaluated as `sqrt(1/patd.invc2)` or `sqrt(1/pafd.invc2)` during the analysis and results stage.

END IMPEDANCE

Select an **Impedance model**.

For The Pipe Acoustics, Transient Interface select **Infinite pipe (low Mach number limit)** (the default) or **User defined**. The **Infinite pipe (low Mach number limit)** models and infinite pipe by specifying the characteristic impedance at that point. This condition creates a nonreflecting boundary. The expression is valid for small values of the Mach number $\mathrm{Ma} = u_0/c$. For **User defined** enter an **End impedance** Z_{end} (SI unit: Pa·s/m). The default is `patd.rho*(sqrt(1/patd.invc2))`, which is $\rho \cdot c$.

For The Pipe Acoustics, Frequency Domain Interface select from the following — **Infinite pipe (low Mach number limit)** (the default); **Infinite pipe**; **Flanged pipe, circular**; **Flanged pipe, rectangular**; **Unflanged pipe, circular (low ka limit)**; **Unflanged pipe, circular**; or **User defined**.

- For **Infinite pipe** enter a **Wave number** k (SI unit: rad/m). The default expression is `pafd.omega*(sqrt(pafd.invc2))`. This end impedance models the infinite pipe using the full (nonlinear) dispersion relation. It is valid for all Mach numbers but requires the additional input of the wave number k.
- For **Flanged pipe, circular** enter an **Inner radius** a (SI unit: m). The default expression is `pafd.dh/2`. This end impedance models the radiation impedance of a circular pipe terminated in an infinite baffle. It is an exact analytical result valid for all

frequencies and pipe radii. In the low frequency limit it reduces to the classical results:

$$Z_{\text{end}} = \rho c\left(\frac{1}{2}(ka)^2 + i(0.8216 \cdot ka)\right)$$

- For **Flanged pipe, rectangular** enter an **Inner width** w_{i} (SI unit: m). The default is 5 cm (0.05 m). Also enter an **Inner height** h_{i} (SI unit: m). The default is 10 cm (0.1 m). This end impedance models the radiation impedance of a pipe of rectangular cross section terminated in an infinite baffle. The model is only valid in the low frequency range where $kw_{\text{i}} \ll 1$ and $kh_{\text{i}} \ll 1$.
- For **Unflanged pipe, circular (low ka limit)** or **Unflanged pipe, circular** enter an **Inner radius** a (SI unit: m). The default expression is `pafd.dh/2`. These two end impedance models prescribe the radiation impedance of an unflanged circular pipe (a pipe ending in free open space). The first model is the classical low-frequency approximation valid for $ka \ll 1$. While the second model extends the frequency range to $ka < 3.83$.

For **User defined** enter an **End impedance** Z_{end} (SI unit: Pa·s/m). The default expression is `pafd.rho*(sqrt(1/pafd.invc2))`, which is $\rho \cdot c$.

For a detailed review of the end impedance models see: Theory for the Pipe Acoustics Boundary Conditions.

Theory for the Pipe Acoustics Interfaces

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduces the equations to a 1D component with scalar dependent variables. The present theory assumes no thermal conduction and thus no losses due to thermal conduction (isentropic sound propagation).

In this section:

- Governing Equations
- Theory for the Pipe Acoustics Boundary Conditions
- Solving Transient Problems
- Cutoff Frequency
- Flow Profile Correction Factor
- References for the Pipe Acoustics Interfaces

Governing Equations

The continuity equation derived for a control volume is given by

$$\frac{\partial(A\rho)}{\partial t} + \nabla \cdot (A\rho\mathbf{u}) = 0 \tag{9-1}$$

and the corresponding momentum balance equation is

$$\frac{\partial(\rho A\mathbf{u})}{\partial t} + \nabla(\rho A\beta\mathbf{u}^2) = -A\nabla p - \tau_w Z + A\mathbf{F} \tag{9-2}$$

where Z is the inner perimeter of the pipe and $A = A(x, p, \ldots)$ is the inner wetted cross-sectional area, $\mathbf{u}$ is the area-averaged mean velocity, which is also defined in the tangential direction $\mathbf{u} = u\mathbf{e}_t$, p is the mean pressure along the pipe, τ_w is the wall drag force, and $\mathbf{F}$ is a volume force. The gradient is taken in the tangential direction $\mathbf{e}_t$. The term β is a flow profile correction factor relating the mean of the squared total velocity to the square of the mean velocity. Such that

$$u = \frac{1}{A}\int \tilde{\mathbf{u}} \cdot \mathbf{dA} \qquad p = \frac{1}{A}\int \tilde{p}\, dA \qquad \beta = \left(\frac{1}{A}\left(\int \tilde{u}^2 dA\right)\right) / u^2 \tag{9-3}$$

where

$$\tilde{p} = \tilde{p}(\mathbf{x}) \text{ and } \tilde{u} = \tilde{u}(\mathbf{x})$$

are the local nonaveraged parameters. Again p and u are the area-averaged dependent variables.

LINEARIZATION

The governing equations are now linearized; that is, all variables are expanded to first order assuming stationary zero-order values (steady-state background properties). The acoustic variations of the dependent variables are assumed small and on top of the background values. This is done according to the following scheme:

$$\begin{aligned}
\mathbf{u}(x,t) &= \mathbf{u}_0(x) + \mathbf{u}_1(x,t) \\
p(x,t) &= p_0(x) + p_1(x,t) \\
\rho(x,t) &= \rho_0(x) + \rho_1(x,t) \\
A(x,t) &= A_0(x) + A_1(x,t)
\end{aligned}$$

where A_0 is often only function of $\mathbf{x}$; however, A_0 can be changed by external factors such as heating or structural deformation, thus the time dependency. The first-order terms represent small perturbations on top of the background values (zero order). They are valid for

$$\rho_1 \ll \rho_0 \qquad p_1 \ll \rho_0 c_0^2 \qquad |\mathbf{u}_1| \ll c_0 \qquad A_1 \ll A_0$$

Moreover, the perturbations for the fluid density and cross-sectional area are expanded to first order in p_0 in a Taylor series such that

$$\rho_1 = \rho - \rho_0 = (p - p_0)\left[\frac{\partial \rho}{\partial p}\right]_s\Bigg|_0$$

$$A_1 = A - A_0 = (p - p_0)\left[\frac{\partial A}{\partial p}\right]_s\Bigg|_0$$

where the subscript s refers to constant entropy; that is, the processes are isentropic. The relations for the fluid compressibility and the cross-sectional area compressibility are

$$\beta_0 = \frac{1}{K_0} = \frac{1}{K_s} = \frac{1}{\rho_0}\left[\frac{\partial \rho}{\partial p}\right]_s\Bigg|_0 = \frac{1}{\rho_0}\frac{1}{c_s^2}$$

$$\beta_A = \frac{1}{A_0}\left[\frac{\partial A}{\partial p}\right]_s\Bigg|_0 = \frac{1}{K_A}$$

Here, β_0 is the fluid compressibility at the given reference pressure p_0, the isentropic bulk speed of sound is denoted c_s, and ρ_0 is the fluid density at the given reference temperature and reference pressure. β_A is the effective compressibility of the pipe's cross-sectional A_0 due to changes in the inner fluid pressure. The bulk modulus K is equal to one over the compressibility.

Inserting the above expansions into the governing equations (Equation 9-1 and Equation 9-2) and retaining only first-order terms yield the pipe acoustics equations including background flow. These are:

$$\begin{aligned} A_0\frac{1}{c^2}\frac{\partial p_1}{\partial t} + \nabla\cdot\left(A_0\rho_0\left(\mathbf{u}_1 + \frac{\mathbf{u}_0}{\rho_0 c^2}p_1\right)\right) &= 0 \\ \rho_0 A_0\left(\frac{\partial \mathbf{u}_1}{\partial t} + \frac{\mathbf{u}_0}{\rho_0 c^2}\frac{\partial p_1}{\partial t}\right) + \nabla\left(A_0\beta\frac{\mathbf{u}_0^2}{c^2}p_1 + 2\rho_0 A_0\beta\mathbf{u}_0\mathbf{u}_1\right) & \\ + A_0(\nabla p_1 + p_1\beta_A\nabla p_0) + \tau_w Z - A\mathbf{F} &= 0 \\ \frac{1}{c^2} = \rho_0(\beta_0 + \beta_A) = \rho_0\left(\frac{1}{K_0} + \frac{1}{K_A}\right) &= \frac{1}{c_s^2} + \frac{\rho_0}{K_A} \end{aligned} \tag{9-4}$$

where c is the effective speed of sound in the pipe (it includes the effect due to the elastic properties of the pipe defined through K_A). The bulk modulus for the cross-sectional area K_A is given by the pipe material properties according to the so-called Korteweg formula (see Ref. 2). For a system with rigid pipe walls $c_s = c$ as K_A tends to infinity.

Using the fact that the velocity is taken along the tangential direction $\mathbf{e}_t$, the governing equations are rewritten in terms of the scalar values u and p and projected onto the tangent. The 0 subscript is dropped on the density and area and the 1 subscript is also dropped on the dependent variables.

$$A\frac{1}{c^2}\frac{\partial p}{\partial t} + \nabla_t\left(A\rho\left(u + \frac{u_0}{\rho c^2}p\right)\right)\cdot \mathbf{e}_t = 0$$

$$\rho A\left(\frac{\partial u}{\partial t} + \frac{u_0}{\rho c^2}\frac{\partial p}{\partial t}\right) + \nabla_t\left(A\beta\frac{u_0^2}{c^2}p + 2\rho A\beta u_0 u\right)\cdot \mathbf{e}_t \qquad (9\text{-}5)$$
$$+ A(\nabla_t p + p\beta_A\nabla_t p_0)\cdot \mathbf{e}_t + \tau_w Z - A(\mathbf{F}\cdot \mathbf{e}_t) = 0$$

$$\frac{1}{c^2} = \rho(\beta_0 + \beta_A) = \rho\left(\frac{1}{K_0} + \frac{1}{K_A}\right) = \frac{1}{c_s^2} + \frac{\rho}{K_A}$$

where ∇_t is the tangential derivative, τ_w is the tangential wall drag force (SI unit: N/m^2) and $\mathbf{F}$ is a volume force (SI unit: N/m^3).

GOVERNING EQUATIONS

Pipe Acoustics, Transient Interface

Finally, the expression for the time derivative of the pressure in the momentum equation is replaced by spatial derivatives using the continuity equation. This yields the equations solved in the Pipe Acoustics, Transient interface:

$$A\frac{1}{c^2}\frac{\partial p}{\partial t} + \nabla_t\left(A\rho\left(u + \frac{u_0}{\rho c^2}p\right)\right)\cdot \mathbf{e}_t = 0$$

$$\rho A\frac{\partial u}{\partial t} + \nabla_t\left(A\beta\frac{u_0^2}{c^2}p + 2\rho A\beta u_0 u\right)\cdot \mathbf{e}_t + u_0\nabla_t\left(A\rho\left(u + \frac{u_0}{\rho c^2}p\right)\right)\cdot \mathbf{e}_t \qquad (9\text{-}6)$$
$$+ A(\nabla_t p + p\beta_A\nabla_t p_0)\cdot \mathbf{e}_t + \tau_w Z - A(\mathbf{F}\cdot \mathbf{e}_t) = 0$$

$$\frac{1}{c^2} = \rho(\beta_0 + \beta_A) = \rho\left(\frac{1}{K_0} + \frac{1}{K_A}\right) = \frac{1}{c_s^2} + \frac{\rho}{K_A}$$

Pipe Acoustics, Frequency Domain Interface

In the frequency domain all variables are assumed to be time harmonic such that

$$p = \tilde{p}(\mathbf{x})e^{i\omega t}$$
$$u = \tilde{u}(\mathbf{x})e^{i\omega t} \qquad (9\text{-}7)$$

inserting this into the governing Equation 9-6 (and dropping the tilde) yields the equations solved in the Pipe Acoustics, Frequency Domain interface:

$$i\omega\frac{A}{c^2}p + \nabla_t\left(A\rho\left(u + \frac{u_0}{\rho c^2}p\right)\right)\cdot\mathbf{e}_t = 0$$

$$i\omega\rho A u + \nabla_t\left(A\beta\frac{u_0^2}{c^2}p + 2\rho A\beta u_0 u\right)\cdot\mathbf{e}_t + u_0\nabla_t\left(A\rho\left(u + \frac{u_0}{\rho c^2}p\right)\right)\cdot\mathbf{e}_t \qquad (9\text{-}8)$$
$$+ A(\nabla_t p + p\beta_A\nabla_t p_0)\cdot\mathbf{e}_t + \tau_w Z - A(\mathbf{F}\cdot\mathbf{e}_t) = 0$$

$$\frac{1}{c^2} = \rho(\beta_0 + \beta_A) = \rho\left(\frac{1}{K_0} + \frac{1}{K_A}\right) = \frac{1}{c_s^2} + \frac{\rho}{K_A}$$

where $\omega = 2\pi f$ is the angular frequency and f is the frequency.

Theory for the Pipe Acoustics Boundary Conditions

PRESSURE, OPEN, AND CLOSED CONDITIONS

The simplest boundary conditions to specify are to prescribe the pressure or the velocity at the pipe ends. These result in the Pressure condition

$$p = p_{in}$$

and the Velocity condition

$$u = u_{in}$$

and can be set independently of each other leaving the other dependent variable free. A special subclass of the velocity condition is the Closed condition, where

$$u = 0$$

corresponds to the sound-hard wall condition in pressure acoustics. It is also assumed here that $u_0 = 0$ at a closed boundary.

END IMPEDANCE CONDITION

At the end of pipes the relation between the pressure and the velocity can be defined in terms of an end impedance Z_{end}. The End Impedance condition is in the Pipe Acoustics interface given by

$$A\left(\rho u + \frac{u_0}{c^2}p\right) = A\left(\rho\frac{1}{Z_{end}} + \frac{u_0}{c^2}\right)p \qquad (9\text{-}9)$$

where $Z_{\text{end}} = p/u$ (SI unit: Pa·s/m). Different models for the end impedance exist in the Pipe Acoustics interfaces. The variety depend on if the transient or the frequency domain equations are solved.

Transient End-Impedance Models

In the transient version of the physics interface the end impedance can be user-defined or set to mimic an infinite long pipe for low Mach number background flow conditions. In this case it is assumed that the pipe continues with constant cross section A and that there is no external body force $\mathbf{F}$ and drag τ_w. Because the acoustic waves are, by design, always normal to the pipe ends. In order to define the relation between the pressure and the velocity (the impedance) the dispersion relation for a plane wave needs to be determined.

In order to do so insert the assumed plane waveform

$$p = \text{Re}(\tilde{p}e^{i(\omega t - kx)})$$
$$u = \text{Re}(\tilde{u}e^{i(\omega t - kx)})$$

into the governing Equation 9-6 and solve for the desired relations. After some manipulation this results in

$$\frac{1}{Z_{\text{end}}} = \frac{u}{p} = \frac{1}{c^2}\frac{1}{\rho}\left(\frac{\omega}{k} - u_0\right)$$

with the dispersion relation

$$\frac{\omega}{k} = \beta u_0 \pm c\sqrt{\beta(\beta-1)\left(\frac{u_0}{c}\right)^2 + \left(1 - \frac{1}{k}\beta_A \nabla p_0\right)} \tag{9-10}$$

This dispersion relation is nonlinear in k. In the limit where β_A tends to zero and for small Mach numbers M (= u_0/c), the expression is expanded to

$$\frac{\omega}{k} \cong \beta u_0 \pm c\left(1 + \frac{1}{2}\beta(\beta-1)\left(\frac{u_0}{c}\right)^2\right)$$

Hence, the infinite pipe (low Mach number limit) end impedance relation reads

$$\frac{1}{Z_{\text{end}}} = \frac{1}{c^2}\frac{1}{\rho}\left[(\beta-1)u_0 \pm c\left(1 + \frac{1}{2}\beta(\beta-1)\left(\frac{u_0}{c}\right)^2\right)\right] \tag{9-11}$$

where the sign in front of c depends on the direction of propagation of the wave.

Frequency Domain End-Impedance Models

In the frequency domain many engineering relations exist for the end impedance or radiation impedance of a pipe or waveguide. Most of the relations apply only to a specific geometry or frequency range. The relations available in the Pipe Acoustics, Frequency Domain interface are:

- *Infinite pipe (low Mach number limit)*: This is the same relation as for the transient study and the end impedance is given by Equation 9-11. This can be thought of as the characteristic impedance of the tube.
- *Infinite pipe*: This relation uses the full dispersion relation given in Equation 9-10 and yields the expression

$$\frac{1}{Z_{\text{end}}} = \frac{1}{c^2}\frac{1}{\rho}\left[(\beta-1)u_0 \pm c\sqrt{\beta(\beta-1)\left(\frac{u_0}{c}\right)^2+\left(1-\frac{1}{k}\beta_A\nabla p_0\right)}\right] \tag{9-12}$$

where the wave number k at the right hand side is a user input. In the frequency domain a good estimate for this quantity is simply ω/c.

- *Flanged pipe, circular*: In the case of a circular pipe terminated in an infinite baffle (a flanged pipe) an analytical expression exists for the radiation impedance (see Ref. 1),

$$Z_{\text{end}} = \rho c\left(1-\frac{2J_1(2ka)}{2ka}+i\frac{2H_1(2ka)}{2ka}\right) \tag{9-13}$$

where J_1 is the Bessel function of order 1, H_1 is the Struve function of order 1, a is the pipe radius, and k is the wave number. The Struve function is approximated according to Ref. 3 by

$$H_1(x) \approx \frac{2}{\pi} - J_0(x) + \left(\frac{16}{\pi}-5\right)\frac{\sin x}{x} + \left(12-\frac{36}{\pi}\right)\frac{1-\cos x}{x^2} \tag{9-14}$$

In the low frequency limit (small ka) Equation 9-13 reduces to the classical expression for the radiation impedance

$$Z_{\text{end}} = \rho c\left(\frac{1}{2}(ka)^2 + i(0{,}8216\cdot ka)\right) \tag{9-15}$$

- *Flanged pipe, rectangular*: In the case of a pipe of rectangular cross-section (with sides w_i and h_i) terminated in an infinite baffle (a flanged pipe) the radiation impedance can be approximated by

$$Z_{\text{end}} = \frac{\rho c}{2\pi}\left(k^2(w_i h_i)^2 + ik(w_i h_i)^{3/2} f\left(\frac{w_i}{h_i}\right)\right) \qquad kw_i \ll 1, \quad kh_i \ll 1$$
$$f(x) = 2x^{1/2}\sinh^{-1}x^{-1} + 2x^{-1/2}\sinh^{-1}x + \frac{2}{3}x^{3/2} + \frac{2}{3}x^{-3/2} - \frac{2}{3}(x + x^{-1})^{3/2} \tag{9-16}$$

see Ref. 4 and Ref. 5.

- *Unflanged pipe, circular (low ka limit)*: In the case of a circular pipe of radius a ending in free air the classical low ka limit for the radiation impedance is given by

$$Z_{\text{end}} = \rho c\left(\frac{1}{4}(ka)^2 + i(0{,}6133 \cdot ka)\right) \qquad ka \ll 1 \tag{9-17}$$

see Ref. 1 and Ref. 5.

- *Unflanged pipe, circular*: A solution for the unflanged pipe exists for the case when $ka \ll 3{,}83 = 1{,}22\pi$, it is presented in Ref. and is based on solving the Wiener-Hopf integral, it reads

$$Z_{\text{end}} = \rho c\frac{1+R}{1-R} \qquad R = |R|e^{2ika\delta}$$
$$|R| = e^{-(ka)^2/2}\left(1 + \frac{1}{6}(ka)^4\left[\ln\left(\left(\frac{1}{\gamma ka}\right) + \frac{19}{12}\right)\right]\right) \quad \gamma = e^{0{,}5772} \quad ka < 1 \tag{9-18}$$
$$|R| = \sqrt{\pi ka}e^{-ka}\left(1 + \frac{3}{32}\frac{1}{(ka)^2}\right) \quad 1 < ka < 3{,}83$$

where δ is an interpolation function found by numerical integration for $ka = 0$, $\delta = 0.6133$.

Common for the last four radiation impedance relations is that they do only apply when there is no background flow present $u_0 = 0$ (or at least when it is very small).

Solving Transient Problems

When solving transient acoustic problems where the wave shape is not necessarily harmonic it might be necessary to resolve its spatial variations with a fine mesh, say with a minimal scale dx. Now, in order for the numerical solution of the temporal development of the acoustic field to be good it is necessary to restrict the maximal time steps dt taken by the solver. The condition is known as the CFL condition (Courant–Friedrichs–Lewy condition). For transient acoustic problems it is defined as

$$C = c \cdot \frac{dt}{dx}$$

where C is the Courant number, and c is the velocity.

For applications where all the shape functions are quadratic the Courant number should be around 0.2. This condition restricts any acoustic disturbances to propagate more than 20% of the mesh size dx during one time step dt. In the Pipe Acoustics interface where a mixed formulation exists, with linear elements for the pressure and quadratic elements for the velocity, the condition might have to be tightened such that $C < 0.2$.

For an example where the CFL condition is used see *Water Hammer*: Application Library path **Pipe_Flow_Module/Verification_Examples/water_hammer_verification**.

Cutoff Frequency

The Pipe Acoustics interface assumes plane wave propagation. This means that it cannot model the propagation of the higher order modes that can propagate above their cutoff frequency f^c. In a rectangular pipe of cross-section width w_i and height h_i, the cutoff frequency is

$$f^c_{mn} = \frac{1}{2}c\sqrt{\left(\frac{m}{w_i}\right)^2 + \left(\frac{n}{h_i}\right)^2}$$

In a pipe of circular cross section (with radius a), the cutoff frequency is

$$f^c_{mn} = \frac{\alpha'_{mn}c}{2\pi a}$$

where α'_{mn} is the nth zero of the differential of the Bessel function $J'_m(x)$ or order m. The first few values are $\alpha'_{01} = 0$, $\alpha'_{02} = 3.83$, $\alpha'_{11} = 1.84$, and $\alpha'_{21} = 3.05$ (see Ref. 1 and Ref. 5 for further details).

Flow Profile Correction Factor

The flow profile correction factor, β, accounts for the ratio of the integrated local square velocity field to the square of the integrated local velocity field (see Equation 9-3). It is defined in terms of the total velocity field (background plus acoustic variations).

In the case of no-background flow ($u_0 = 0$) β is 1 in the absence of a wall drag coefficient, as only plane wave modes propagate. If a wall drag force is introduced, to model some loss mechanism, β starts to differ slightly from 1. This can for example be losses introduced to model viscous and thermal effects in narrow pipes.

In the presence of a background u_0 the factor β can be set different 1 in order to model a nonflat velocity profile inside the tube. The value of b (and the actual shape of the background field) influences the convective momentum transfer balances. The places where β enter the governing equations are multiplied with either the Mach number or the Mach number squared, indicating that the effects become important for an increasing background flow.

References for the Pipe Acoustics Interfaces

1. D.T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.

2. M.S. Ghidaoui, M. Zhao, D.A. McInnis, and D.H. Axworthy, "A Review of Water Hammer Theory and Practice", *Applied Mechanics Reviews*, ASME, 2005.

3. R.M. Aarts and A.J.E.M. Janssen, "Approximation of the Struve Function H1 Occurring in Impedance Calculations", *J. Acoust. Soc. Am.*, vol. 113, pp. 2635–2637, 2003.

4. O.A. Lindemann, "Radiation Impedance of a Rectangular Piston at Very Low Frequencies", *J. Acoust. Soc. Am.*, vol 44, pp. 1738–1739, 1968.

5. A.D. Pierce, *Acoustics: An Introduction to its Physical Principles and Applications*, Acoustics Society of America, 1994.

H. Levine and J. Schwinger, "On the Radiation of Sound from an Unflanged Circular Pipe", *Phys. Rev.,* vol. 73, pp. 383–406, 1948.

10

Acoustic Streaming

The Acoustics Module contains multiphysics capabilities to couple acoustics and fluid flow physics interfaces to model acoustic streaming phenomena. Acoustic streaming describes the physical process where an acoustic field can induce movement in a fluid (a flow). This is a nonlinear effect that can be modeled using dedicated multiphysics couplings.

In this chapter:

- Acoustic Streaming Multiphysics
- Modeling Acoustic Streaming
- Theory Background for the Acoustic Streaming Multiphysics

Acoustic Streaming Multiphysics

In this section:

- Acoustic Streaming Multiphysics Interfaces

Acoustic Streaming Multiphysics Interfaces

The **Acoustic Streaming from Pressure Acoustics** and **Acoustic Streaming from Thermoviscous Acoustics** multiphysics interfaces are used to model the physical phenomena acoustic streaming where an acoustic field can induce a fluid flow. The multiphysics couplings Acoustic Streaming Domain Coupling and Acoustic Streaming Boundary Coupling are used to compute the forces, stresses, and boundary slip velocities that the acoustic field induce in the fluid, generating flow. The interfaces couple a frequency domain acoustic field to a stationary or time-dependent fluid flow. The acoustic streaming phenomenon is important in microfluidics and lab-on-a-chip systems for applications such as particle handling, mixing of fluids, and microfluidic pumps.

The **Acoustic Streaming Domain Coupling** is used to compute all the forces and stresses in the bulk of the fluid domain, while the **Acoustic Streaming Boundary Coupling** is used to compute a slip-velocity at a fluid boundary due to boundary contributions. For some simulation models it is sufficient to include either the **Acoustic Streaming Domain Coupling** or **Acoustic Streaming Boundary Coupling** while for other models it is necessary to include both multiphysics couplings, see Modeling Acoustic Streaming for modeling advice.

The coupling features couples a frequency domain acoustics interface to a stationary or time-dependent single-phase fluid flow interface. The acoustic interface can either be the **Pressure Acoustics, Frequency Domain** interface or the **Thermoviscous Acoustics, Frequency Domain** interface, while it has to couple to a single-phase fluid flow interface. Typically the Laminar flow or the Creeping flow interfaces. The interface can couple to a turbulent flow interface but it is advice to take extra caution since the derivation of the source terms does not take turbulent flow into consideration.

The acoustic streaming is a nonlinear phenomenon that occurs due to nonlinearities in the Navier–Stokes equations. When linearizing the Navier–Stokes equations the acoustic fields are the first-order fields in a perturbation theory. The acoustic streaming fields are the time-averaged second-order fields of the perturbation scheme. Thus, the

time-varying second-order fields are neglected together with higher-order terms. Therefore, nonlinear effects as higher-order harmonics and nonlinear behavior of the acoustic field are not included in the derivation of the acoustic streaming fields. For a more detailed introduction to the theory see the section Theory Background for the Acoustic Streaming Multiphysics.

- *Acoustic Streaming in a Microchannel Cross Section*. Application Library path **Acoustics_Module/Nonlinear_Acoustics/acoustic_streaming_microchannel_cross_section**
- *Acoustic Microfluidic Pump*. Application Library path **Acoustics_Module/Nonlinear_Acoustics/acoustic_microfluidic_pump**

Modeling Acoustic Streaming

In this section:

- Sources of Acoustic Streaming
- Acoustic Streaming from Thermoviscous Acoustics
- Acoustic Streaming from Pressure Acoustics
- Lagrangian Velocity Field
- Study Types

Sources of Acoustic Streaming

Acoustic streaming can be induced by different physical mechanisms that are included either in the Acoustic Streaming Domain Coupling or Acoustic Streaming Boundary Coupling, so to understand which multiphysics couplings are important to include, it is necessary to understand the underlying mechanisms. The five main types of acoustic streaming are:

- Rayleigh streaming induced by stresses in the viscous boundary layers. Rayleigh streaming is well known and is often dominant in microfluidic systems with a standing pressure field.
- Eckart streaming induced from viscous damping in the domain of the fluid. Eckart streaming is often important for traveling acoustic fields.
- Streaming due to a vibrating boundary, known as Stokes slip. This is typically important when the wall velocity is comparable to the acoustic velocity in the fluid, which is often the case for systems working off resonance.
- Thermoacoustic streaming (baroclinic streaming) induced when there are gradients in the density and/or compressibility of the fluid. These gradients can be due to either a solvent or temperature gradient (in the background temperature).
- Streaming from sharp edges. Sharp edges are geometry features with a length scale smaller than or similar to the viscous boundary length. This effect can only be modeled when using the Acoustic Streaming from Thermoviscous Acoustics interface.

Acoustic Streaming from Thermoviscous Acoustics

The multiphysics interface **Acoustic Streaming from Thermoviscous Acoustics** couples the **Thermoviscous Acoustics, Frequency Domain** interface with a **Laminar Flow** interface. In the thermoviscous acoustic interface, the viscous and thermal boundary layers are resolved numerically and therefore their streaming contribution is part of the Acoustic Streaming Domain Coupling. Therefore, it is important that the mesh has a good resolution of the boundary layers.

The **Acoustic Streaming Domain Coupling** includes all the effects from the viscous stresses both in the boundary layers and in the domain of the fluid and is therefore often the dominant source of acoustic streaming when coupling from thermoviscous acoustics.

The Acoustic Streaming Boundary Coupling only contains the Stokes-slip contribution and is only important if the boundary of the fluid-solid interfaces is vibrating.

Note that the forces applied on the fluid flow rely on the gradients of the acoustic fields. Therefore, to obtain continuous and smooth derivatives, it is recommended to use a higher element order than the default setting for The Thermoviscous Acoustics, Frequency Domain Interface (see the Discretization section). It is recommended to use quadratic serendipity for the acoustic pressure and cubic serendipity for the acoustic velocity and temperature. This discretization is automatically set when adding the predefined multiphysics interfaces.

Acoustic Streaming from Pressure Acoustics

The **Acoustic Streaming from Pressure Acoustics** multiphysics interface couples the **Pressure Acoustics, Frequency Domain** interface with a **Laminar Flow** interface. The multiphysics interface is appropriate to use when either:

- The viscous boundary layers thickness is a lot smaller than the acoustic wavelength and the geometrical length scales of the system.
- The contribution from the viscous boundary layers can be neglected.

The Acoustic Streaming Domain Coupling includes the bulk contributions to the acoustic streaming field, but importantly not the contribution from the viscous boundary layers. This is different from the coupling from thermoviscous acoustics. So the domain coupling includes the sources that result in Eckart streaming and

thermoacoustic streaming. It is recommended to use the Thermally Conducting and/or Viscous Fluid Model in the pressure acoustics interface to capture all losses. This fluid model is automatically selected when adding the predefined multiphysics interface.

The Acoustic Streaming Boundary Coupling includes the contributions from the viscous boundary layers and the Stokes slip due to vibrations of the boundary. Thus, the **Acoustic Streaming Boundary Coupling** contains the sources that result in traditional Rayleigh streaming. The effects from the viscous boundary layers are included in the slip velocity set by the boundary coupling. It is therefore necessary to use the Thermoviscous Boundary Layer Impedance condition in The Pressure Acoustics, Frequency Domain Interface, so that the correct boundary losses are included in the acoustic model. The **Thermoviscous Boundary Layer Impedance** boundary condition is automatically added when adding the predefined multiphysics interface.

Lagrangian Velocity Field

When modeling acoustic streaming with the multiphysics couplings the flow field in the Laminar Flow interface represents the Eulerian velocity field. When modeling the drag force on a particle the Lagrangian velocity field should be used instead of the Eulerian velocity field. The Lagrangian velocity field is defined as,

$$\mathbf{u}_{\text{Lag}} = \mathbf{u}_{\text{Eul}} + \langle \frac{\mathbf{u}_1}{i\omega} \cdot (\nabla \mathbf{u}_1) \rangle \tag{10-1}$$

where the last term is the so-called Stokes drift. The Acoustic Streaming Domain Coupling defines variables for the Lagrangian velocity field (`asdc1.u_lagx`, `asdc1.u_lagy`, `asdc1.u_lagz`) and a variable for the magnitude of the Lagrangian velocity field (`asdc1.U_lag`). The variables are announced and can easily be selected in the **Drag Force** feature in the Particle Tracing for Fluid Flow physics interface.

The Lagrangian velocity differs from the Eulerian velocity inside the viscous boundary layer and in the case of evanescent, rotating, or traveling acoustic fields.

Acoustic Streaming in a Microchannel Cross Section. Application Library path **Acoustics_Module/Nonlinear_Acoustics/acoustic_streaming_microchannel_cross_section**

Study Types

The multiphysics interfaces **Acoustic Streaming from Thermoviscous Acoustics** and **Acoustic Streaming from Pressure Acoustics**, couples a frequency domain acoustic study to an either stationary or time-dependent fluid flow. If the flow is time dependent, the coupling is only valid if the time-scale of the acoustic field is a lot shorter than the characteristic time-scale of the fluid flow. This requirement is fulfilled in microfluidic devices ran at typical frequencies used in microfluidics (MHz to GHz).

The simplest study configuration is to have two study steps. First a **Frequency Domain** study step that includes the acoustic fields, and secondly a **Stationary** study step that includes the fluid-flow interface and the multiphysics coupling. The **Stationary** study step needs to use the solution from the **Frequency Domain** study step for **Values for variables not solved for**. This is a one-way coupling from the acoustic field to the fluid flow.

For studying the transient behavior of acoustic streaming there are two possibilities. First if the acoustic pressure field is constant, the solver can be set up as the previous stationary study. First a **Frequency Domain** study step that includes the acoustic fields, and then a **Time-dependent** study step that includes the fluid flow interface and the multiphysics couplings. The **Time-dependent** study step needs to use the solution from the **Frequency Domain** study step for the **Values for variables not solved for**. If the acoustic pressure field changes with time, due to changes in actuation or changes in material properties due to heating, then the **Frequency-Transient** study step can be used. It is important to note that all changes in the acoustic fields has to be on a time-scale a lot longer than the acoustic time-scale. The **Frequency-Transient** study solves for the acoustic fields in each time step of the transient solver.

Theory Background for the Acoustic Streaming Multiphysics

The field of acoustic streaming deals with the steady fluid flow induced by an acoustic field. This is a well-known effect described by Lord Rayleigh (Ref. 1) who explained the source of acoustic streaming induced by an acoustic field near a boundary. The modern research on acoustic streaming is focused on applications such as microfluidic devices for handling particles Ref. 2, mixing fluids, and forced convective cooling Ref. 3.

In the following sections, a short introduction to the theoretical background of acoustic streaming is presented. The theory is based on the work in Ref. 2, Ref. 4, Ref. 6, and Ref. 7.

In this section:

- Governing Equations
- Perturbation Theory
- Effective Theory
- References

Governing Equations

The behavior of the fluid is described by dependent variables for the pressure p and velocity $\mathbf{v}$. In the fluid, the density ρ is determined by the pressure p and temperature T through the equation of state,

$$\frac{1}{\rho}\mathrm{d}\rho = \kappa_T \mathrm{d}p - \alpha_p \mathrm{d}T \tag{10-2}$$

where κ_T is the isothermal compressibility and α_p is the (isobaric) coefficient of thermal expansion. The pressure and velocity fields are governed by the transport equations for mass and momentum. The continuity equation governs the conservation of mass,

$$\frac{\partial \rho}{\partial t} = -\nabla \cdot (\rho \mathbf{v}) \tag{10-3}$$

while the Navier–Stokes equation governs the conservation of momentum,

$$\frac{\partial}{\partial t}(\rho \mathbf{v}) = \nabla \cdot (\sigma - \rho \mathbf{v}\mathbf{v}) \tag{10-4}$$

Here σ is the stress tensor as is given as

$$\sigma = -p\mathbf{I} + \tau \tag{10-5}$$

Where the pressure is on the diagonal and τ is the viscous stress tensor.

Perturbation Theory

In acoustics it is typical to linearize the continuity equation Equation 10-2 and the Navier–Stokes equation Equation 10-3 to derive a set of governing equations for the acoustic field. The linearization of the governing equations is the first-order equations of a perturbation scheme. Acoustic streaming is the time-averaged second-order equations of the perturbations scheme. Thus, a field can be described by the background zeroth-order field, first-order acoustic field and time-averaged second-order field. The pressure and velocity fields are decomposed as

$$\begin{aligned} p(\mathbf{x}, t) &= p_0 + \mathrm{Re}\{p_1(\mathbf{x})\exp(i\omega t)\} + p_2(\mathbf{x}, t) \\ \mathbf{v}(\mathbf{x}, t) &= \mathrm{Re}\{\mathbf{v}_1(\mathbf{x})\exp(i\omega t)\} + \mathbf{v}_2(\mathbf{x}, t) \end{aligned} \tag{10-6}$$

and likewise for all physical fields. In the derivation, it is assumed that the background flow is zero so $\mathbf{v}_0 = 0$.

The time-averaged second-order equations consist of the second-order fields and the time-averaged products of the acoustic first-order fields. The products of first-order fields have the form (Ref. 2)

$$\langle A_1 B_1 \rangle = (1/2)\mathrm{Re}\{A_1 B^*{}_1\} \tag{10-7}$$

The acoustic source terms in the second-order contributions appear in both the continuity equation as a mass source term Q_m, in Navier–Stokes equations as an acoustic body force $\mathbf{f}_\mathrm{aco}$, and on the boundary condition for the velocity field as a slip-velocity $\mathbf{v}_\mathrm{slip}$.

The second-order equation for Equation 10-3 is given as

$$\frac{\partial \rho_2}{\partial t} = -\nabla \cdot (\rho_0 \mathbf{v}_2) + Q_m \tag{10-8}$$

$$Q_m = -\nabla \cdot \langle \rho_1 \mathbf{v}_1 \rangle \tag{10-9}$$

where Q_m is the acoustic source term to the second-order fluid equations. The second-order equation for the Navier–Stokes equations (Equation 10-4) is given as

$$\frac{\partial}{\partial t}(\rho_0 \mathbf{v}_2) = \nabla \cdot \sigma_2 + \mathbf{f}_{aco} \tag{10-10}$$

$$\mathbf{f}_{aco} = \nabla \cdot \tau_{11} - \nabla \cdot \langle \rho_0 \mathbf{v}_1 \mathbf{v}_1 \rangle \tag{10-11}$$

Here, the acoustic body force appears as the divergence of the momentum flux tensor and the second order stress tensor that depends on the acoustic fields τ_{11}. The stress tensor τ_{11} is given as

$$\tau_{11} = \langle \mu_1 (\nabla \mathbf{v}_1 + (\nabla \mathbf{v}_1)^T) \rangle \tag{10-12}$$

Here, μ_1 is the acoustic perturbation for the viscosity due to the acoustic pressure and thermal field. The contribution from τ_{11} is only included if the checkbox **Include first order viscosity terms** is selected. The acoustic perturbation of the viscosity is given as

$$\mu_1 = \frac{\partial \mu}{\partial p} p_1 + \frac{\partial \mu}{\partial T} T_1 \tag{10-13}$$

The last acoustic contribution to the fluid flow equation is the addition of a slip velocity due to the vibrations of the boundary. This occurs because the no-slip boundary condition should be enforced on the vibrating boundary. For a boundary at stationary position $\mathbf{s}_0$ and acoustic vibration of $\mathbf{s}_1$ the no-slip condition becomes

$$\mathbf{v}_2(\mathbf{s}_0) + s_1 \cdot \nabla \mathbf{v}_1 = \mathbf{v}_{wall} \tag{10-14}$$

This gives a slip velocity between the fluid and the wall, known as Stokes slip.

When coupling from **Thermoviscous Acoustics, Frequency Domain**, the contributions to the fluid flow equations is the mass source term Q_m, the acoustic body force $\mathbf{f}_{aco}$, and the Stokes slip. The fluid flow described in Equation 10-10 is a **Creeping Flow** since the inertial term is neglected. It is possible to use separation of timescales instead of perturbation theory to derive the governing equations, in this case the inertial terms are present, so it is mathematical sound to couple to the **Laminar Flow** interface, see Ref. 7.

Effective Theory

When coupling from **Pressure Acoustics, Frequency Domain**, an effective theory is used to include the effects of viscous boundary layers. It is a theory that has been developed

for modeling acoustics in microfluidic devices, a research area called Acoustofluidics, see Ref. 4, Ref. 6, and Ref. 7. The theory solves the acoustic fields with pressure acoustics and use the Thermoviscous Boundary Layer Impedance boundary condition to include the effects from the viscous boundary layers. It then computes the force contribution from the viscous boundary layers from an analytical expression and impose it as slip velocity on the boundary. This model is valid if the viscous boundary layer thickness is a lot smaller than the acoustic and the characteristic geometry length scales. The benefits of this model is that it is not necessary to numerically resolve the viscous boundary layers.

The acoustic sources are then a slip velocity including both the Stokes slip and the viscous stresses from the boundary layers and a bulk body force containing the acoustic source terms related to the bulk of the domain. The bulk body force can be simplified because the acoustic velocity is given by the pressure field and the term Q_m can be neglected, see Ref. 4. The second-order equations for the continuity and Navier–Stokes equations become

$$\frac{\partial \rho_2}{\partial t} = -\nabla \cdot (\rho_0 \mathbf{v}_2) \tag{10-15}$$

$$\frac{\partial}{\partial t}(\rho_0 \mathbf{v}_2) = \nabla \cdot \sigma_2 + \mathbf{f}_{\text{aco}} \tag{10-16}$$

$$\mathbf{f}_{\text{aco}} = -\frac{1}{4}|p_1|^2 \nabla \kappa_s - \frac{1}{4}|\mathbf{v}_1|^2 \nabla \rho_0 + \frac{\Gamma \omega}{c^2}\langle \mathbf{v}_1 p_1 \rangle + \nabla \cdot \tau_{11} \tag{10-17}$$

$$\mathbf{v}_2(\mathbf{s}_0) = \mathbf{v}_{\text{slip}}(\mathbf{s}_0) + \mathbf{v}_{\text{wall}}(\mathbf{s}_0) \tag{10-18}$$

The acoustic body force $\mathbf{f}_{\text{aco}}$ contains the term that gives rise to traditional Eckart streaming, the stress tensor due to the acoustic perturbation of the viscosity and terms that depend on the gradient of the material parameters (Ref. 6). The gradients in material parameters can either be caused by a solvent (Ref. 5) or a temperature gradient (Ref. 8) and are the source terms for thermoacoustic and baroclinic streaming. The contribution from τ_{11} is only included if the checkbox **Include first order viscosity terms** is checked. The slip velocity $\mathbf{v}_{\text{slip}}$ is given by the acoustic pressure field p_1.

The effective theory is used when coupling from **Pressure Acoustics, Frequency Domain**. It has been validated against a full model and experiments in the research area of acoustics in microfluidic devices, see Ref. 4, Ref. 6, and Ref. 8. For more information on the theoretical derivation see Ref. 4 and Ref. 6.

References

1. Lord Rayleigh, "On the circulation of air observed in Kundt' tubes, and on some allied acoustical problems," *Phil. Trans. R. Soc. London*, vol. 175, p. 1, 1884.

2. P.B. Muller and H. Bruus, "Numerical study of thermoviscous effects in ultrasound-induced acoustic streaming in microchannels," *Phys. Rev. E*, vol. 90, p. 043016, 2014.

3. G. Michel and C. Gissinger, "Cooling by Baroclinic Acoustic Streaming", *Phys. Rev. Applied*, vol. 16, p. L051003, 2021.

4. J. S. Bach and H. Bruus, "Theory of pressure acoustics with viscous boundary layers and streaming in curved elastic cavities," *J. Acoust. Soc. Am.*, vol. 144, pp. 766–784, 2018.

5. J. T. Karlsen, P. Augustsson, and H. Bruus, "Acoustic Force Density Acting on Inhomogeneous Fluids in Acoustic Fields," *Phys. Rev. Lett.*, vol. 117, p. 114504, 2016.

6. J. H. Joergensen and H. Bruus, "Theory of pressure acoustics with thermoviscous boundary layers and streaming in elastic cavities," *J. Acoust. Soc. Am.*, vol. 149, pp. 3599–3610, 2021

7. J. H. Joergensen, *Theory and modeling of thermoviscous acoustofluidics*, PhD Thesis, Technical University of Denmark, 2022.

8. W. Qiu and others, "Fast Microscale Acoustic Streaming Driven by a Temperature-Gradient-Induced Nondissipative Acoustic Body Force," *Phys. Rev. Lett.*, vol. 127, p. 06450, 2021.

11

Multiphysics Couplings

This chapter contains a description of all the Multiphysics coupling features of the Acoustics Module. Several of the coupling features are used in predefined multiphysics interfaces but can as well be added manually when modeling multiphysics systems.

- Coupling Features
- Predefined Multiphysics Interfaces
- Modeling with Multiphysics Couplings

Coupling Features

The Acoustics Module contains a number of Multiphysics Coupling features that can couple acoustic domains to structures and porous materials, thermoviscous acoustics to pressure acoustics, FEM to BEM, a flow to an aeroacoustic interface (convected acoustics), or an aeroacoustic flow source to pressure acoustics. The coupling features are described below.

The available couplings features are:

- Acoustic–Structure Boundary
- Pair Acoustic–Structure Boundary
- Thermoviscous Acoustic–Structure Boundary
- Pair Thermoviscous Acoustic–Structure Boundary
- Thermoviscous Acoustic–Thermoelasticity Boundary
- Thermoviscous Acoustic–Thermal Perturbation Boundary
- Thermoviscous Acoustic SLNS-Structure Boundary
- Aeroacoustic–Structure Boundary
- Acoustic–Thermoviscous Acoustic Boundary
- Pair Acoustic–Thermoviscous Acoustic Boundary
- Acoustic–Porous Boundary
- Porous–Structure Boundary
- Background Potential Flow Coupling
- Background Fluid Flow Coupling
- Aeroacoustic Flow Source Coupling
- Acoustic FEM–BEM Boundary
- Acoustic–Pipe Acoustic Connection
- Acoustic–Structure Boundary, Time Explicit
- Pair Acoustic–Structure Boundary, Time Explicit
- Convected Acoustic–Structure Boundary, Time Explicit
- Pair Convected Acoustic–Structure Boundary, Time Explicit
- Piezoelectricity, Time Explicit
- Piezoelectricity

- Magnetomechanics
- Acoustic Streaming Domain Coupling
- Acoustic Streaming Boundary Coupling

The available subfeatures to multiphysics couplings:

- Thermoviscous Boundary Layer Impedance (subfeature)
- Impedance (subfeature)

Acoustic–Structure Boundary

The **Acoustic–Structure Boundary** coupling ( ) is used to couple a *Pressure Acoustics* model to any structural component. This includes both the FEM based acoustics interfaces and the BEM based acoustics interface. The feature couples to **Solid Mechanics**, **Shell**, **Layered Shell**, **Membrane**, and **Multibody Dynamics** interfaces.

The coupling includes the fluid load on the structure and the structural acceleration as experienced by the fluid. For thin interior structures like shells or membranes with fluid on both sides, a slit is added to the pressure variable and care is taken to couple the up and down sides.

Mathematically the condition on exterior boundaries reads

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right) = -\mathbf{n}\cdot\mathbf{u}_{tt}$$

$$\mathbf{f}_a = p_t\mathbf{n}$$

where $\mathbf{u}_{tt}$ is the structural acceleration, $\mathbf{n}$ is the surface normal, p_t is the total acoustic pressure and $\mathbf{f}_a$ is the acoustic load (force per unit area) experienced by the structure. On interior boundaries the condition reads

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{up} = -\mathbf{n}\cdot\mathbf{u}_{tt}$$

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t-\mathbf{q}_d)\right)_{down} = -\mathbf{n}\cdot\mathbf{u}_{tt}$$

$$\mathbf{f}_a = (p_{t,down}-p_{t,up})\mathbf{n}$$

such that the acoustic load is given by the pressure drop across the thin structure. The up and down subscripts refer to the two sides of the interior boundary.

For coupling to the **Layered Shell**, the interior condition is slightly different. Here the specific accelerations on the up- and downsides are used.

The effects of losses due to thermal and viscous dissipation, in the acoustic boundary layers at the wall where the coupling is applied, can be including using the Thermoviscous Boundary Layer Impedance (subfeature).

	In 2D components it is possible to define the out-of-plane thickness for the structural interfaces while in acoustics this thickness effectively is unity (set to 1 m). A rescaling is automatically performed for the pressure load to account for this difference in interpretation in 2D.

	In models with large deformations of the structure, for example, if a membrane is prestressed and deforms, the resulting area change is automatically included when using the multiphysics coupling between the structure and the acoustic domain.

	The interior version of the coupling is not applicable when coupling **Shell** or **Membrane** to the **Pressure Acoustics, Boundary Element** physics interface. For this type of application, add a domain around the thin structure where regular **Pressure Acoustics, Frequency Domain** is used, then also add the Acoustic FEM–BEM Boundary.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `asb1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces.

The default values depend on how this node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when a multiphysics interface is selected in the **Model Wizard** or **Add Physics** window, then the two participating interfaces are selected.

You can also select **None** from either list to uncouple this node from a physics interface. If the physics interface is removed from the **Model Builder**, for example if **Pressure Acoustics, Frequency Domain** is deleted, then the **Acoustics** list defaults to **None** as there is nothing to couple to.

If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the **Acoustics** or **Structure** lists. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

Thermoviscous Boundary Layer Impedance (subfeature)

The **Thermoviscous Boundary Layer Impedance** subnode can be added as a subfeature to the Acoustic–Structure Boundary multiphysics coupling, in order to include the losses associated with the thermal and viscous boundary layers. The subfeature can be applied to all or to a subset of the boundaries in the parent multiphysics coupling. For details about the formulation and applicability of this feature see the Thermoviscous Boundary Layer Impedance documentation for The Pressure Acoustics, Frequency Domain Interface.

In this version of the feature, applied to the multiphysics coupling, there is no **Mechanical Condition** section as it is per default fully coupled to the vibrating surface. The velocity of the structure is applied to the fluid and the full mechanical traction, including viscous stresses components, is added to the solid. The details about the settings for the **Thermal Condition** and the **Fluid Properties** sections are found in the Thermoviscous Boundary Layer Impedance section.

The thermoviscous boundary layer impedance subfeature is used in the tutorial *Piezoelectric MEMS Speaker*. Application Library path **Acoustics_Module/Electroacoustic_Transducers/piezo_mems_speaker**

Pair Acoustic–Structure Boundary

The **Pair Acoustic–Structure Boundary** coupling ( ) is used to couple the **Pressure Acoustics, Frequency Domain** or the **Pressure Acoustics, Transient** interface to the **Solid Mechanics** interface in an assembly geometry where identity pairs have been created. This allows the use of a nonconforming mesh at the acoustic-structure boundary. As the wave speeds differ in solids and fluids, the computational mesh can take advantage of this when resolving the waves and in this way saving degrees of freedom when solving.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces.

PAIR SELECTION

Add and select the identity pairs where the pair multiphysics coupling feature is applied.

Thermoviscous Acoustic–Structure Boundary

The **Thermoviscous Acoustic–Structure Boundary** coupling ( ) is used to couple a *Thermoviscous Acoustics* interface to any structural component. The feature couples to **Solid Mechanics**, **Shell**, **Layered Shell**, **Membrane**, and **Multibody Dynamics** interfaces.

The coupling has two options for the mechanical components. It either prescribes continuity in the full displacement field, with the no-slip options, given by

$$\mathbf{u}_{\mathrm{t,fluid}} = i\omega\mathbf{u}_{\mathrm{solid}} \qquad \text{or} \qquad \mathbf{u}_{\mathrm{t,fluid}} = \frac{\partial \mathbf{u}_{\mathrm{solid}}}{\partial t}$$

or continuity only in the normal displacement, given by

$$\mathbf{n}\cdot\mathbf{u}_{\mathrm{t,fluid}} = i\omega(\mathbf{n}\cdot\mathbf{u}_{\mathrm{solid}}) \qquad \text{or} \qquad \mathbf{n}\cdot\mathbf{u}_{\mathrm{t,fluid}} = \mathbf{n}\cdot\frac{\partial \mathbf{u}_{\mathrm{solid}}}{\partial t}$$

where $\mathbf{u}_{\mathrm{t,fluid}}$ is the total fluid velocity (including a background component if applicable), $\mathbf{u}_{\mathrm{solid}}$ is the solid displacement, and $\mathbf{n}$ is the surface normal. The first equation is in both cases in the frequency domain and the second is in the time domain. This coupling results in the stress (or normal stress) also being continuous across the boundary. The condition for the total temperature T_{t} can be set to either isothermal or adiabatic. In the case where a shell or membrane is interior to the thermoviscous

acoustics domain, a slit is automatically applied to the pressure p and temperature T degrees of freedom (DOFs).

The coupling has a special formulation for eigenfrequency studies, which allows the program to find the eigenvalue for coupled systems.

For an example using this multiphysics coupling see the *Vibrating Micromirror with Viscous and Thermal Damping* model: Application Library path **Acoustics_Module/Vibrations_and_FSI/vibrating_micromirror**

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `tsb1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Thermoviscous Acoustics** and **Structure** lists include all applicable physics interfaces. See the Coupled Interfaces in Acoustic–Structure Boundary for details.

MECHANICAL

Select the **Mechanical condition** as **No slip** (the default) or **Slip (perfect)**. For the no-slip option all the structure and fluid velocity/displacement components are coupled. This condition also results in the generation of a viscous boundary layer (which should be resolved by the mesh). For the slip (perfect) option, the velocity/displacement is only coupled in the normal direction to the surface; this condition is equivalent to the coupling in pressure acoustics and no viscous boundary layer is modeled. The slip (perfect) condition can be used for larger systems that have a geometric scale large then the viscous boundary layer thickness.

THERMAL

Select the **Thermal condition** that applies on the solid surface to be either **Isothermal** (the default selected) or **Adiabatic**. The isothermal condition results in the generation of a thermal boundary layer, while the adiabatic does not. Use the adiabatic option for larger models that have a geometric scale larger than the thermal boundary layer thickness.

CONSTRAINT SETTINGS

These settings are shown when the **Advanced Physics Options** is selected. It controls how the constraints (continuity in displacement and the thermal condition) are handled. For the **Mechanical constraints type**, select either **Study controlled** (the default), **Weak constraints**, **Nitsche constraints**, or **Pointwise constraints**. For the **Thermal constraint type**, select either **Weak constraints**, **Pointwise constraint** (the default) or **Nitsche constraints**. Finally, select the **Constraint method** to apply to the pointwise constraints, either **Elemental** or **Nodal** (the default).

When the **Mechanical constraints type** is set to **Study controlled**, the weak constraints are automatically selected when an eigenfrequency study is performed. This type of coupling is necessary as the eigenvalue (the angular frequency ω) enters the coupling expression. This will create extra variables at the boundary (so-called Lagrange multipliers), ensuring the correct behavior and solution. For a normal frequency domain study, the pointwise constraint is automatically selected. In the time domain there are no options, and the weak formulation is always used.

At very large frequencies (larger than about 15.9 MHz) the default constraint settings in a frequency domain model can lead to incorrect results for the default **Nodal** option in the **Constraint method**. This is due to an optimized code being used for the Nodal formulation, where constraints, that have a large order of magnitude in difference, are removed. An internal scaling, of the governing equations, can lead to this issue. A solver warning "`New constraint force nodes detected: These are not stored. This only affects the reacf() operator in postprocessing`" will be issued. To remedy this switch to the **Elemental** formulation for the **Constraint method**.

Pair Thermoviscous Acoustic–Structure Boundary

The **Pair Thermoviscous Acoustic–Structure Boundary** coupling ( ) is used to couple the **Thermoviscous Acoustics, Frequency Domain** or the **Thermoviscous Acoustics, Transient** interface to the **Solid Mechanics** interface in an assembly geometry where identity pairs have been created. This allows the use of a nonconforming mesh at the acoustic-structure boundary. As the wave speeds differ in solids and fluids, the computational mesh can take advantage of this when resolving the waves. In this way saving degrees of freedom when solving.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces.

THERMAL

Select a condition for the temperature variations either **Isothermal** (the default) or **Adiabatic**.

CONSTRAINT SETTINGS

These settings are shown when the **Advanced Physics Options** is selected. It controls how the constraints (continuity in displacement and the thermal condition) are handled. For the **Mechanical constraints type**, select either **Study controlled** (the default), **Nitsche constraints**, or **Pointwise constraints**. For the **Thermal constraint type**, select either **Pointwise constraint** (the default) or **Nitsche constraints**. Finally, select the **Constraint method** to apply to the pointwise constraints, either **Elemental** or **Nodal** (the default).

When the **Mechanical constraints type** is set to **Study controlled**, the weak constraints are automatically selected when an eigenfrequencies study is performed. This type of coupling is necessary as the eigenvalue (the angular frequency ω) enters the coupling expression. This will create extra variables at the boundary (so-called Lagrange multipliers), ensuring the correct behavior and solution. For a normal frequency domain study, the pointwise constraint is automatically selected. In the time domain a penalty/Nitsche formulation is always used automatically.

PAIR SELECTION

Add and select the identity pairs where the pair multiphysics coupling feature is applied.

Thermoviscous Acoustic–Thermoelasticity Boundary

The **Thermoviscous Acoustic–Thermoelasticity Boundary** coupling ( ) is used to couple **Thermoviscous Acoustics, Frequency Domain** (or **Transient**) in a fluid to the combined **Solid Mechanics** and **Heat Transfer in Solids** in a solid domain. The **Thermal Expansion** multiphysics coupling is required in the solid to model the coupled **Thermoelasticity** multiphysics phenomenon. The Thermoelasticity Interface exists as a predefined multiphysics interface under the **Structural Mechanics** > **Thermal–Structure Interaction** branch when adding a physics interface using the Model Wizard or Add Physics windows.

This multiphysics coupling allows modeling the combined thermoviscous losses in acoustic-structure interaction problems to great detail. In particular, the effect of a nonideal thermal condition at the fluid-structure interface can be captured. This is of particular interest in, for example, MEMS systems; or in systems where the thermal properties of the fluid and the solid are similar to some extent, for example, in a water-glass systems.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Thermoviscous acoustics**, **Structure**, and **Thermal perturbation** lists include all applicable physics interfaces. For the **Thermoviscous acoustics**, both the **Thermoviscous Acoustics, Frequency Domain** and **Thermoviscous Acoustics, Transient** interfaces can be selected.

MECHANICAL

Select the **Mechanical condition** as **No slip** (the default) or **Slip (perfect)**.

The **Thermoviscous Acoustic–Thermoelasticity Boundary** coupling is only applicable for *Frequency-Domain Perturbation* (possibly combined with a *Stationary* study) and *Time Dependent* studies. This is the case as the Heat Transfer in Solids interface is not formulated in terms of perturbations as all acoustic interfaces are. This typically requires solving for the stationary temperature field in the solid, which then forms the linearization point for the acoustic model. The temperature linearization point can also be taken from the **Initial expression**. The stationary temperature field can be selected in **Model Input** for the acoustic model to get consistent material data.

See The Thermoviscous Acoustic–Thermoelasticity Interaction Multiphysics Interfaces section in the Thermoviscous Acoustics Interfaces chapter for more details.

Prestressed Micromirror Vibrations: Thermoviscous–Thermoelasticity Coupling. Application Library path **Acoustics_Module/Vibrations_and_FSI/micromirror_prestressed_vibrations**

Thermoviscous Acoustic–Thermal Perturbation Boundary

The **Thermoviscous Acoustic–Thermal Perturbation Boundary** coupling ( ) is used to couple the temperature variation field in the **Thermoviscous Acoustics, Frequency Domain** (or **Transient**) interface to the **Heat Transfer in Solids** or **Heat Transfer in Fluids** interfaces. Typically, the coupling is to the **Heat Transfer in Solids** interface to get a physically consistent model. The multiphysics coupling should always be combined with a mechanical condition on the acoustic side. If the **Adiabatic formulation** is used in thermoviscous acoustics, no coupling is performed.

This multiphysics coupling makes it possible to model the effect of a nonideal thermal condition at the fluid-solid interface. This effect is of particular interest in, for example, MEMS systems or systems where the thermal properties of the fluid and the solid are similar to some extent (water-glass systems, for example).

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Thermoviscous acoustics** and **Thermal perturbation** lists include all applicable physics interfaces.

The **Thermoviscous Acoustic–Thermal Perturbation Boundary** coupling is only applicable for *Frequency-Domain Perturbation* (possibly combined with a *Stationary* study) and *Time Dependent* studies. This is the case as the Heat Transfer in Solids (and Fluids) interface is not formulated in terms of perturbations as all acoustic interfaces are. This typically requires solving for the stationary temperature field in the solid, which then forms the linearization point for the acoustic model. The temperature linearization point can also be taken from the **Initial expression**. The stationary temperature field can be selected as a **Model Input** for the acoustic model to get consistent material data.

Thermoacoustic Engine and Heat Pump. Application Library path **Acoustics_Module/Nonlinear_Acoustics/thermoacoustic_engine_heat_pump**

Thermoviscous Acoustic SLNS-Structure Boundary

The **Thermoviscous Acoustic SLNS-Structure Boundary** coupling ( ) is used to couple a *Thermoviscous Acoustics, SLNS Approximation* interface to any structural

component. The feature couples to the **Solid Mechanics**, **Shell**, **Layered Shell**, **Membrane**, and **Multibody Dynamics** interfaces.

The coupling includes the fluid load on the structure and the structural acceleration as experienced by the fluid. For thin interior structures like shells or membranes with fluid on both sides, a slit is added to the pressure variable and care is taken to couple the up and down sides. This is done as for the Acoustic–Structure Boundary. The multiphysics coupling also sets up all relevant conditions for the viscous scaling function Ψ_v and the thermal scaling function Ψ_{th}, on the boundary.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `tssb1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces.

MECHANICAL

Select the **Mechanical condition** as **No slip** or **Slip (perfect)** (the default). The default for this multiphysics coupling is **Slip (perfect)**, which gives a model with no viscous boundary layer on the coupled wall. For models with geometric dimensions comparable to the viscous boundary layer thickness use the no-slip option. For **No slip**, the formulation — used in the SLNS formulation for thermoviscous acoustics — can result in a solution that looks unphysical near the moving wall. However, these artifacts are only local effects that do not influence the overall results.

THERMAL

Select the **Thermal condition** as **Isothermal** or **Adiabatic** (the default).

CONSTRAINT SETTINGS

These settings are shown when the **Advanced Physics Options** is selected. It controls how the constraints are handled.

Aeroacoustic–Structure Boundary

The **Aeroacoustic–Structure Boundary** coupling ( ) is used to couple an *Aeroacoustic* interface (it only applies to the *Linearized Navier–Stokes* interfaces) to any structural component. The feature couples to **Solid Mechanics**, **Shell**, **Layered Shell**, **Membrane**, and

Multibody Dynamics interfaces. The coupling can be used to, for example, model fluid–structure interaction (FSI) in the frequency domain.

The coupling has two options for the mechanical components. It either prescribes continuity in the full displacement field, with the no-slip options, given by

$$\mathbf{u}_{\mathrm{t,fluid}} = i\omega\mathbf{u}_{\mathrm{solid}} \qquad \text{or} \qquad \mathbf{u}_{\mathrm{t,fluid}} = \frac{\partial \mathbf{u}_{\mathrm{solid}}}{\partial t}$$

or continuity only in the normal displacement, given by

$$\mathbf{n}\cdot\mathbf{u}_{\mathrm{t,fluid}} = i\omega(\mathbf{n}\cdot\mathbf{u}_{\mathrm{solid}}) \qquad \text{or} \qquad \mathbf{n}\cdot\mathbf{u}_{\mathrm{t,fluid}} = \mathbf{n}\cdot\frac{\partial \mathbf{u}_{\mathrm{solid}}}{\partial t}$$

where $\mathbf{u}_{\mathrm{t,fluid}}$ is the total fluid velocity (including a background component if applicable), $\mathbf{u}_{\mathrm{solid}}$ is the solid displacement, and $\mathbf{n}$ is the surface normal. The first equation is in both cases the frequency domain and the second is in the time domain. This coupling results in the stress (or normal stress) also being continuous across the boundary. The condition for the total temperature T_{t} can be set to either isothermal or adiabatic. In the case where a shell or membrane is interior to the thermoviscous acoustics domain, a slit is automatically applied to the pressure p and temperature T degrees of freedom (DOFs).

The coupling has a special formulation for eigenfrequency studies, which allows the user to find the eigenvalue for coupled systems.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `aesb1`.

THERMAL

Select the **Thermal condition** that applies on the solid surface to be either **Isothermal** (the default selected) or **Adiabatic**.

MECHANICAL

Select the **Mechanical condition** as **No slip** (the default) or **Slip (perfect)**.

CONSTRAINT SETTINGS

See Constraint Settings under the Thermoviscous Acoustic–Structure Boundary.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Aeroacoustics** and **Structure** lists include all applicable physics interfaces. See the Coupled Interfaces in Acoustic–Structure Boundary for details.

Acoustic–Thermoviscous Acoustic Boundary

The **Acoustic–Thermoviscous Acoustic Boundary** coupling ( ) is used to couple a *Thermoviscous Acoustics* interface to a *Pressure Acoustics* interface (in both frequency and time domain). This includes both the FEM-based acoustics interfaces and the BEM-based acoustics interface. As it is only necessary to solve the full detailed thermally conducting and viscous model near walls in the boundary layer region, it makes sense to switch to classical pressure acoustics outside this region. This approach saves a lot of memory and solution time due to the reduced number of degrees of freedom.

Note that when using the coupling at places where pressure acoustics and thermoviscous acoustics interact at a common wall, the coupling becomes unphysical if no-slip and isothermal conditions are used in the thermoviscous domain. The reason is that the conditions at the wall must match between the physics. The coupling assumes adiabatic behavior and normal-stress coupling only, which is not valid in the thermal and viscous boundary layers. The introduced error depends on the model size and the thickness of the acoustic boundary layer. If it makes physical sense, then use the slip and adiabatic options on that specific wall (add an extra wall boundary condition). In general, using the coupling in a cross section of a waveguide will not be physically correct if the boundary layer effects are of importance.

The coupling prescribes continuity in the total normal stress (dynamic condition) and the total normal acceleration for the mechanical part (kinematic condition). An adiabatic condition is prescribed for the total temperature to match the physical assumptions of the pressure acoustics formulation. The coupling reads

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t^{pa}-\mathbf{q}_d)\right) = -\mathbf{n}\cdot i\omega\mathbf{u}_t \qquad \text{(frequncy domain)}$$

$$-\mathbf{n}\cdot\left(-\frac{1}{\rho_c}(\nabla p_t^{pa}-\mathbf{q}_d)\right) = -\mathbf{n}\cdot\frac{\partial\mathbf{u}_t}{\partial t} \qquad \text{(time domain)}$$

$$\left[-p_t\mathbf{I}+\mu(\nabla\mathbf{u}_t+(\nabla\mathbf{u}_t)^T)-\left(\frac{2\mu}{3}-\mu_B\right)(\nabla\cdot\mathbf{u}_t)\mathrm{I}\right]\mathbf{n} = -p_t^{pa}\mathbf{n}$$

$$-\mathbf{n}\cdot(-k\nabla T_t) = 0$$

The thermoviscous acoustic pressure variable is p_t and the pressure acoustic pressure is here denoted p_t^{pa} for clarity.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `atb1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Thermoviscous Acoustics** lists include all applicable physics interfaces. See the Coupled Interfaces in Acoustic–Structure Boundary for details.

Pair Acoustic–Thermoviscous Acoustic Boundary

The **Pair Acoustic–Thermoviscous Acoustic Boundary** coupling ( ) is used to couple a *Thermoviscous Acoustics* interface to a *Pressure Acoustics* interface (in both frequency and time domain) in an assembly geometry where identity pairs have been created. This includes both the FEM-based pressure acoustics interfaces and the BEM-based pressure acoustics interface. This allows the use of a nonconforming mesh at the acoustic-thermoviscous acoustic boundary. As the mesh requirements are different for the two, the computational mesh can take advantage of this when resolving the waves and the thermoviscous boundary layers, in this way saving degrees of freedom when solving.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Thermoviscous Acoustics** lists include all applicable physics interfaces. See the Coupled Interfaces in Acoustic–Structure Boundary for details.

PAIR SELECTION

Add and select the identity pairs where the pair multiphysics coupling feature is applied.

When coupling on pairs with this feature, it is recommended to use the **Create imprints** option in the final **Form Assembly** step in the **Geometry** sequence. Moreover, make sure that the **Allowed region** is set to **Nonpair region** in the **Applicable Pair Region** section in the Wall feature. In all cases, check the selections of all features in the model.

Acoustic–Porous Boundary

The **Acoustic–Porous Boundary** coupling ( ) is used to couple a **Pressure Acoustics, Frequency Domain** interface to a porous material (Biot's model) domain from the **Poroelastic Waves** interface. This includes both the FEM based acoustics interfaces and the BEM based acoustics interface. The boundary coupling feature includes the following interaction between the fluid and the porous domains:

- Continuity of the fluid pressure on the boundaries where the fluid interacts with the porous domain. The pore pressure in the porous domain is set equal to the total pressure in the fluid domain

$$p_{\text{pore}} = p_{\text{t}}$$

- A pressure load from the fluid pressure is experienced by the elastic waves in the porous material

$$\mathbf{n} \cdot (\sigma_{\text{d}} - \alpha_{\text{B}} p \mathbf{I}) = -\mathbf{n} p_{\text{t}}$$

  where p_{t} is the total acoustic pressure in the fluid domain and the left-hand side represents the total stress for the saturated porous domain.

- The pressure acoustic domain also experiences a normal acceleration that depends both on the acceleration of the porous matrix skeleton but also on the pore pressure. Because of the pressure boundary condition, which is a bidirectional constraint, this condition reduces to the fluid experiencing a normal acceleration

$$\mathbf{a}_{\text{n}} = (i\omega)^2 \mathbf{u}$$

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `apb1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Porous media** lists include all applicable physics interfaces. See the Coupled Interfaces in Acoustic–Structure Boundary for details.

	Headphone on an Artificial Ear. Application Library path **Acoustics_Module/Electroacoustic_Transducers/headphone_artificial_ear**

Porous–Structure Boundary

The **Porous–Structure Boundary** coupling ( ) is used to couple a porous domain (Biot's model) domain from the **Poroelastic Waves** interface to a **Solid Mechanics**, **Shell**, **Layered Shell**, **Membrane**, or **Multibody Dynamics** interface. At the solid-porous boundaries, continuity of the displacement field is applied.

In 2D components, it is possible to define the out-of-plane thickness for the structural interfaces while in acoustics this thickness effectively is unity (set to 1 m). A rescaling is automatically performed for the pressure load to account for this difference in interpretation in 2D.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `psb1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Porous media** and **Structure** lists include all applicable physics interfaces. See the Coupled Interfaces in Acoustic–Structure Boundary for details.

COUPLING

Select **Coupling** as either **Perfect bonding** (the default) or **Imperfect bonding**. The first option represent a scenario where the solid is glued or bonded to porous material. The second option, represents a case with only coupling in the normal direction; this can represent a case where the porous material is placed on to or next to the solid.

Acoustics of a Particulate-Filter-Like System. Application Library path **Acoustics_Module/Automotive/acoustics_particulate_filter**

Background Potential Flow Coupling

The **Background Potential Flow Coupling** ( ) is a unidirectional multiphysics coupling that couples the background flow calculated by a **Compressible Potential Flow** interface to the **Linearized Potential Flow** interfaces. The coupling provides the flow inputs for the **Linearized Potential Flow Model**.

For good numerical performance and accuracy, the Compressible Potential Flow model and the linearized acoustics model need to be solved on the same mesh. This multiphysics coupling does not rely on a mapping strategy like the Background Fluid Flow Coupling used to couple a fluid flow solution to the linearized Euler and linearized Navier–Stokes interfaces. The coupling is also applicable in PML domains where the Compressible Flow model can be solved.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `pfc1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Source** and **Destinations** lists include all applicable physics interfaces. See the Coupled Interfaces in Acoustic–Structure Boundary for details.

	Flow Duct. Application Library path **Acoustics_Module/ Aeroacoustics_and_Noise/flow_duct**

Background Fluid Flow Coupling

The **Background Fluid Flow Coupling** ( ) is a unidirectional multiphysics coupling between a *Fluid Flow* interface and either the *Linearized Navier–Stokes*, *Linearized Euler*, or *Convected Wave Equation* interfaces. The coupling should be solved for in a separate study using the dedicated Mapping ( ) study.

The multiphysics coupling feature and mapping study ensures that the fluid flow variables are mapped correctly from the fluid flow mesh to the acoustics mesh. The two meshes are typically different because of different resolution requirements, or in the case of the Convected Wave Equation interface because different shape function orders are used. The mapping also per default adds a small amount of smoothing of the

mapped flow solution. This ensures that no unphysical numerical noise is introduced to the acoustics model through the reactive terms in the governing equations.

- *Helmholtz Resonator with Flow: Interaction of Flow and Acoustics.* Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/helmholtz_resonator_with_flow**
- *Ultrasonic Flowmeter with Piezoelectric Transducers.* Application Library path **Acoustics_Module/Ultrasound/flow_meter_piezoelectric_transducers**

When coupling a *Fluid Flow* interface to the **Convected Wave Equation, Time Explicit** interface it is important to have consistent settings for the Geometry Shape Function and the Discretization of the physics. The **Automatic** setting for the **Geometry shape function** (in the Curved Mesh Elements section on the Components node's settings) results in a linear geometry representation that fits the *Fluid Flow* interface. This can lead to numerical errors when solving the **Convected Wave Equation, Time Explicit** physics as the default is to use fourth-order (quartic) spatial discretization of the dependent variables. Errors typically occur when curved boundaries are present in the model or when using multiphysics pair couplings. To remedy this change the **Geometry shape function** to **Quadratic Lagrange**.

If the settings in a model are inconsistent, a warning is given in the solver with the text:

```
The geometry shape function order is not consistent with
the discretization of the dependent variables.
Consider increasing the geometry shape function order to
Quadratic Lagrange in the Curved Mesh Element section in
the Component node's settings.
```

COUPLED INTERFACES

Select the **Source** fluid flow interface and the **Destination** acoustics interface.

VARIABLES TO MAP

Select the variables that need to be mapped; this depends on the physics selected. For most cases, the background mean flow velocity and pressure need to be mapped. If the flow is compressible, also map the density. If the flow is nonisothermal, also map the

temperature. For linearized Navier–Stokes models, the turbulent viscosity can also be mapped; this ensures that acoustic waves are correctly attenuated as they propagate through regions of high turbulence.

- Select **Map the pressure** (selected per default), **Map the velocity** (selected per default), **Map the density**, **Map the temperature**, or **Map the turbulent viscosity** as necessary.
- For the velocity, select **Use no slip boundary on no slip walls** if the mapped variables should have a true/forced no-slip condition applied. This may modify the flow solution slightly near walls and should be used with care. When combined with the linearized Navier–Stokes physics, this may be necessary to get consistent no-slip conditions at walls.
- The **Use symmetry on symmetry walls** option, selected per default, ensures that the mapped flow field is symmetric for symmetry conditions.
- For the pressure, velocity, density, temperature, and turbulent viscosity variables, the option **Constrain the (variable) on exterior boundaries** option sets up a constraint on all exterior boundaries, where the mapped variable is set equal to the source fluid flow variable.

SMOOTHING

Select the **Smoothing method** as **Isotropic diffusion** (the default) or **None**. The smoothing represents numerical consistent stabilization comparable to source term stabilization. It is consistent in the sense that it scales with mesh and discretization. For the isotropic diffusion option, set the (numerical) **Diffusion constant** (the default: `1e-2`). It is recommended to always use a small amount of smoothing to ensure smooth gradients of the background mean flow variables. To verify that the amount added is adequate, for example, compare the mapped variables and the fluid flow variables in a plot.

Special care should be taken if perfectly matched layers (PMLs) are present in the acoustic model. In this case, make sure to disable the PMLs (either in the model builder tree or in the study using the **Modify model configuration for the study step** option) when setting up and solving the fluid flow problem. Remember to enable the PMLs again (if disabled in the model tree) when solving the acoustics problem.

Aeroacoustic Flow Source Coupling

The **Aeroacoustic Flow Source Coupling** ( ) is a unidirectional multiphysics coupling between an LES (large eddy simulations) or DES (detached eddy simulation) *Fluid*

Flow interface and the Aeroacoustic Flow Source feature in the **Pressure Acoustics, Frequency Domain** interface.

The coupling feature should be solved for in a separate study using the dedicated Transient Mapping ( ) study. This step will map necessary data from the CFD source mesh to an acoustics mesh. Following the Transient Mapping, a separate Time to Frequency FFT study has to be used to transform the flow source data from the time domain to the frequency domain. Then the Pressure Acoustics, Frequency Domain model is solved.

For more details about theory and setting up an aeroacoustic model with flow-induced noise, see Modeling with the Aeroacoustic Flow Source: Flow-Induced Noise.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `afsc1`.

DOMAIN SELECTION

Select the domains where the Lighthill stress tensor is computed, mapped to the acoustic mesh, and used in the Aeroacoustic Flow Source feature as a source term.

BOUNDARY SELECTION

If the **Acoustic analogy** option is set to **Lighthill** (see below), select boundaries where the normal mass flow is mapped. This corresponds to sources generated by moving/vibrating boundaries in the flow simulation. If all boundaries are rigid, no boundary selection is necessary.

COUPLED INTERFACES

Select the **Source** and **Destination** physics. The source can only be an LES or DES fluid-flow physics interface and the destination the Pressure Acoustics, Frequency Domain interface.

AEROACOUSTIC FLOW SOURCE

Select the **Acoustic analogy**, used for the computational aeroacoustics (CAA) simulation, as **Lighthill** (the default) or **Aeroacoustic wave equation**. For the coupling to

work, the **Acoustic analogy** selection in the Aeroacoustic Flow Source has to match the options selected in the multiphysics coupling.

- For **Lighthill**, select **Include excess pressure effects** or **Include viscous stress**. The excess pressure option adds corrections to the Lighthill stress tensor that stem from nonlinear effects, that is, where the linear pressure-density constitutive relation does not hold. The viscous stress option adds the viscous stress components of the tensor. These are, for example, necessary for models with small dimensions where damping can be important. The data mapped/coupled to the Aeroacoustic Flow Source feature will contain the necessary 6 elements of the Lighthill stress tensor (as the tensor is symmetric). The full tensor is used in the subsequent acoustic simulation.
- For **Aeroacoustic wave equation**, only the scalar data associated with the second time derivative of the pressure is mapped/coupled to the Aeroacoustic Flow Source feature. This option includes neither viscous effects nor the effects of vibrating boundaries.

SMOOTHING

Select the **Smoothing method** as **Isotropic diffusion** (the default) or **None**. For the isotropic diffusion option, set the (numerical) **Diffusion constant** (the default: `1e-4`). It is recommended to always use a small amount of smoothing to ensure smooth gradients of the background mean flow variables.

Acoustic FEM–BEM Boundary

The **Acoustic FEM–BEM Boundary** coupling ( ) is used to couple the FEM based **Pressure Acoustics, Frequency Domain** interface to the BEM based **Pressure Acoustics, Boundary Elements** interface. The coupling works as a bidirectional coupling as most other multiphysics couplings in the Acoustics Module. The coupling prescribes continuity in the total pressure

$$p_{\mathrm{t,bem}} = p_{\mathrm{t,fem}}$$

where $p_{\mathrm{t.bem}}$ is the total pressure in the BEM based pressure acoustics interface and $p_{\mathrm{t,fem}}$ is the total pressure in the FEM based pressure acoustics interface.

Using this coupling, the strengths of the FEM and BEM methods can be used where they apply best. For example, the FEM based interface should be used in interior domains or to model porous domains with the Poroacoustics feature. When coupled to an open space where radiation occurs, it can be advantageous to use BEM here. This

is, for example, the case when modeling loudspeaker cabinets and enclosures that have a vent.

Another powerful application of the FEM–BEM coupling is to use the BEM based interface, when coupled to Pressure Acoustics, Frequency Domain, as a combined radiation condition and far-field calculating feature. The radiation condition will be perfect as the BEM problem is solved exactly and the BEM variables can be used in postprocessing instead of the usual far-field variables. The BEM variables like, for example, the sound pressure level `pabe.Lp` can be used in the existing far-field plots.

An interior impedance condition can be added between the FEM and the BEM domain using the Impedance (subfeature).

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `apb1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Boundary elements acoustics** and **Finite elements acoustics** lists include all applicable physics interfaces. See the Coupled Interfaces in Acoustic–Structure Boundary for details.

Impedance (subfeature)

The **Impedance** node can be added as a subfeature to the Acoustic FEM–BEM Boundary coupling. It is used to define an interior impedance between the FEM and the BEM acoustic domains. It can, for example, be used to model a perforated plate between the domains or as a simplified model of a thin elastic layer.

IMPEDANCE

Choose the (interior) **Impedance model** from the list as **User defined** (the default), **Perforated plate**, **Thin plate**, **Membrane**, or **Porous mass layer**. For the details of the settings, see the Interior Impedance/Pair Impedance feature in The Pressure Acoustics, Frequency Domain Interface.

Acoustic–Pipe Acoustic Connection

The **Acoustic–Pipe Acoustic Connection** multiphysics coupling ( ) is use used to couple the *Pipe Acoustics* interfaces to the *Pressure Acoustics* interface. It applies in quiescent

conditions, that is, for zero background mean flow in the pipe acoustic interface. The coupling is between a point (the end of a 1D pipe) in the pipe acoustics interface and a boundary in the pressure acoustics interface.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `apc1`.

BOUNDARY SELECTION, ACOUSTICS

Select the boundary in the pressure acoustics model that corresponds to the inlet/outlet of the 3D model coupled to pipe acoustics.

POINT SELECTION, PIPE ACOUSTICS

Select the point in the pipe acoustics model that corresponds to the inlet/outlet of the pipe model coupled to the 3D pressure acoustics domain.

- *Acoustics of a Pipe System with 3D Bend and Junction*. Application Library path **Acoustics_Module/Tutorials,_Pipe_Acoustics/acoustics_pipe_system**
- *Probe Tube Microphone*. Application Library path **Acoustics_Module/Tutorials,_Pipe_Acoustics/probe_tube_microphone**

Acoustic–Structure Boundary, Time Explicit

The **Acoustic–Structure Boundary, Time Explicit** multiphysics coupling ( ) is used to couple the **Pressure Acoustics, Time Explicit** or the **Nonlinear Pressure Acoustics, Time Explicit** interface with the **Elastic Waves, Time Explicit** interface. The condition is added on the boundary between the fluid domain and the solid domain for modeling acoustic structure interaction (ASI) or vibroacoustic problems. If possible, the Pair Acoustic–Structure Boundary, Time Explicit condition should be used as it will improve the performance of the time explicit method.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `asbte1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces. That is The Pressure Acoustics, Time Explicit Interface and The Nonlinear Pressure Acoustics, Time Explicit Interface, and The Elastic Waves, Time Explicit Interface can be selected here.

Propagation of Seismic Waves Through Earth. Application Library path **Acoustics_Module/Elastic_Waves/seismic_waves_earth**

Pair Acoustic–Structure Boundary, Time Explicit

The **Pair Acoustic–Structure Boundary, Time Explicit** multiphysics coupling ( ) is used to couple the **Pressure Acoustics, Time Explicit** or the **Nonlinear Pressure Acoustics, Time Explicit** interface with the **Elastic Waves, Time Explicit** interface, in an assembly geometry where identity pairs have been created.

This pair version of the coupling makes it possible to take full advantage of the time explicit formulation, where the global time step is controlled by the minimum of all local mesh based time steps (complying with the local CFL condition), that is using a mesh size based on the local minimum wave speed $c_{\min}$, as the mesh sizes do not need to be conforming where the fluid and the solid domains meet. The local mesh size $h_{\max}$ should always comply with

$$h_{\max} = \frac{c_{\min}}{N f_{\max}}$$

where N depends on the element approximation order (1.5-2 for the default settings) and $f_{\max}$ is the maximum frequency to resolve. There will often be a clear advantage in the computation time using the **Pair Acoustic–Structure Boundary, Time Explicit** over the **Acoustic–Structure Boundary, Time Explicit** feature due to a larger internal time step and in general fewer degrees of freedom solved for.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `aspte1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces. That is The Pressure Acoustics, Time Explicit Interface and The Nonlinear Pressure Acoustics, Time Explicit Interface, and The Elastic Waves, Time Explicit Interface can be selected here.

PAIR SELECTION

Add and select the identity pairs where the pair multiphysics coupling feature is applied.

Create separate **Identity boundary Pairs** for components of the boundary (where the coupling feature is used) if they meet at a corner or sharp edge. One identity pair for each boundary. By default, identity pairs are created automatically when forming an assembly. Use the automatically created identity boundary pair, duplicate it and then edit them removing boundaries, where necessary.

Convected Acoustic–Structure Boundary, Time Explicit

The **Convected Acoustic–Structure Boundary, Time Explicit** multiphysics coupling ( ) is used to couple the **Convected Wave Equation, Time Explicit** with the **Elastic Waves, Time Explicit** interface. The condition is added on the boundary between the fluid domain and the solid domain for modeling acoustic structure interaction (ASI) or vibroacoustic problems in the presence of a stationary background flow. The condition is applicable for physically correct configurations, specifically where the normal component of the background flow is zero, $\mathbf{n} \cdot \mathbf{u}_0 = 0$.

If possible, the Pair Convected Acoustic–Structure Boundary, Time Explicit condition should be used as it will improve the performance of the time explicit method.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `csbte1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces. The Convected Wave

Equation, Time Explicit Interface and The Elastic Waves, Time Explicit Interface can be selected here.

Pair Convected Acoustic–Structure Boundary, Time Explicit

The **Pair Convected Acoustic-Structure Boundary, Time Explicit** multiphysics coupling ( ) is used to couple the **Convected Wave Equation, Time Explicit** interface with the **Elastic Waves, Time Explicit** interface, in an assembly geometry where identity pairs have been created.

This pair version of the Convected Acoustic–Structure Boundary, Time Explicit coupling makes it possible to take full advantage of the time explicit formulation. See the Pair Acoustic–Structure Boundary, Time Explicit feature for more details.

	Ultrasonic Flowmeter with Piezoelectric Transducers. Application Library path **Acoustics_Module/Ultrasound/flow_meter_piezoelectric_transducers**

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Acoustics** and **Structure** lists include all applicable physics interfaces. The Convected Wave Equation, Time Explicit Interface and The Elastic Waves, Time Explicit Interface can be selected here.

PAIR SELECTION

Add and select the identity pairs where the pair multiphysics coupling feature is applied.

	Create separate **Identity boundary Pairs** for components of the boundary (where the coupling feature is used) if they meet at a corner or sharp edge. One identity pair for each boundary. By default, identity pairs are created automatically when forming an assembly. Use the automatically created identity boundary pair, duplicate it and then edit them removing boundaries, where necessary.

Piezoelectricity, Time Explicit

The **Piezoelectricity, Time Explicit** ( ) feature is a multiphysics coupling feature between the **Elastic Waves, Time Explicit** interface and the **Electrostatics** interface used to model a piezoelectric material. It couples the Piezoelectric Material node in elastic waves to the Charge Conservation, Piezoelectric node in Electrostatics. The multiphysics coupling is automatically added when The Piezoelectric Waves, Time Explicit Interface is added as a new multiphysics interface.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `pzete1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. Only the **Elastic Waves, Time Explicit** and **Electrostatics** physics interfaces can be selected.

- *Angle Beam Nondestructive Testing*. Application Library path **Acoustics_Module/Ultrasound/angle_beam_ndt**
- *Ultrasonic Flowmeter with Piezoelectric Transducers*. Application Library path **Acoustics_Module/Ultrasound/flow_meter_piezoelectric_transducers**

Piezoelectricity

The **Piezoelectricity** ( ) feature is a multiphysics coupling feature between the **Solid Mechanics** interface and the **Electrostatics** interface used to model a piezoelectric material. This feature is used for the finite element method (FEM) based interfaces and is applicable for transient, frequency domain, and stationary studies. The **Piezoelectricity** coupling is described for The Piezoelectricity Interface.

Magnetomechanics

To model interactions between deformable solids and magnetic fields, use the **Magnetomechanics** multiphysics coupling. For details, see the Magnetomechanics and The Magnetomechanics Interface documentation of the *Structural Mechanics User's Guide*.

Acoustic Streaming Domain Coupling

The **Acoustic Streaming Domain Coupling** ( ) feature is a multiphysics coupling from an acoustic interface to a fluid flow (CFD) model, used to add the domain source contributions necessary to model an acoustic streaming flow. The multiphysics coupling should be used in combination with the Acoustic Streaming Boundary Coupling to ensure that all sources are modeled.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `asdc1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The multiphysics interface couples a **Source** acoustic interface to a **Destination** single-phase fluid flow interface. The acoustic interface can be either **Pressure Acoustics, Frequency Domain** or **Thermoviscous Acoustics, Frequency Domain**. In the **Pressure Acoustics, Frequency Domain** interface it cannot couple to a **Poroacoustics**, **Narrow Region Acoustics**, **Anisotropic Acoustics**, or **Anisotropic Poroacoustics** domain. The fluid flow interface should be a **Single-Phase Flow** interface, but it cannot couple to a **Porous Medium** domain. The interface can couple to a turbulent flow interface, but it is advised to take extra caution since the derivation of the source terms does not take turbulent flow into consideration. For a mathematically consistent formulation, the coupling is typically to the **Creeping Flow** or the **Laminar Flow** interface.

ACOUSTIC STREAMING DOMAIN COUPLING

Choose to select the **Subtract Lagrangian energy density** options (selected per default) to subtract the Lagrangian energy density from the momentum flux tensor. The Lagrangian energy density does not induce streaming flow $\mathbf{u}_2$ since it results in a pure gradient force; however, it does induce a gradient in the streaming (time averaged) pressure field p_2. Subtracting the Lagrangian energy density can make the simulation converge more easily on a coarser numerical mesh. The Lagrangian energy density has a large numerical value compared to the other terms even though it does not induce streaming. Note that because the Lagrangian energy density is subtracted, the resulting pressure calculated in the flow module is not the correct physical pressure, but the velocity field is correct. The Lagrangian energy variable `asdc1.Lac` can be added to the pressure p_2 in postprocessing to get the correct pressure.

Choose to select the **Include first order viscosity terms** option to include linear perturbation contributions to the viscosity in the governing equations. When selected choose the **Derivatives of dynamic viscosity** to be either **From material** (the default) or **User defined**.

- When **From material** is selected, it is important that the viscosity in the material (under the **Materials** node) is both pressure and temperature dependent, if not the specific contribution will evaluate to zero.
- For **User Defined,** enter the derivative of the dynamic viscosity with respect to temperature and pressure (evaluated at T_0 and p_0). This can, for example, be a constant value, an analytical expression, or data from an interpolation function.

The forces applied on the fluid flow depends on the derivatives of the acoustic fields. Therefore, it is recommended to use quadratic element order for the acoustic pressure, and when coupling to **Thermoviscous Acoustics, Frequency Domain**, use cubic element order for the acoustic velocity and temperature.

Get further details and modeling advice, and read about the theory in the Acoustic Streaming chapter.

Acoustic Streaming Boundary Coupling

The **Acoustic Streaming Boundary Coupling** ( ) is a multiphysics coupling from an acoustic interface to a fluid flow (CFD) model, used to add the boundary source contributions necessary to model an acoustic streaming flow. The multiphysics coupling should be used in combination with the Acoustic Streaming Domain Coupling to ensure that all sources are modeled.

The coupling is between either the **Thermoviscous Acoustics, Frequency Domain** or the **Pressure Acoustics, Frequency Domain** and a fluid flow interface. When **Pressure Acoustics, Frequency Domain** is used as the acoustic source interface, the **Fluid model** should be set to the **Thermally conducting and viscous** option (see Thermally Conducting and/or Viscous Fluid Model) and the Thermoviscous Boundary Layer Impedance condition is best used to include boundary layer losses. In situations with highly curved or sharp edges, when the boundary layer impedance formulation is not valid, the acoustic interface should be changed to thermoviscous acoustics instead of pressure acoustics.

SETTINGS

See Settings for further details about **Label** and **Name**.

The default **Name** (for the first multiphysics coupling feature in the model) is `asbc1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The multiphysics interface couples a **Source** acoustic interface to a **Destination** single-phase fluid flow interface. The acoustic interface can be either **Pressure Acoustics, Frequency Domain** or **Thermoviscous Acoustics, Frequency Domain**. In the **Pressure Acoustics, Frequency Domain** interface it cannot couple to a **Poroacoustics**, **Narrow Region Acoustics**, **Anisotropic Acoustics**, or **Anisotropic Poroacoustics** domain. The fluid flow interface can be a **Single-Phase Flow** interface, but it cannot couple to a **Porous Medium** domain. The interface can couple to a turbulent flow interface, but it is advised to take caution since the derivation of the source terms does not take turbulent flow into consideration. For a mathematically consistent formulation, the coupling is typically to the **Creeping Flow** or the **Laminar Flow** interface.

ACOUSTIC STREAMING BOUNDARY COUPLING

Choose to select the **Include first order viscosity terms** option to include linear perturbation contributions to the viscosity in the governing equations. This only applies when coupling from Pressure Acoustics, Frequency Domain. When selected choose the **Derivatives of dynamic viscosity** to be either **From material** (the default) or **User defined**.

- When **From material** is selected it is important that the viscosity in the material (under the **Materials** node) is both has a pressure and temperature dependent, if not the specific contribution will evaluate to zero.
- For **User Defined** enter the derivative of the dynamic viscosity with respect to temperature and pressure (evaluated at T_0 and p_0). This can, for example, be a constant value, an analytical expression, or data from an interpolation function.

CONSTRAINT SETTINGS

These settings are shown when the **Advanced Physics Options** is selected. It controls how the constraint that adds the boundary slip velocity contributions is handled in the Fluid Flow interface. For the **Constraints** select either **Nitsche constraints** (the default) or **Pointwise constraints**. The **Pointwise constraints** option is really only applicable on flat boundaries; as soon as the boundary is curved, so-called numerical locking effects can occur for this option.

Predefined Multiphysics Interfaces

The Acoustics Module includes several predefined multiphysics interfaces. When these interfaces are selected from the **Model Wizard** or the **Add Physics** window both contributing physics and the multiphysics couplings are added to the **Model Builder**. It is always possible to build a model sequentially adding one physics at the time and then coupling them. Note that far from all possible couplings are represented in the predefined physics interfaces. All the couplings are listed in the Coupling Features section.

When physics interfaces are added using the predefined multiphysics interfaces, for example **Thermoviscous Acoustic–Structure Interaction**, the **Selection** of the coupling feature, in this case the **Thermoviscous Acoustic–Structure Boundary**, is automatically set to **All boundaries**. In this way, the multiphysics coupling is automatically active on all boundaries with **Thermoviscous Acoustics** on one side and **Solid Mechanics** on the other.

If physics interfaces are added one at a time, followed by the coupling features, the selections are not automatically set. For example, if the single interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics Couplings** node. You can choose the available coupling features, but you need to manually select on which boundaries they are applied, or select **All boundaries** to recover the predefined behavior.

The Acoustics Module includes the following predefined multiphysics interfaces:

- The Acoustic–Solid Interaction, Frequency Domain Interface
- The Acoustic–Solid Interaction, Transient Interface
- The Acoustic–Piezoelectric Interaction, Frequency Domain Interface
- The Acoustic–Piezoelectric Interaction, Transient Interface
- The Acoustic–Solid–Poroelastic Waves Interaction Interface
- The Acoustic–Poroelastic Waves Interaction Interface
- The Acoustic–Solid Interaction, Time Explicit Interface
- The Acoustic–Shell Interaction, Frequency Domain Interface
- The Acoustic–Shell Interaction, Transient Interface
- The Acoustic–Thermoviscous Acoustic Interaction, Frequency Domain Interface
- The Thermoviscous Acoustic–Solid Interaction, Frequency Domain Interface
- The Thermoviscous Acoustic–Shell Interaction, Frequency Domain Interface

- The Thermoviscous Acoustic–Thermoelasticity Interaction Multiphysics Interfaces
- The Piezoelectric Waves, Time Explicit Interface
- Acoustic Streaming from Thermoviscous Acoustics
- Acoustic Streaming from Pressure Acoustics

Another option is to create the coupled predefined multiphysics interface by adding it from the **Add Multiphysics** window after adding the separate physics interfaces.

Modeling with Multiphysics Couplings

Using the predefined multiphysics couplings has many advantages. It improves the flexibility and enables sequential modeling, increasing the complexity of a model step by step. Care is also taken to make the correct couplings both on the physics level but also from a numerics point of view. It is of course possible to couple physics manually and this is maybe also necessary in some cases when more advanced and exotic couplings are necessary.

The predefined multiphysics coupling approach used improves the flexibility and design options for your modeling. For details, see The Multiphysics Branch and Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

In this section, a few tips and tricks for modeling acoustics and vibration problems using the multiphysics couplings are presented.

- Use Selections
- The Override Behavior
- The Solvers
- Perfectly Matched Layers (PMLs)

Use Selections

When modeling, it is always recommended to use selections to predefine and give names to several domains or boundaries. Define the selections under the **Definitions** node. Setting up these selections will for multiphysics modeling avoid adding two incompatible physics to the same domain. If acoustics and structures are coupled this happens at boundaries, where two distinct physics should meet. For, for example, piezoelectric devices, the coupling between the electric and mechanical effect happen on the domain level. Operations can also be performed on the selections, if the geometry is complex, to find intersections, complement sets and so forth.

See Named Selections in the *COMSOL Multiphysics Reference Manual*.

The Override Behavior

When coupling acoustics, both Pressure Acoustics, Thermoviscous Acoustics, and Aeroacoustics, to a structure, the behavior of the multiphysics coupling is such that it will override all boundary condition given on the acoustic side. Loads and constraints defined on the common boundaries in the structural physics interface will contribute with the coupling. In this way, external forces that drive the system can be added here. It is also possible to constrain the structure on these boundaries while it will still experience the acoustic loads.

The Solvers

When solving a Multiphysics problem, it is necessary to ensure that the solvers behave correctly. It is especially important to check if a model is solved **Fully Coupled** or in a **Segregated** way. COMSOL has built in logic to handle most multiphysics problems when the predefined multiphysics couplings are used, ensuring that the model solves correctly. If many physics or couplings are added at the same time, this logic can fail and manual settings should be performed. For large models COMSOL generates Iterative Solver Suggestions (see below).

See Introduction to Solvers and Studies and Solution Operation Nodes and Solvers in the *COMSOL Multiphysics Reference Manual*.

It is also possible to enable and disable both physics and couplings from within a Study Step. This can be done in the **Physics and Variables Selection** section in **Settings**. This functionality can be used to solve a model both coupled and uncoupled by adding two **Studies**: one where the Multiphysics coupling is active and one where it has been deactivated.

ITERATIVE SOLVER SUGGESTIONS

Multiphysics models can in 3D easily grow rapidly in size and become difficult to solve with a direct solver, simply because you run out of RAM. In this case, it can be necessary to switch to an iterative solver approach. For most multiphysics couplings, iterative solver suggestions are automatically generated. Set up a new solver configuration (first time the model is solved) or right-click the study node and select **Show Default Solver**, then expand the **Solver Configuration** tree under **Stationary Solver** or **Time-Dependent Solver**. The predefined iterative solver suggestions that are automatically generated can be seen here. In the predefined cases, a direct solver is used

as default and iterative solvers are suggested and disabled. To turn on one of these approaches, right-click the solver and select **Enable** (or press F4).

All solver suggestions have a tag that makes it possible to see which physics feature that has generated them, for example, *(asb1)* means that the suggestion comes from the *Acoustic–Structure Boundary 1* feature.

Some tutorial models from the library that use a solver suggestions:

- **Electroacoustic_Transducers/vented_loudspeaker_enclosure**
- **Tutorials,_Thermoviscous_Acoustics/transfer_impedance_perforate**
- **Vibrations_and_FSI/vibrating_micromirror**

TIME STEPPING IN TRANSIENT MODELS

When a multiphysics model is solved in the time domain, it is important to update the **Transient Solver Settings** section of the acoustics interface (and structural interfaces if present) used in the model. The time stepping settings generated by these settings will also be applied to the coupled problem when a **Multiphysics** couplings is used.

Perfectly Matched Layers (PMLs)

Care should be taken when setting up perfectly matched layers (PMLs) in multiphysics models where different waves propagate. A discussion about the Acoustic-Structure interaction situation is given in Configuration of Perfectly Matched Layers (PMLs) for Acoustic–Structure Interaction Models in Modeling with the Acoustic–Structure Interaction Branch. The approach described is valid in general for models where several sound speeds are present.

12

Structural Mechanics with the Acoustics Module

Elastic waves and vibrations are an important part of acoustics. The Acoustics Module is therefore equipped with an extended version of the base package Solid Mechanics interface shipped with COMSOL Multiphysics. This interface contains less features compared to the full Solid Mechanics interface available with the Structural Mechanics Module.

The **Solid Mechanics** physics interfaces are found under the **Structural Mechanics** branch ( ) or as a shortcut **Solid Mechanics (Elastic Waves)** under the **Acoustics Module** branch ( ) in the **Elastic Waves** folder.

For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

In this chapter:

- Vibroacoustic Applications
- The Solid Mechanics Interface
- The Piezoelectricity Interface
- Acoustic–Structure Multiphysics Interaction

Vibroacoustic Applications

The Solid Mechanics and Piezoelectricity interfaces are available with the Acoustics Module such that vibroacoustic problems involving solids, piezoelectric materials, and acoustic domains can be modeled. These multiphysics applications include piezo transducers, loudspeaker drivers, loudspeaker cabinets, automotive applications, and sound transmission and isolation problems. Pure elastic wave problems can also be modeled as the Solid Mechanics equations are formulated to include the full dynamic behavior.

Additional multiphysics capabilities of the Acoustics Module are enabled when it is combined with the Structural Mechanics Module. The Shell interface, which is part of the Structural Mechanics Module, enables predefined acoustic–shell couplings. The Acoustic–Shell and Thermoacoustic–Shell interaction multiphysics interfaces are enabled. The option to couple to shells, membranes, and multibody dynamics using the Acoustic–Structure Boundary is also enabled, see the Multiphysics Couplings chapter for further details.

The Solid Mechanics Interface

Included in the Acoustics Module is an extended version of the Solid Mechanics interface shipped with COMSOL Multiphysics. The interface contain less features when compared to the full Solid Mechanics interface available with the Structural Mechanics Module.

The full documentation of all features in the full Solid Mechanics interface and the Piezoelectricity interface is found in the *Structural Mechanics Module User's Guide*, specifically in these sections:

- Solid Mechanics
- Structural Mechanics Modeling
- Structural Mechanics Theory
- Multiphysics Interfaces and Couplings and specifically The Piezoelectricity, Solid Interface

The Piezoelectricity Interface

The Piezoelectricity interface is also included in the Acoustics Module. This is a Multiphysics interface that couples the Electrostatics interface with the Solid

Mechanics interface. See Modeling Piezoelectric Problems section in the *Structural Mechanics Module User's Guide* or further details.

Acoustic–Structure Multiphysics Interaction

When an acoustics interface and a structural mechanics interface are added to the same model, a **Multiphysics** node will appear if there is a predefined coupling between the two interfaces. Under the **Multiphysics** node you can add the predefined multiphysics couplings that couple the fluid domain and to the solid domain. The couplings are described in detail in the Multiphysics Couplings chapter.

You can also find a set of predefined multiphysics interface that will automatically set up the involved single physics and the multiphysics coupling. These are described in the Acoustic–Structure Interaction Interfaces chapter. Here you will also find modeling tips for acoustic–structure interaction problems in the Modeling with the Acoustic–Structure Interaction Branch section.

13

Study Types

The Acoustics Module contains a number of study types suited for acoustics simulations.

This chapter provides information about those study types — including stationary, frequency-domain, and time-dependent study types — and how you can use them for different types of acoustics simulations.

In this chapter:

- Acoustics Module Study Types

Acoustics Module Study Types

The Acoustics Module is designed for both frequency-domain simulations and transient analysis. The frequency domain simulations include the related eigenvalue and mode analysis problems. The Pressure Acoustics interfaces support most study types with certain exceptions; the Pressure Acoustics, Boundary Element interface only supports frequency domain and the Pressure Acoustics, Time Explicit only the time domain analysis. The Thermoviscous Acoustics and the linearized Aeroacoustic (or convected acoustics) interfaces all support both transient and frequency-domain analysis. The Compressible Potential Flow interface is tailored to model a stationary background flow to be used in a subsequent time-harmonic aeroacoustics simulation. In the Solid Mechanics interface, the static analysis type is also included and can be used to model the stationary state of prestressed systems subject to time-harmonic vibrations.

The analysis types require different solvers and equations. The following study types (in alphabetic order), briefly discussed in this section, can help you find good candidates for the application:

- Adaptive Frequency Sweep
- Additional Analysis Capabilities
- Boundary Mode Analysis
- Eigenfrequency Study
- Frequency Domain Study
- Frequency Domain, Modal and Time-Dependent, Modal Studies
- Frequency-Stationary
- Frequency-Transient
- Mapping
- Mode Analysis Study
- Modal Reduced-Order Model
- Ray Tracing
- Stationary Study
- Time Dependent Study
- Transient Mapping

Studies and Solvers and Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis in the *COMSOL Multiphysics Reference Manual*.

Prestressed Acoustic–Structure Interaction type of problems are also sported using a combination of a stationary solver and a Frequency-Domain Perturbation solver.

Stationary Study

A stationary analysis solves for stationary displacements or a steady-state condition. All loads and constraints are constant. For a stationary analysis, use a Stationary study type ( ). For all pure acoustic and vibration problems, this type of analysis yields the zero solution as, by definition, these represent and describe propagating varying fields — either time dependent or time harmonic in the frequency domain.

Stationary in the *COMSOL Multiphysics Reference Manual*

Frequency Domain Study

Wave propagation is modeled by equations from linearized fluid dynamics (pressure waves) and structural dynamics (elastic waves). The full equations are time dependent, but noting that a harmonic excitation of the field p has a time dependence of the form

$$p(t) = pe^{i\omega t}$$

gives rise to an equally harmonic response with the same frequency; the time can be eliminated completely from the equations. Instead the angular frequency $\omega = 2\pi f$, enters as a parameter where f is the frequency. In COMSOL, the $+i\omega$ convection is used for the time harmonic formulations.

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.

For the interfaces in the Acoustics Module that are based on the finite element method (FEM) or the boundary element method (BEM), a frequency domain simulation is well suited. Therefore, select the Frequency Domain study type ( ) over a Time Dependent study whenever possible. Certain important software features, like damping due to porous materials or advanced impedance boundary conditions, are only present when using the frequency domain physics interfaces.

The result of a frequency domain analysis is a complex time-dependent field p, which can be interpreted as an amplitude $p_{\text{amp}} = \text{abs}(p)$ and a phase angle $p_{\text{phase}} = \arg(p)$. The actual pressure at any point in time is the real part of the solution

$$p(t) = p_{\text{amp}} \cos(2\pi f \cdot t + p_{\text{phase}})$$

Visualize the amplitudes and phases as well as the solution at a specific angle (time). When using the Solution datasets, the *solution at angle (phase)* parameter makes this task easy. When plotting the solution, COMSOL Multiphysics multiplies it by $e^{i\varphi}$, where φ is the angle in radians that corresponds to the angle (specified in degrees) in the **Solution at angle** field. The plot shows the real part of the evaluated expression:

$$p = p_{\text{amp}} \cos(\varphi + p_{\text{phase}})$$

The angle φ is available as the variable `phase` (in radians) and is allowed in plot expressions. Both the frequency `freq` and angular frequency `omega` are available variables.

In a frequency domain study, almost everything is treated as harmonic — prescribed pressures, displacements, velocities, and accelerations — not only the forces and dependent fields. Notable exceptions are certain quantities, such as the sound pressure level or intensity, which by definition are time averages.

Frequency Domain and Solution (datasets) in the *COMSOL Multiphysics Reference Manual*

Eigenfrequency Study

If all sources are removed from a frequency-domain equation, its solution becomes zero for all but a discrete set of angular frequencies ω, where the solution has a well-defined shape but undefined magnitude. These solutions are known as eigenmodes and their corresponding frequencies as eigenfrequencies.

The eigenmodes and eigenfrequencies have many interesting mathematical properties, but also direct physical significance because they identify the *resonance frequency* (or frequencies) of the structure. When approaching a resonance frequency in a harmonically-driven problem, a weaker and weaker source is needed to maintain a

given response level. At the actual eigenfrequency, the time-harmonic problem loses the uniqueness of the solution for a nonzero excitation.

Select the **Eigenfrequency** study type ( ) when you are interested in the resonance frequencies of the acoustic domain or the structure, whether you want to exploit them, as in a musical instrument, or avoid them, as in a reactive muffler or inside a hi-fi speaker system. To an engineer, the distribution of eigenfrequencies and the shape of eigenmodes can also give a good first impression about the behavior of a system.

An eigenfrequency analysis solves for the eigenfrequencies and the shape of the eigenmodes. When performing an eigenfrequency analysis, specify whether to look at the mathematically more fundamental eigenvalue λ (available as the variable `lambda`) or the eigenfrequency f, which is more commonly used in an acoustics context:

$$f = \frac{-\lambda}{2\pi i}$$

NONLINEAR EIGENFREQUENCY ANALYSIS

In certain circumstances the material properties or boundary conditions can be frequency dependent. This is, for example, the case if a model set up with The Pressure Acoustics, Frequency Domain Interface contains a Narrow Region Acoustics or a Poroacoustics equivalent fluid model. The same is true if the Impedance boundary condition is added: most of the options listed in Theory for the Boundary Impedance Models define the acoustic impedance as a function of frequency. If the frequency dependence in these models is nonlinear, this will lead to a so-called nonlinear eigenvalue problem, which is more complex and therefore often must be treated carefully.

Nonlinear Eigenfrequency Solver

A dedicated eigenfrequency solver called **ARPACK nonlinear** can be selected on the eigenfrequency study step. This solver is capable of solving eigenvalue problems that are nonlinear in the eigenvalue (the eigenfrequency).

See Eigenfrequency in the *COMSOL Multiphysics Reference Manual*, and in particular Study Settings for ARPACK and ARPACK Nonlinear.

Manual Linearization for Nonlinear Eigenfrequency Problems

While solving a nonlinear eigenvalue problem (using the ARPACK or FEAST solvers), the linearization around some linearization point takes place behind the scenes. Expand the **Study** node, right-click the **Solver Configurations** node, and select **Show**

Default Solver. In the generated solver sequence, navigate to the **Eigenvalue Solver** node and find the **Transform point** field. The value specified there will be used as the linearization point in the eigenvalue study, and therefore it is very important to set up for nonlinear eigenvalue problems.

The default **Transform point** value used in the acoustics interfaces is 100 Hz. If the eigenfrequencies of interest lie in kHz or MHz range, the default value of the linearization point may not ensure an accurate solution of a nonlinear eigenvalue problem — depending on its nonlinear properties. In this case, it is good practice to move the linearization point closer to the range of desired eigenfrequencies. For example, it can be the mean of the range. The other option is to define the **Transform point** as a parameter, for example `TP`, add a **Parametric Sweep** over this parameter to the study, and search for one eigenfrequency around `TP`. This manual approach will be more accurate if the model exhibits strong nonlinear behavior within the range of desired eigenfrequencies, where a constant linearization point in not enough.

FILTERING AND SORTING

It is possible filter and sort the solutions of an Eigenfrequency study. The filtering can be done based on any global expression (the value of the eigenfrequency or some integral expression). The sorting is useful for the ordering of the solutions in the results dataset.

See Filtering and Sorting for Eigenfrequency in the *COMSOL Multiphysics Reference Manual*.

Mode Analysis Study

The Mode Analysis study ( ) is available with The Pressure Acoustics, Frequency Domain Interface, The Linearized Potential Flow, Frequency Domain Interface, and The Thermoviscous Acoustics, Frequency Domain Interface in plane 2D and axially symmetric 1D acoustics interfaces.

The Pressure Acoustics, Boundary Mode Interface, The Thermoviscous Acoustics, Boundary Mode Interface, and The Linearized Potential Flow, Boundary Mode Interface are special interfaces for more advanced Mode Analysis studies on boundaries in 3D and 2D axisymmetry. Acoustic waves can propagate over large distances in ducts and pipes, with a generic name referred to as *waveguides*. After some distance of propagation in a waveguide of uniform cross section, such guided waves can be described as a sum of just a few discrete *propagating modes*, each with its own shape

and phase speed. The equation governing these modes can be obtained as a spatial Fourier transform of the time-harmonic equation in the waveguide axial direction (here assuming propagation in the z direction) or by inserting the assumption that the mode is harmonic in space,

$$p(\mathbf{x}) = p(x,y)e^{-ik_z z}$$

and eliminating all out-of-plane z dependence. Here $p(x,y)$ is the in-plane mode shape.

The axial wave number k_z is a parameter in the 2D acoustics interfaces.

Similar to the full time-harmonic equation, the transformed equation can be solved at a given frequency with a nonzero excitation for most axial wave numbers k_z. But at certain discrete values the equation breaks down. These values are the propagation constants or wave numbers of the propagating or evanescent waveguide modes. The eigenvalue solver can solve for these propagation constants together with the corresponding mode shapes.

The propagating wave number is a function of the frequency. The relation between the two is commonly referred to as a *dispersion curve*.

The most common use for the Mode Analysis is to define sources for a subsequent time-harmonic simulation. If there is a component with one or more waveguide connections, its behavior can be described by simulating its response to the discrete set of propagating modes on the waveguide port cross sections. The Port boundary condition in Pressure Acoustics uses this concept to define sources and absorb outgoing modes. In thermoviscous acoustics, a Mode Analysis study can also be used for the Port, it can moreover provides information about the absorption coefficient for the propagating modes. The complex wave number (solved for) and characteristic impedance (defined in results) define the homogenized propagation variables that can be used in Narrow Region Acoustics.

For an example where modes are computed and used in a Port condition, see the *Flow Duct — With Boundary Mode Analysis* tutorial.

Application Library path **Acoustics_Module/Aeroacoustics_and_Noise/flow_duct_boundary_mode**

	Mode Analysis in the *COMSOL Multiphysics Reference Manual*

FILTERING AND SORTING

It is possible filter and sort the solutions of a Mode Analysis study. The filtering can be done based on any global expression (the value of the eigenfrequency or some integral expression). The sorting is useful for the ordering of the solutions in the results dataset.

	The settings for Filtering and Sorting in the Mode Analysis study follow the settings for Eigenfrequency, as described in the *COMSOL Multiphysics Reference Manual*.

Boundary Mode Analysis

The Boundary Mode Analysis study ( ) is used when solving for the propagating modes used in a Port boundary condition in The Pressure Acoustics, Frequency Domain Interface. The study has special settings that can match the shape and wave number of the mode to a given port condition.

Time Dependent Study

The complete equations behind the theory of acoustic wave propagation are time dependent, as discussed in the Frequency Domain Study section. Solving time-domain equations is more complicated from a numerical point of view. They are necessary when studying systems where the time of flight is of importance, like in a flowmeter application. Short-term transient processes like step and impulse responses can benefit from modeling in the time domain, if not for efficiency so for convenience.

A dedicated **Transient Solver Settings** settings section exist at the physics level for all the transient acoustics interfaces where the **Maximum frequency to resolve** should be entered.

	Some central modeling techniques, such as the use of PMLs, are not available for all physics with the **Time Dependent** study type. PMLs only exist in the time domain for the Pressure Acoustics, Time Domain interface. Be careful when defining your sources to avoid, as far as possible, exciting waves at frequencies that the mesh cannot resolve.

Time Dependent in the *COMSOL Multiphysics Reference Manual*.

Further details on Time Stepping: Resolving the Waves in Time in the Modeling with the Pressure Acoustics Branch (FEM-Based Interfaces) section.

Frequency Domain, Modal and Time-Dependent, Modal Studies

The **Frequency Domain, Modal** study type ( ) is used to do modal analysis in the frequency domain and the **Time-Dependent, Modal** study ( ) is used to do time-dependent modal analysis. Both study types can also be used for multiphysics models like a vibroacoustic problem. These problems can result in nonsymmetric matrices for which both left and right eigenvectors are automatically stored.

When defining sources in models solved with the modal solvers, the source expressions (magnitude) need to be wrapped with the `linper()` operator. For example, a pressure source of amplitude `1[Pa]` should be defined as `linper(1[Pa])`.

To obtain a sufficient basis, it is in most cases necessary to include eigenmodes representing eigenfrequencies up to twice the maximum frequency to be resolved. Note also that in many acoustic problems you need to include the 0 Hz mode (DC mode) to get the correct behavior at low frequencies.

Frequency Domain, Modal and Time Dependent, Modal in the *COMSOL Multiphysics Reference Manual*

Helmholtz Resonator Analyzed with Different Frequency Domain Solvers: Application Library path **Acoustics_Module/Tutorials,_Pressure_Acoustics/helmholtz_resonator_solvers**

Adaptive Frequency Sweep

The **Adaptive Frequency Sweep** study type ( ) is available as a predefined study for the Pressure Acoustics, Frequency Domain interface. The **Adaptive Frequency Sweep** study gives easy access to the asymptotic waveform evaluation (AWE) functionality available with a regular **Frequency Domain** study where it is located under the **Study Extensions** section.

In the **Adaptive Frequency Sweep** study the **AWE expression type** is set to **Physics controlled** as default. The **Physics controlled** expression is defined using the AWE Expression feature in the physics. For **User controlled** enter the **Asymptotic waveform evaluation (AWE)** expressions in the table. The **Relative tolerance** can also be entered, the default value is 0.01.

	Adaptive Frequency Sweep and AWE Solver in the *COMSOL Multiphysics Reference Manual*.

	Helmholtz Resonator Analyzed with Different Frequency Domain Solvers: Application Library path **Acoustics_Module/ Tutorials,_Pressure_Acoustics/helmholtz_resonator_solvers**

Ray Tracing

The Ray Tracing study type ( ) is used when solving acoustic problems using The Ray Acoustics Interface. It is a special case of the Time Dependent study and is used to compute the trajectories of rays. Special built-in stop conditions can be used to stop the solver when no active rays remain or the intensity of the rays is below a certain criterion.

Modal Reduced-Order Model

The **Model Reduction** and **Modal Reduced-Order Model** study steps have the property that they can assemble the modal matrices and make them available for output. In the **Model Reduction** node, the **Store reduced matrices** checkbox must be selected.

After solving the model, right-click the **Derived Values** node (under **Results**) and select **System Matrix**. In the output section, select the **Matrix** to display and the **Format**. Using

the Matrix settings it is possible to access the stiffness, damping, and mass matrices of the system, for example.

	In the *COMSOL Multiphysics Reference Manual:* • Modal Reduced-Order Model and Introduction to Solvers and Studies • System Matrix and Results Analysis and Plots

Additional Analysis Capabilities

In a multiphysics interface you might want to use different analysis types for the different dependent variables. This can be done by adding an Empty Study ( ), and then adding different study steps to this study. Also perform parametric analyses by using the **Parametric Sweep** study node ( ). Typical parameters to vary include geometric properties, the frequency band in ray tracing, and the out-of-plane or axial wave number.

	Parametric Sweep in the *COMSOL Multiphysics Reference Manual*

Mapping

The **Mapping** ( ) study is a dedicated study to use when the Background Fluid Flow Coupling couples a *Fluid Flow* interface and an *Aeroacoustics* interface. The study should only solve for the multiphysics coupling. The study uses a Stationary solver. Together with the multiphysics coupling, the study handles the mapping of dependent variables between the fluid flow mesh and the acoustics mesh for consistent simulation results.

PHYSICS AND VARIABLES SELECTION

Only select the *Background Fluid Flow Coupling* as being **Solved for**. No other physics will be solved by the mapping study.

SOLUTION TO MAP

From the **Study** list select the study containing the coupled fluid flow interface to map. Depending on the type of study, used for the fluid flow simulation, you will get a different type of list to choose from: time steps or parameter values. From the list,

select the appropriate solution or solutions to map. The option **All** will automatically add a parameter sweep and map all parameter values from the selected solution. The **Mapping** study step will automatically ensure that those parameter values are used.

Note that the mapping does not benefit from running continuation for any parameter. When the solution is mapped for all parameter values, for a multiple parameter sweep where continuation was used in the fluid flow study, the resulting dataset structure of the two studies will therefore be slightly different.

DESTINATION MESH SELECTION

Select the destination/target mesh of the mapping, that is, the acoustics mesh used for the aeroacoustics simulation.

Transient Mapping

The **Transient Mapping** ( ) study is a dedicated study to use when the Aeroacoustic Flow Source Coupling couples a LES *Fluid Flow* interface with the Aeroacoustic Flow Source in pressure acoustics. The study should only solve for the multiphysics coupling. The study uses a tailored **Transient** solver. Together with the multiphysics coupling, the study handles the mapping of flow source variables (the Lighthill stress tensor) between the fluid flow mesh and the acoustics mesh for consistent simulation results.

SOLUTION TO MAP

From the **Study** list select the study containing the coupled fluid flow interface to map.

DESTINATION MESH SELECTION

Select the destination/target mesh of the mapping, that is, the acoustics mesh used for the aeroacoustics simulation.

Frequency-Stationary

The **Frequency-Stationary** study is a dedicated study step to use when the **Acoustic Streaming Boundary Coupling** or **Acoustic Streaming Domain Coupling** is used to couple an acoustic frequency domain interface to a stationary fluid flow interface. The study uses a tailored solver that solves for the acoustic fields and stationary fields in the same study step.

PHYSICS AND VARIABLES SELECTION

In the **Physics and Variables Selection** under the study step the equation form of the acoustic interface is designated a **Automatic (Frequency Domain)**, the stationary interfaces is set to **Automatic (Stationary)**, and the two acoustic streaming multiphysics couplings are set to **Automatic (Frequency-stationary)**.

The **Frequency-Stationary** study step requires an Acoustics Module license. The study step is only possible to select and use together with the multiphysics couplings **Acoustic Streaming Boundary Coupling** or **Acoustic Streaming Domain Coupling**.

Frequency-Stationary in the *COMSOL Multiphysics Reference Manual*

Frequency-Transient

The **Frequency-Transient** study is a dedicated study step to use when the **Acoustic Streaming Boundary Coupling** or **Acoustic Streaming Domain Coupling** is used to couple an acoustic frequency domain interface to a time-dependent fluid flow interface. The study uses a tailored solver that solves for the acoustic fields and time dependent fields in the same study step. The solver is only valid if the timescale of the acoustics (determined by the frequency) is a lot shorter than the time-scale of the time-dependent fluid flow. The difference in time-scales is used to assume that the time-dependent flow variables are stationary (quasi stationary) in the acoustic equations.

In the **Frequency-Transient** study the equations related to the acoustic fields are evaluated at every time-step solved for. This makes it possible to vary the acoustic field in time (for example effects due to heating and changes in material properties), as long as the variations occur on a time-scale much longer than the timescale set by the acoustic frequency.

PHYSICS AND VARIABLES SELECTION

In the **Physics and Variables Selection** under the study step the equation form of the acoustic interface is designated a **Automatic (Frequency Domain)**, the stationary

interfaces is set to **Automatic (Time dependent)**, and the two acoustic streaming multiphysics couplings are set to **Automatic (Frequency-transient)**.

The **Frequency-Transient** study step requires an Acoustics Module license. The study step is only possible to select and use together with the multiphysics couplings **Acoustic Streaming Boundary Coupling** or **Acoustic Streaming Domain Coupling**.

Frequency-Transient in the *COMSOL Multiphysics Reference Manual*

14

Acoustic Properties of Fluids

The Acoustics Module contains a number of models describing attenuation processes and loss mechanisms, and they have various inputs for the acoustic relevant material properties of fluids. This chapter provides information about those properties and puts them all in context.

In this chapter:

- Material Properties
- The Bulk Viscosity
- Attenuation and Loss Models
- References for The Acoustic Properties of Fluids

Material Properties

The Acoustics Module includes two material databases: Liquids and Gases, with temperature-dependent fluid dynamic and thermal properties, and a Piezoelectric Materials database with common piezoelectric materials.

Probably, the two most common materials used in acoustics are air and water. In this section we shortly discuss the definition and properties of the built-in Air and Water, liquid materials from the Material Library.

The built-in air and water materials are general and to some extent simplified models. For example, the air represents dry air and the material properties do not depend on moisture. This means that for high-precision absolute-value simulations, detailed material data need to be entered. Two options exist:

1. Using the **Predefined System** for **Moist air**, **Dry air**, or **Water-steam** defined in the Thermodynamics feature available with the *Liquid & Gas Properties Module*. Once set up, a material can be automatically generated.
2. Create your own material, for example, following the IEC standard for moist air (see Ref. 13). Set up a material and let it depend on the necessary model inputs (pressure, temperature, or relative humidity). Then, set up the necessary interpolation functions or analytical expressions and store the material under the **User-Defined Library** for future use.

Moist air is generated from **Thermodynamics** in the *Pressure Reciprocity Calibration Coupler with Detailed Moist Air Material Properties* tutorial. Application Library path **Acoustics_Module/Tutorials,_Thermoviscous_Acoustics/pressure_reciprocity_calibration_coupler**

The Air and Water, liquid materials were updated for the COMSOL version 5.4 release. Some material data has been added to simplify modeling, and inconsistencies due to rounding have been removed.

To take advantage of the new and updated material data, when opening an existing model (older than version 5.4), delete the material and add it again from the Materials window.

For detailed information about Materials, the Liquids and Gases Material Library, and the Piezoelectric Materials Library see the *COMSOL Multiphysics Reference Manual.*

For detailed information about the **Predefined system** see the Thermodynamics chapter in the *Liquid & Gas Properties User's Guide.*

THE (DRY) AIR MATERIAL

The built-in Air material, located both in the Built-In and the Liquids and Gases library, is commonly used in applications and models. The material properties represent those of dry air without the inclusion of moisture.

The air material defines common material parameters and their dependency on the ambient pressure p_A and temperature T. The relations are simplified and not all thermodynamic dependencies are taken into account. The speed of sound c and the density ρ are defined through the ideal gas law (assuming adiabatic behavior) following

$$c^2 = \gamma \frac{R}{M_n} T \qquad \rho = \frac{M_n p_A}{RT}$$

with the ratio of specific heats $\gamma = 1.4$, the gas constant R (COMSOL has a built-in constant called `R_const`), and the molar mass $M_n = 0.02897$ kg/mol. This is an idealization of air valid in many cases; see Ref. 2. This means that the speed of sound is not a function of the ambient pressure for this built-in Air material.

Some material properties are only temperature dependent and are given by polynomial fit curves like:

- The dynamic viscosity $\mu = \mu(T)$
- The (specific) heat capacity at constant pressure $C_p = C_p(T)$
- The thermal conductivity $k = k(T)$

Some properties are defined as constant:

- The ratio of specific heats $\gamma = 1.4$
- The molar mass $M_n = 0.02897$ kg/mol

Furthermore, there are material parameters and properties that are derived from their definitions and the above material data:

- The bulk viscosity μ_B is defined as $\mu_B = 0.6 \cdot \mu(T)$ in order to comply with the absorption behavior (see discussion in The Bulk Viscosity section).
- The (isobaric) coefficient of thermal expansion is defined as $\alpha_p = 1/T$, following the ideal gas law.
- The parameter of nonlinearity B/A is also defined for gases following the ideal gas law as $B/A = (\gamma+1)/2$ (see Ref. 12).

THE (CLEAN) WATER MATERIAL

The built-in Water, liquid material, located both in the Built-In and the Liquids and Gases library, is also commonly used in applications and models. These material properties represent those of clean water without the dependency on, for example, salinity or pH value necessary to describe salt water in the ocean.

Specifically for acoustics applications, the water material properties include properties that are temperature dependent like:

- The speed of sound $c = c(T)$
- The density $\rho = \rho(T)$
- The dynamic viscosity $\mu = \mu(T)$
- Thermal conductivity $k = k(T)$
- The heat capacity at constant pressure $C_p = C_p(T)$

Furthermore, there are detailed material parameters that are derived from the above definitions:

- The bulk viscosity is defined as $\mu_B = 2.79 \cdot \mu(T)$. In order to comply with the absorption behavior, (see the discussion in The Bulk Viscosity section).
- The (isobaric) coefficient of thermal expansion is defined through the thermodynamic definition as

$$\alpha_p = -\frac{1}{\rho}\left[\frac{\partial \rho}{\partial T}\right]_{p_A}$$

- The ratio of specific heats $\gamma = \gamma(T)$ is based on the thermodynamic relation

$$\gamma = 1 + \frac{T\alpha_p^2 c^2}{C_p}.$$

This definition is necessary in detailed models solving the thermoviscous acoustics or the linearized Navier–Stokes equations in water. The definition ensures that it is not just set to $\gamma = 1$.

The Bulk Viscosity

The bulk viscosity μ_B of a fluid is also known as the volume viscosity or the second viscosity. It is often not used in classical CFD applications but it plays a role in acoustics as well as in high-velocity compressible flows. The bulk viscosity is associated with losses due to expansion and compression, that is, the compressional resistance. The parameter describes the difference between the mechanical and thermodynamic pressures. It is often described as representing all the effects that describe the time it takes to establish thermodynamic equilibrium for vibrational and rotational motion of the fluid molecules (see Ref. 18). In some sense, it can be said to describe low frequency relaxation processes.

In acoustics, the losses associated with the bulk viscosity only really play a role in high-frequency ultrasound propagation. The bulk viscosity term in the thermoviscous acoustics or the linearized Navier–Stokes equations is sometimes used to model the frequency dependent losses due to relaxation processes. Various models exist to include these effects; details can be found in Ref. 17.

The Value of the Bulk Viscosity

The value of the bulk viscosity is difficult to measure and typically requires absorption experiments to be determined. For most fluids, its numerical value is of the same order as the dynamic viscosity. For some further details see, for example, Ref. 14 for fluids, Ref. 16 for water values, and Ref. 15 for gases.

The bulk viscosity enters the classical (pure) thermoviscous attenuation coefficient defined as

$$\alpha_{tv} = \frac{\omega^2}{2\rho c^3}\left(\frac{4}{3}\mu + \mu_B + \frac{k(\gamma-1)}{C_p}\right)$$

This is the attenuation experienced by plane propagating waves away from boundaries. The expression is derived directly from the full thermoviscous acoustics equations. In the **Air** and **Water, liquid** materials in the Material Library, the bulk viscosity is defined to match the high-frequency absorption properties of the fluids.

In Figure 14-1 standardized atmosphere attenuation curves (see the Atmosphere Attenuation Fluid Model in pressure acoustics) are depicted for various relative humidities together with the classical thermoviscous attention. The graph also shows

the case for zero bulk viscosity. The bulk viscosity is defined such that the thermoviscous absorption curve matches the standard at high frequencies, this happens for $\mu_B = 0.6 \cdot \mu$. This corresponds to how the value is determined in an experimental setup.

The same graphs are depicted in Figure 14-2 for water using the Ocean Attenuation Fluid Model, here for various values of the practical salinity (with a pH value of 7 and evaluated at the sea surface). The attenuation match gives a value for the bulk viscosity of $\mu_B = 2.79 \cdot \mu$. This is also the value reported in the literature (see Ref. 14 and 16).

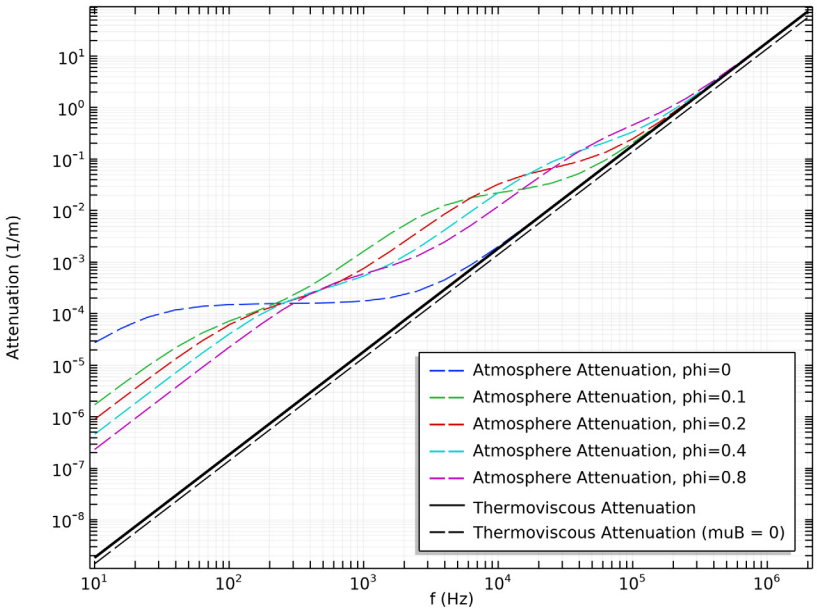


Figure 14-1: Atmosphere attenuation for various relative humidities (phi), the full classical thermoviscous attenuation, and thermoviscous attenuation for zero bulk viscosity.

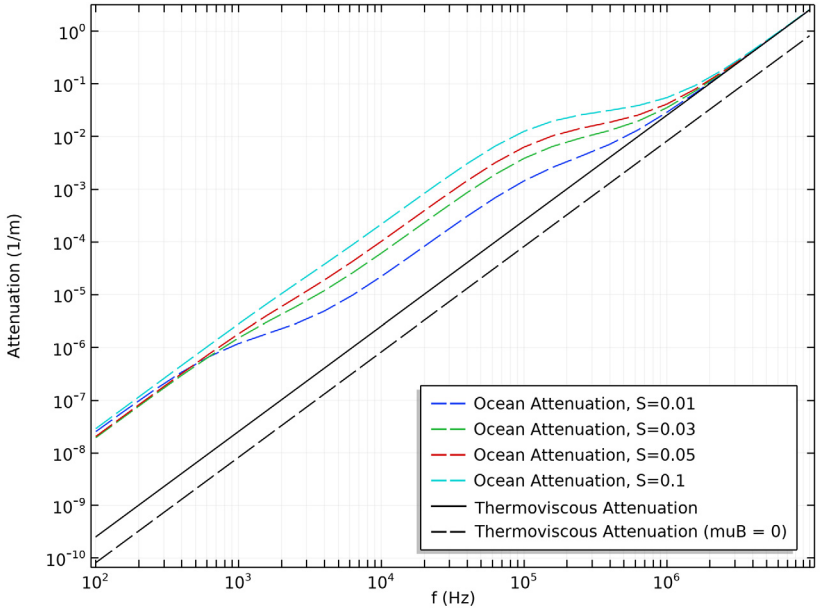


Figure 14-2: Ocean attenuation for various practical salinities (S·1000), the full classical thermoviscous attenuation, and thermoviscous attenuation for zero bulk viscosity.

Attenuation and Loss Models

The Acoustics Module includes several different physics interfaces and various features that introduce acoustic losses. These describe everything from thermoviscous boundary layer losses to the attenuation experienced when acoustic waves propagate over long distances in the ocean. The models have different applicability depending on the frequency regime and the system modeled.

This section first gives a short description of the different loss mechanism, then the applicability and the regime where boundary layer and bulk losses apply is discussed, and finally some remarks are given on the bulk viscosity and how it is related to attenuation.

In this section:

- Loss Mechanisms
- The Atmosphere and Ocean Attenuation Models
- Boundary Layer vs. Bulk Losses

Loss Mechanisms

Loss mechanisms in acoustics due to viscosity, thermal conduction, relaxation processes, and other processes cause absorption and dissipation of the acoustic energy. These result in a reduction in the pressure wave amplitude. This is not due to geometrical spreading where no energy is lost, but because heating actually takes place. When an acoustic wave undergoes absorption, this process is in general also accompanied by dispersion, that is, the dependence of the speed of sound on frequency.

Loss mechanism in acoustics can roughly be divided into four categories, but can of course also happen simultaneously. The division is mostly conceptual:

1. Bulk or volume losses are associated with the propagation of waves over long distances or at very high frequencies (also known as internal damping). The (plane wave) attenuation coefficient α (SI unit: 1/m) is often associated with this mechanism. Several loss models are included in the Pressure Acoustics model (see also Theory for the Equivalent Fluid Models) including: User-Defined Attenuation Fluid Model, Atmosphere Attenuation Fluid Model, Ocean Attenuation Fluid Model, or Thermally Conducting and/or Viscous Fluid Model. Note that the

Thermally Conducting and/or Viscous Fluid Model should not be confused with boundary layer losses (see next point). Bulk losses are due to several different mechanisms including viscous and thermal dissipation, relaxation processes, and other loss mechanism.

2. Viscous and thermal boundary-layer losses occur at hard walls because of the effective no-slip and isothermal conditions. These are most important in geometries of small dimensions comparable to the boundary layer thickness. The losses can be included in a very general manner using one of the Thermoviscous Acoustics Interfaces. The Narrow Region Acoustics feature can be used in narrow waveguides of constant cross section, while the Thermoviscous Boundary Layer Impedance condition can be used an any boundary as long as no boundary layers are overlapping. The latter two features are great engineering approximations (often yielding exact results) to include the losses at a lower computational cost using The Pressure Acoustics, Frequency Domain Interface.

3. In porous materials, losses are again due to viscous and thermal boundary layer losses, here inside the channels of the porous matrix. The losses are also caused by damping because of the coupling to the porous matrix structure. Detailed modeling is done using The Poroelastic Waves Interface that solves the full Biot model for the coupled pressure and structural waves. A simplified so-called equivalent fluid description can be done using the Poroacoustics feature (see also Theory for the Equivalent Fluid Models).

- Losses can also occur due to interaction with the surroundings like solids and membranes. This is best modeled using multiphysics, see Acoustic–Structure Boundary in the Multiphysics Couplings chapter. Here, simplified models exist using the many options available with the Impedance boundary conditions (see Theory for the Boundary Impedance Models).

The Atmosphere and Ocean Attenuation Models

The Pressure Acoustics, Frequency Domain Interface and The Ray Acoustics Interface include two semianalytical fluid models that are calibrated with extensive measurement data. This is the **Atmospheric attenuation** and the **Ocean attenuation** models. Both include effects due to viscosity and thermal conduction and relaxation processes of molecules.

The **Atmospheric attenuation** model defines attenuation in atmospheric air that follows the ANSI standard S1.26-2014, see Ref. 3 (Appendix B), 4, 5, and 6 for details. The model describes attenuation due to thermal and viscous effects (primarily pure air), the

relaxation processes for nitrogen and oxygen, and the dependency on atmospheric pressure (absolute pressure), temperature, and relative humidity. The attenuation in air is important for propagation over large distances and for high frequency processes. This also means that the attenuation effect is more important in ray tracing simulations where propagation can be simulated over much larger distances and especially at higher frequencies. In the next section, we will see when the full details of this model are necessary when compared to boundary layer losses.

The **Ocean attenuation** model defines attenuation in seawater of the ocean. The model is based on a semianalytical model with parameters that are based on extensive measurement data. No standard exists for the attenuation in sea water. It includes effects due to viscosity in pure water, the relaxation processes of boric acid and magnesium sulfate, as well as depth, temperature, practical salinity, and pH value. For further details, see Ref. 7, 8, 9, 10, and 11. Like the atmosphere model, the ocean attenuation model is important in ray tracing simulations where propagation can be simulated over much larger distances.

Boundary Layer vs. Bulk Losses

As discussed in the above sections, the Acoustics Module includes models for both bulk losses and boundary layer losses. In pressure acoustics, you can include either form of losses. For example, the Atmosphere attenuation fluid model in pressure acoustics is a detailed bulk loss model that includes all dissipation effects and it complies with the ANSI standard. On the other hand, the Narrow Region Acoustics feature includes the boundary layer losses for narrow waveguide structures.

The importance of the two loss mechanisms (bulk and boundary layer) depend on the geometry scale and the frequency range studied. As an example, the losses experienced by a plane wave propagating in an air filled narrow circular duct is depicted in Figure 14-3. The graph includes:

- The bulk atmosphere attenuation for different relative humidities (ϕ = 0%, 20%, and 80%).
- The classical (bulk) thermoviscous attenuation.
- The thermoviscous boundary layer attenuation for different duct radii (a = 0.5 mm, 2.0 mm, and 10.0 mm) modeled using the Narrow Region Acoustic feature.
- The solution using the full thermoviscous acoustics equations in the cylindrical duct of radius a = 2.0 mm. This can be seen as a reference solution since it models both boundary layer and thermoviscous bulk losses.

From the graph, it is evident that in relatively narrow waveguide structures the boundary layer losses far surpass the bulk losses in a large frequency range. The transition happens in the ultrasound regime. The graph also shows that (for air) the real life relaxation effects captured by the atmosphere attenuation model can be disregarded in most waveguides and really only are important away from boundaries and for propagation over large distances.

The transition between the two loss regimes is captured by the full thermoviscous acoustics models. For the vast majority of practical applications, when using the Thermoviscous Acoustics physics interfaces, it is not necessary to include the "true" bulk behavior captured by the atmosphere model. This is simply because it is not computationally possible to model large scale acoustics with the Thermoviscous Acoustics interfaces - their true application is in complex narrow geometries.

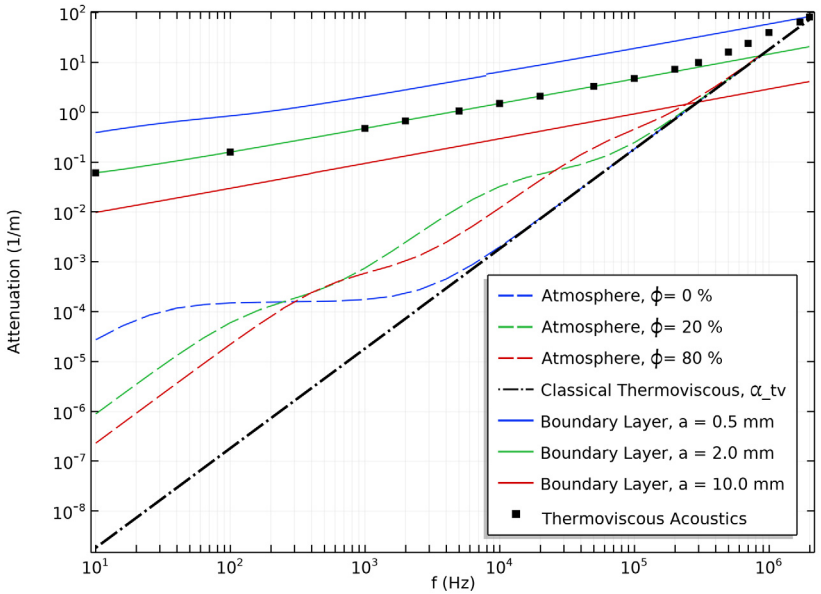


Figure 14-3: The importance of various attenuation mechanisms for the propagation of plane waves in a cylindrical duct of radius a. The plot shows the (bulk) atmosphere attenuation contribution for different relative humidities ϕ, the classical (bulk) thermoviscous attenuation, the thermoviscous boundary layer attenuation for different duct radii a, and reference solution using the full thermoviscous acoustics model (in the duct with a radius of 2.0 mm).

References for The Acoustic Properties of Fluids

1. S. Temkin, *Elements of Acoustics*, Acoustical Society of America, 2001.

2. A.D. Pierce, *Acoustics: An Introduction to its Physical Principles and Applications*, Acoustical Society of America (second print), 1991.

3. D.T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.

4. . E. Bass, L.C. Sutherland, A.J. Zuckerwar, D.T. Blackstock, and D.M. Hester, "Atmospheric absorption of sound: Further developments," *J. Acoust. Soc. Am.*, vol. 97, pp. 680–683, 1995; "Erratum," *J. Acoust. Soc. Am.*, vol. 99, p. 1259. 1996.

5. ANSI S1.26-2014 (supersedes ANSI S1.26-1995) "American National Standard method for calculation of the absorption of sound by the atmosphere," (Acoustical Society of America, New York, 2014).

6. National Physics Lab (NPL), "NPL Acoustics: Calculation of absorption of sound by the atmosphere," http://resource.npl.co.uk/acoustics/techguides/absorption/

7. R.E. Francois and G.R. Garrison, "Sound absorption based on ocean measurements: Part I: Pure water and magnesium sulfate contributions," *J. Acoust. Soc. Am.*, vol. 72, pp. 896–907, 1982.

8. R.E. Francois and G.R. Garrison, "Sound absorption based on ocean measurements: Part II: Boric acid contribution and equation for total absorption," J. *Acoust. Soc. Am.*, vol. 72, pp. 1879–1890, 1982.

9. M.A. Ainslie and J.G. McColm, "A simplified formula for viscous and chemical absorption in sea water," *J. Acoust. Soc. Am.*, vol. 103, pp. 1671–1672, 1998.

10. F.H. Fisher and V.P. Simmons, "Sound absorption in seawater," *J. Acoust. Soc. Am.*, vol. 62, pp. 558–564, 1977.

11. National Physics Lab (NPL), "Calculation of absorption of sound in seawater," http://resource.npl.co.uk/acoustics/techguides/seaabsorption/

12. M.F. Hamilton and D.T. Blackstock, "On the coefficient of nonlinearity in nonlinear acoustics," J. *Acoust. Soc. Am.*, vol. 83, pp. 74–77, 1988.

13. IEC 61094-2:2009, "Electroacoustics — Measurement microphones — Part 2: Primary method for pressure calibration of laboratory standard microphones by the reciprocity technique," International Standard.

14. A.S. Dukhin and P.J. Goetz, "Bulk viscosity and compressibility measurements using acoustic spectroscopy," *J. Chem. Phys.*, vol. 130, p. 124519-1, 2009.

15. M.S. Cramer, "Numerical estimate for the bulk viscosity of ideal gases," *Phys. Fluids*, vol. 30, p. 066102, 2012.

16. P.B. Muller and H. Bruus, "Numerical study of thermoviscous effects in ultrasound-induced acoustic streaming in microchannels," *Phys. Rev. E*, vol. 90, p. 043016, 2014.

17. E.M. Viggen, *The lattice Boltzmann method: Fundamentals and acoustics*, PhD Thesis, NTNU-Trondheim, 2014.

18. P. M. Morse and K.U. Ingard, *Theoretical Acoustics*, Princeton University Press, 1986.

15

Glossary

This Glossary of Terms contains finite element modeling terms in an acoustics context. For mathematical terms as well as geometry and CAD terms specific to the COMSOL Multiphysics® software and documentation, see the glossary in the *COMSOL Multiphysics Reference Manual*. For references to more information about a term, see the index.

Glossary of Terms

acoustic impedance At a specified surface, the complex quotient of acoustic pressure by normal fluid velocity. SI unit: Pa/(m/s).

acoustic reactance The imaginary part of the acoustic impedance.

acoustic resistance The real part of the acoustic impedance.

acoustic-structure interaction A multiphysics phenomenon where the fluid's pressure causes a fluid load on the solid domain, and the structural acceleration affects the fluid domain as a normal acceleration across the fluid-solid boundary.

adiabatic bulk modulus One over the compressibility β_s measured at constant entropy. The adiabatic bulk modulus is denoted K_s and gives a measure of the compressibility of the fluid and is directly related to the speed of sound c_s in the fluid. SI unit: Pa.

$$K_s = \frac{1}{\beta_s} = \rho_0 c_s^2$$

admittance The reciprocal of impedance.

aeroacoustics The scientific field of study used to couple acoustics and fluid dynamics.

anisotropy Variation of material properties with direction.

arbitrary Lagrangian-Eulerian (ALE) method A technique to formulate equations in a mixed kinematical description. An ALE referential coordinate system is typically a mix between the material (Lagrangian) and spatial (Eulerian) coordinate systems.

Bernoulli equation An integrated form of Euler's momentum equation along a line of flow. The equation gives an expression for an invariant quantity in an inviscid fluid. A decrease in the speed of the fluid translates to an increase in the fluid pressure and/or potential energy.

bulk modulus One over the compressibility. It gives a measure of the compressibility of the fluid and is related to the speed of sound in the fluid. SI unit: Pa. See also *adiabatic bulk modulus.*

characteristic impedance The product of the equilibrium density and the speed of sound in a medium. SI unit: Pa/(m/s).

compliance Reciprocal of *stiffness*.

compliance matrix The inverse of the elasticity matrix. See *elasticity matrix*.

Cauchy stress The most fundamental stress measure defined as force/deformed area in fixed directions not following the body.

constitutive equations Equations that relate two physical quantities. In thermoviscous acoustics both the stress tensor (relating velocity to stress) and Fourier's law of heat conduction (relating heat conduction to temperature) are constitutive relations. In structural mechanics this is the equation formulating the stress-strain relationship of a material. Constitutive equations are supplemented by equilibrium equations (mass, momentum, and energy) and an *equation of state* to provide a full physical description.

creep Time-dependent material nonlinearity that usually occurs in metals at high temperatures in which the effect of the variation of stress and strain with time is of interest.

damping Dissipation of energy in the fluid or a vibrating structure. The damping is typically due to viscous losses or thermal conduction. In acoustics this happens in structures with small geometrical dimensions, for example, small pipes or porous materials. In structures a common assumption is viscous damping where the damping is proportional to the velocity. See also *Rayleigh damping*.

decibel (dB) Logarithmic unit that indicates the ratio of a physical quantity relative to a reference value.

dipole source An acoustic source that behaves as a translational oscillating sphere.

Doppler effect Change in the observed frequency of a wave caused by a time rate of change in the effective length of the path of travel between the source and the observation point.

effective sound pressure RMS instantaneous sound pressure at a point during a time interval, T, long enough that the measured value is effectively independent of small changes in T. SI unit: Pa = N/m^2.

equation of state The thermodynamic relation between three independent thermodynamic variables. Typically in acoustics it is the density $\rho = \rho(p,s)$ given as function of the entropy s and the pressure p.

eigenmode A possible propagating mode of an acoustic wave.

elasticity matrix The matrix D relating strain to stresses:

$$\sigma = D\varepsilon$$

Eulerian Model described and solved in a coordinate system that is fixed (spatial). See also *Lagrangian* and *arbitrary Lagrangian–Eulerian method*.

Green–Lagrange strain Nonlinear strain measure used in large-deformation analysis. In a small strain, large rotation analysis, the *Green–Lagrange strain* corresponds to the engineering strain, with the strain values interpreted in the original directions. The *Green–Lagrange* strain is a natural choice when formulating a problem in the undeformed state. The conjugate stress is the *second Piola–Kirchhoff stress*.

impedance At a specified frequency, the quotient of a dynamic field quantity (such as force, sound, or pressure) by a kinematic field quantity (such as vibration velocity or particle velocity).

instantaneous sound pressure Total instantaneous pressure at a point in a medium minus the static pressure at the same point. SI unit: Pa = N/m^2.

irrotational background velocity field A velocity field $\mathbf{u}$ that has the property of having rotation $\nabla \times \mathbf{u} = \mathbf{0}$ everywhere, where the first term is the vorticity of the fluid. In such a fluid the viscous stress does not contribute to the acceleration of the fluid. The mean pressure in this fluid is described by *Bernoulli's equation*.

Lagrangian Model described and solved in a coordinate system that moves with the material. See also *Eulerian* and *arbitrary Lagrangian–Eulerian method*.

monopole source An acoustic source that behaves as a radially oscillating sphere.

particle velocity In a sound field, the velocity caused by a sound wave of a given infinitesimal part of the medium relative to the medium as a whole.

PML (perfectly matched layer) Domain adjoined at a system boundary designed to emulate a nonreflecting boundary condition independently of the shape and frequency of the incident wavefront.

principle of virtual work States that the variation in internal strain energy is equal to the work done by external forces.

propagating acoustic modes The acoustic modes or wave shapes that propagate with no significant damping for a given frequency in a duct of a given cross-section.

Rayleigh damping A viscous damping model where the damping is proportional to the mass and stiffness through the mass and stiffness damping parameters.

reference sound pressure See definition in the entry for *sound pressure level*.

resonance frequency A frequency at which the system has the tendency to oscillate at a greater amplitude than at nonresonance frequencies. At the resonance frequencies the system can easily transfer energy from the actuation to the vibrating structure or acoustic wave.

RMS value Root-mean-square value; for the (complex) sound pressure, $p(t)$, over the time interval $T_1 < t < T_2$ defined as

$$p_{\mathrm{RMS}} = \sqrt{\frac{1}{T_2 - T_1}\int_{T_1}^{T_2} \mathrm{Re}[p(t)]^2 dt}$$

For a harmonic pressure wave, $p(t) = p_0 e^{i\omega t}$, the time interval is taken to be a complete period, resulting in $p_{\mathrm{RMS}} = p_0/\sqrt{2}$.

second Piola-Kirchhoff stress Conjugate stress to *Green–Lagrange strain* used in large deformation analysis.

sound energy Total energy in a given part of a medium minus the energy that would exist at the same part in the absence of sound waves. SI unit: J.

sound-energy flux density See *sound intensity*.

sound intensity Average rate of *sound energy* transmitted in a specified direction at a point through a unit area normal to this direction. SI unit: $\mathrm{W/m^2}$.

sound pressure See *effective sound pressure*.

sound pressure amplitude Absolute *instantaneous sound pressure* in any given cycle of a sound wave at some specified time. SI unit: W/m^2.

sound power density See *sound intensity*.

sound pressure level Ten times the logarithm (to the base ten) of the ratio of the time-mean-square pressure of a sound, in a stated frequency band, to the square of a *reference sound pressure*, p_{ref}. For gases, $p_{ref} = 20\ \mu Pa$, for other media (unless otherwise specified) $p_{ref} = 1\ \mu Pa$. Unit: dB (*decibel*).

sound source strength Maximum instantaneous rate of volume displacement produced by a source when emitting a harmonic sound wave. SI unit: m^3/s.

specific acoustic impedance At a point in a sound field, the quotient of sound pressure by particle velocity. SI unit: Pa/(m/s).

speed of sound The rate of change of particle displacement with distance for a sound wave. SI unit: m/s.

spin tensor The skew-symmetric part of the velocity gradient tensor.

static pressure Pressure that would exist at a point in the absence of a sound wave.

stiffness Ratio of change of force (or torque) to the corresponding change in translational (or rotational) displacement of an elastic element.

thermoviscous acoustics The interaction between thermodynamic and acoustic phenomena, which takes into account the temperature oscillations that accompany the acoustic pressure oscillations. The combination of these oscillations produces *thermoviscous acoustic* effects. Thermoviscous acoustic phenomena are modeled by solving the full linearized Navier–Stokes equation (momentum equation), the continuity equation, and the energy equation. Thermoviscous acoustics is also known as viscothermal acoustics or thermoacoustics.

velocity potential When a flow is irrotational $\nabla \times \mathbf{u} = \mathbf{0}$ the vector field (velocity field) can always be derived from a scalar potential $\phi(\mathbf{x})$ as $\mathbf{u} = \nabla\phi$, where ϕ is the velocity potential. See also *irrotational background velocity field*.

waveguide structures Structures that have the property of guiding sound waves. See also *propagating acoustic modes*.

Index

A

accumulator (node), boundaries 784
accumulator (node), domains 774
accuracy order, of time stepping 760
acoustic diffusion equation interface 861
acoustic diffusion model (node) 865
acoustic intensity vector 260
acoustic-elastic waves interaction interface 446
acoustic-piezoelectric interaction, frequency domain (acpz) interface 438
acoustic-piezoelectric interaction, transient (acpztd) interface 441
acoustic-poroelastic waves interaction interface 444
acoustic-porous boundary (multiphysics coupling) 932
acoustic-shell interaction, frequency domain interface 451
acoustic-shell interaction, transient interface 454
acoustic-solid interaction, frequency domain interface 432
acoustic-solid interaction, transient interface 435
acoustic-structure boundary (multiphysics coupling) 919
acoustic-thermoacoustic boundary (multiphysics coupling) 930
acoustic-thermoacoustic interaction, frequency domain 670
adiabatic (node) 517, 640
adiabatic bulk modulus 292
adiabatic, reversible processes 698
aeroacoustic-structure boundary (multiphysics coupling) 928
analysis. see study types.
Application Libraries window 54
application library examples
  acpr interface 70, 932–933, 940, 974
  actd interface 167, 172, 299
  ae interface 528, 542, 551
  background pressure field 97, 101
  Biot equivalent fluid model 343
  cpf interface 555
  cylindrical wave radiation 139
  eigenfrequency study 297
  Eigenmodes in a Muffler 187
  far field calculation 118, 129, 137, 150, 180, 184
  far field plots 270
  flow point source 156
  fluid model 83
  Gaussian pulse point source 176
  Generic 711 Coupler — An Occluded Ear-Canal Simulator 107
  Lumped Receiver Connected to Test Setup with an 0.4-cc Coupler 90
  mass flow point source 545
  mode analysis study 965
  pipe acoustics, transient 902
  point source 109
  poroelastic waves 365, 377
  pressure acoustics, frequency domain 70, 932–933, 940, 974
  spherical wave radiation 190
  ta interface 601, 923
artificial boundary conditions 43
asymptotic far-field radiation (node) 482
attenuation coefficient 76, 703
auxiliary dependent variable 783
auxiliary dependent variable (node) 798
axial symmetry (node) 102, 546

B
background potential flow (multiphysics coupling) 933
background pressure field (node) 94, 173
background pressure wave 66
background velocity (node) 775
barotropic fluids 588
Bernoulli equation 588
Biot theory 417
Biot's high frequency range 418
Bloch periodicity 477, 508, 628
boundary mode acoustics (acbm) interface 186
theory 299
boundary nodes
acpr interface 70
actd interface 168
ae interface 528
aebm interface 552
aetd interface 548
cpf interface 555
cwe interface 715
lef and let interfaces 467
lnsf and lnst interfaces 497
pelw interfaces 367
ta interface 601, 645
tabm interface 657
boundary-layer absorption 347
bulk modulus 896
elastic moduli 395
pressure acoustics 40

C
caustic surface 839
CFL condition 901
circular duct 350
circular source (node) 162
circumferential wave number 295, 300
common settings 52
complex conjugate operator 263
complex impedance 41
complex wave numbers 41
compressible potential flow (cpf) interface 554
compressible potential flow model (node) 556
compute intensity 757
compute intensity and power 758
compute intensity and power in graded media 758
compute intensity in graded media 758
continuity (node)
acpr interface 151
ae interface 483, 544
lns interface 518
convected wave equation interface 712
count reflections 759
Coupling Operators 827
Courant number 902
cwe nodes
acpr interface 715
cylindrical wave radiation (node) 139

D
damping
boundaries 43
degrees of freedom 241
Density of exterior domains 757, 825
diffuse scattering, wall condition 778
dipole point source (node) 156
dipole source (node) 98
dipole sources 292
disappear, wall condition 778
dispersion curves 965
dispersion relation 704
documentation 53
domain nodes
acpr interface 70
actd interface 168
ae interface 528
aetd interface 548

cpf interface 555
cwe interface 715
lef and let interfaces 467
lnsf and lnst interfaces 497
pelw interfaces 367
ta interface 601, 645
domain source (node) 869
domain sources (node) 472, 503
double dot operator 689

E edge nodes
acpr interface 70
actd interface 168
ae interface 528
aebm interface 552
aetd interface 548
eigenfrequency study 962
pressure acoustics 296
elastic waves (elw) interface
theory 416
elkernel element 305
emailing COMSOL 54
empty study 969
end impedance (node) 892
equilateral triangular duct 350
equivalent fluid model 75, 199, 333
equivalent fluid models 328
Euler equations 587
evanescent modes 964
evanescent wave components 265

F far field variables 268
far-field calculation (node) 114
far-field limits 267, 304
far-field regions 265
far-field variables 267
first order material parameters (node)
503
fitted domain (node) 869
fixed (node) 381
Floquet periodicity 477, 508, 628
fluid models 328
fluid models, pressure acoustics 74, 198
Fraunhofer diffraction 266
freeze, wall condition 778
frequency domain modal study 967
frequency domain study 961
ae interface 585
Fresnel numbers 266

G general reflection, wall condition 778
Green's function 303

H Hankel function 304
harmonic time dependence 260
heat flux (node) 517
heat source (node) 608
Helmholtz–Kirchhoff integral 266, 303
Higdon conditions 301
high frequency range 418

I ideal barotropic fluids 588
ideal gas fluid model 79
impedance (node) 475
acpr interface 104
ae interface 536
impedance, complex 41
incident acoustic fields (node) 472
incident pressure field (node) 140, 175
incident velocity potential (node) 534
initial values (node)
abe interface 868
acbm interface 188
acpr interface 97, 720
ae interface 531
cpf interface 557
elw interface 378
lem interface 471
let interface 487
lnsf and lnst interfaces 504

pafd interface 888
patd interface 888
ta interface 608
inlet on axis (node) 795
insertion loss curves 260
intensity 757
intensity attenuation coefficient 76
intensity computation 837–838
intensity variables 260
interior impedance (node) 146, 481
interior normal acceleration (node) 144
interior normal displacement (node) 145, 182
interior normal impedance (node) 511, 630
interior normal velocity (node) 145, 182
interior perforated plate (node) 147
interior sound hard boundary (wall) (node)
acpr interface 144
ae interface 543
interior velocity (node) 631
interior wall (node) 480, 510, 629
internet resources 52
inward energy flux (node) 868
irrotational velocity fields 584
isentropic processes 698
isentropic speed of sound 688
isothermal (node) 516, 640
isotropic meshes 241
isotropic scattering, wall condition 778

J
JCA model 84
JCAL model 85
JCAPL model 85

K
knowledge base, COMSOL 55
Korteweg formula 896

L
Lagrange elements 241
Lamé parameters 395
large acoustics problems 247
Lax-Friedrichs flux parameter 397, 718
Lax–Friedrichs flux parameter 214
limp frame 81
limp porous matrix model 333
line source (node) 151
line source on axis (node) 153
linear elastic fluid model 328
linearized Euler model (node) 467
linearized Euler, frequency domain interface 463
linearized Euler, transient interface 484
linearized Navier-Stokes model (node) 498
linearized Navier-Stokes, boundary mode interface 522
linearized Navier-Stokes, frequency domain interface 492
linearized Navier-Stokes, transient interface 519
linearized potential flow model (node) 529
linearized potential flow, boundary mode interface 550
linearized potential flow, frequency domain interface 525
linearized potential flow, transient interface 547
liquids and gases materials 273, 974
low reduced frequency (LRF) 350

M
mapped room coupling (node) 870
mass flow (node) 558
mass flow circular source (node) 545
mass flow edge source (node) 544
mass flow line source on axis (node) 545
mass flow point source (node) 545
mass flows 532

matched boundary (node) 113
matched boundary conditions 43
material discontinuity (node) 776
materials
porous absorbing 41
medium properties (node) 763
microacoustics 594
mixed diffuse and specular reflection, wall condition 778
mode analysis study 964–965
boundary mode acoustics 299
theory 703
monopole point source (node) 154
monopole source (node) 97
monopole sources 292
motionless skeleton model 333
moving wall (node) 479, 487
MPH-files 54
multigrid solvers 247
multiphysics 432, 435, 451, 454, 670, 673
multiphysics coupling
acoustic-porous boundary (node) 932
acoustic-structure boundary (node) 919
acoustic-thermoacoustic boundary (node) 930
aeroacoustic-structure boundary (node) 928
background potential flow (node) 933
porous-structure boundary (node) 933
thermoviscous acoustic-structure boundary (node) 922
multiscale acoustics modeling 806

N narrow region acoustics (node) 87
near-field regions 265
no slip (node) 512, 635
no stress (node) 514, 637
nodes, common settings 52
no-flow conditions 531, 543
nonlinear pressure acoustics time explicit interface 725
nonlocal accumulator (node) 792
nonreflecting boundary conditions 43
normal acceleration (node) 102–103, 177–178, 221, 721
normal flow (node) 558
normal impedance (node) 515, 638
normal mass flow (node) 532
normal velocity (node) 535
NRBC. see nonreflecting boundary condition. 113
Nyquist criterion 252

O outflow boundary (node) 483
out-of-plane wave number 295, 585

P pair impedance (node)
acpr interface 146
ae interface 542
pair nodes
acpr interface 70
ae interface 528
aebm interface 552
aetd interface 548
cpf interface 555
cwe interface 715
lef and let interfaces 467
lnsf and lnst interfaces 497
pelw interfaces 367
ta interface 601, 645
tabm interface 657
pair perforated plate (node) 147
pair room coupling (node) 870
parameter of nonlinearity 298
particle velocity 260
pass through, wall condition 778
perfectly matched layers (node) 254
periodic condition (node) 110

phase factors, pressure acoustics 295
phase variables 962
physics interfaces, common settings 52
piezoelectric waves, time explicit interface 407
pipe acoustics, frequency domain (pafd) interface 884
pipe acoustics, transient (patd) interface 886
  theory 894
plane of incidence 839
plane wave radiation (node)
  acpr interface 138
  ae interface 533
point nodes
  acpr interface 70
  actd interface 168
  ae interface 528
  aebm interface 552
  aetd interface 548
  cwe interface 715
point source (node)
  ade interface 871
Poisson's ratio 395
poroacoustics (node) 80
poroelastic material (node) 368
poroelastic waves (elw) interface 364
poroelastic waves (pelw) interface
  theory 416
porous absorbing materials 41
porous-structure boundary (multiphysics coupling) 933
power dissipation variables 262
ppr() operator 118
Prandtl number 596
prescribed acceleration (node) 383
prescribed acoustic fields (node) 474
prescribed displacement (node) 381
prescribed pressure (node) 514
prescribed temperature (node) 517
prescribed velocity (node) 382, 513
pressure (adiabatic) (node) 506
pressure (isentropic) (node) 473
pressure (node) 104, 720
pressure acoustics (node) 73, 716
pressure acoustics interfaces 290
pressure acoustics model, fluid models 329
pressure acoustics models, theory 328
pressure acoustics theory 290
pressure acoustics, asymptotic scattering (paas) interface 229, 234
pressure acoustics, frequency domain (acpr) interface 65, 189
pressure acoustics, time explicit (pate) interface 206
pressure acoustics, time explicit model (node) 211
pressure acoustics, transient (actd) interface 165
pressure, adiabatic (node) 609
pressure-wave speeds 395
principal radii of curvature 838
propagating acoustic modes 703
propagating modes 964

Q quadrupole point source (node) 158

R radiation boundary conditions 43
random number generation 784
ray acoustics interface 754
ray continuity (node) 796
ray properties (node) 798
ray termination (node) 811
ray termination, theory 851
Rayl unit 105
rectangular duct 350
release (node) 766

release from boundary (node)
ray acoustics 788
release from data file (node) 803
release from edge (node) 797
release from exterior field calculation (node) 805
release from grid (node) 799
release from grid on axis (node) 803
release from point (node) 797
release from point on axis (node) 798
resonance frequency 962
results analysis and variables 260
rigid frame 81
rigid porous matrix model 333
rigid wall (node) 470
roller (node) 384
room (node) 866
room coupling (node) 869

S sequential linearized Navier-Stokes 660
shear modulus expression 395
shear-wave speeds 395
slip (node) 512, 636
slip velocity (node) 557
slit 350
SLNS 660
solving large problems 247
sound hard boundary (wall) (node)
acpr interface 101, 720
ae interface 531
sound pressure level calculation (node), boundaries 785
sound soft boundary (node)
acpr interface 104
ae interface 534
source term 262, 688
source with directivity (node) 808
specular reflection, wall condition 778
Speed of sound in exterior domains 757, 825
spherical wave radiation (node) 139
standard settings 52
stationary study 589, 961
stick, wall condition 778
study types
acbm interface 299
actd interface 297
ae interface 585
swirl-correction factor 894
symmetry (node) 110, 476, 720
ta interface 610
symmetry planes, far-field analysis 116

T Taylor series 895
technical support, COMSOL 54
temperature variation (node) 640
theory 290
acbm interface 299
elw and pelw interfaces 416
patd interface 894
pressure acoustics models 328
ta interface 692
tas interface 692
thermal dissipation 262, 688
thermally conducting and viscous fluid model 332
thermally conducting fluid model 331
thermoacoustics 594
thermoviscous acoustics model (node) 602, 658
thermoviscous acoustics, frequency domain (ta) interface 594, 654
theory 692
thermoviscous acoustic-shell interaction, frequency domain (tash) interface 675
thermoviscous acoustic-solid interaction, frequency domain 673

thermoviscous acoustic-solid interaction, frequency domain (tas) interface
theory 692
thermoviscous acoustic-structure boundary (multiphysics coupling) 922
time dependent study 966
pressure acoustics, transient 297
time-dependent, modal study 967
time-harmonic study
aeroacoustics 585
transient pressure acoustics model (node) 168
transmission loss 260

U U-P formulation 419
user-defined attenuation fluid model 330

V variables
for far fields 268
intensity 260
phase 962
power dissipation 262
results analysis 260
velocity (node) 637, 891
velocity potential (node) 532
velocity potential, compressible potential flow, and 588
very narrow circular ducts (isothermal) 349
viscothermal acoustics 594
viscous dissipation 262, 688
viscous fluid model 78, 331
vortex sheet (node) 543

W wall (node) 505, 608, 867
ray optics 777
wall condition 778
wave numbers 41, 295, 585
wave radiation equations 301
wave speeds 395
waveguide 964
waveguide structures 703
websites, COMSOL 55
wide ducts 348
Womersley number 682

Y Young's modulus expression 395