

RF Module

User's Guide

RF Module User's Guide

© 1998–2019 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; 9,454,625; and 10,019,544. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) 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 5.5

Contact Information

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

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

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

Part number: CM021001

C o n t e n t s

Chapter 1: Introduction

About the RF Module	12
What Can the RF Module Do?	12
What Problems Can You Solve?	13
The RF Module Physics Interface Guide	14
Common Physics Interface and Feature Settings and Nodes	16
Selecting the Study Type	16
The RF Module Modeling Process	17
Where Do I Access the Documentation and Application Libraries?	18
Overview of the User's Guide	22

Chapter 2: RF Modeling

Preparing for RF Modeling	27
Simplifying Geometries	28
2D Models	28
3D Models	30
Using Efficient Boundary Conditions	31
Applying Electromagnetic Sources	31
Meshing and Solving.	32
Periodic Boundary Conditions	33
Scattered Field Formulation	35
Modeling with Far-Field Calculations	37
Far-Field Support in the Electromagnetic Waves, Frequency Domain Interface.	37
The Radiation Pattern Plots	38

S-Parameters and Ports	44
S-Parameters in Terms of Voltage and Electric Field	44
S-Parameter Calculations.	45
S-Parameter Variables	45
Port Sweeps/Manual Terminal Sweeps and Touchstone Export	46
Lumped Ports with Voltage Input	47
About Lumped Ports	47
Lumped Port Parameters.	48
Lumped Ports in the RF Module	49
Jones Vectors and Polarization Plots	51
Jones Vectors for Polarization Analysis	51
Polarization Plots.	52
Jones Vector Variables.	52
Lossy Eigenvalue Calculations	54
Eigenfrequency Analysis	54
Mode Analysis	56
Connecting to Electrical Circuits	58
About Connecting Electrical Circuits to Physics Interfaces	58
Connecting Electrical Circuits Using Predefined Couplings	59
Connecting Electrical Circuits by User-Defined Couplings	59
Solving.	61
Postprocessing.	61
SPICE Import and Export	62
SPICE Import	62
SPICE Export	63
Reference	63
Reduced Order Modeling	64
Adaptive Frequency Sweep Using Asymptotic Waveform Evaluation (AWE) Method	64
Frequency Domain, Modal Method	66

Part Libraries	68
Material Libraries	69

Chapter 3: Electromagnetics Theory

Maxwell's Equations	72
Introduction to Maxwell's Equations	72
Constitutive Relations	73
Potentials.	75
Electromagnetic Energy	75
Material Properties	76
About the RF Materials Database.	78
Boundary and Interface Conditions	78
Phasors	79
Special Calculations	81
S-Parameter Calculations.	81
Far-Field Calculations Theory	84
References	85
Electromagnetic Quantities	86

Chapter 4: Radio Frequency Interfaces

The Electromagnetic Waves, Frequency Domain Interface	90
Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Frequency Domain Interface	97
Wave Equation, Electric	99
Divergence Constraint.	104
Initial Values	105
External Current Density	105
Far-Field Domain.	105
Far-Field Calculation	106
Specific Absorption Rate	107

Perfect Electric Conductor	107
Perfect Magnetic Conductor	108
Port.	109
Integration Line for Current	118
Integration Line for Voltage	118
Ground	118
Electric Potential	118
Circular Port Reference Axis	119
Diffraction Order	119
Orthogonal Polarization	121
Periodic Port Reference Point	122
Lumped Port	123
Lumped Element	126
Uniform Element	127
Electric Field	127
Magnetic Field	128
Scattering Boundary Condition	128
Reference Point	131
Impedance Boundary Condition	132
Surface Current Density	133
Surface Magnetic Current Density	134
Surface Roughness	135
Transition Boundary Condition	136
Periodic Condition	137
Magnetic Current	139
Two-Port Network.	139
Two-Port Network Port.	140
Mixed Mode S-Parameters	140
Edge Current	140
Electric Point Dipole	141
Magnetic Point Dipole	141
Line Current (Out-of-Plane)	142
Archie's Law	142
Effective Medium	143
The Electromagnetic Waves, Transient Interface	145
Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Transient Interface	146

Wave Equation, Electric	148
Initial Values	151
Drude-Lorentz Polarization	151
The Transmission Line Interface	153
Domain, Boundary, Edge, Point, and Pair Nodes for the Transmission	
Line Equation Interface	154
Transmission Line Equation	155
Initial Values	156
Absorbing Boundary	156
Incoming Wave	156
Open Circuit	157
Terminating Impedance	157
Short Circuit	158
Lumped Port	158
The Electromagnetic Waves, Time Explicit Interface	160
Domain, Boundary, and Pair Nodes for the Electromagnetic Waves,	
Time Explicit Interface	161
Wave Equations	162
Initial Values	164
Electric Current Density	165
Magnetic Current Density	165
Electric Field	165
Perfect Electric Conductor	166
Magnetic Field	166
Perfect Magnetic Conductor	166
Surface Current Density	167
Scattering Boundary Condition	167
Flux/Source	168
Background Field	168
Far-Field Domain	169
Far-Field Calculation	169
Theory for the Electromagnetic Waves Interfaces	171
Introduction to the Physics Interface Equations	171
Frequency Domain Equation	172
Time Domain Equation	177

Vector Elements	179
Eigenfrequency Calculations.	180
Gaussian Beams as Background Fields	180
Effective Material Properties in Effective Media and Mixtures	182
Effective Conductivity in Effective Media and Mixtures.	183
Effective Relative Permittivity in Effective Media and Mixtures	184
Effective Relative Permeability in Effective Media and Mixtures	185
Archie's Law Theory	186
Reference for Archie's Law	186
Theory for the Transmission Line Interface	187
Introduction to Transmission Line Theory	187
Theory for the Transmission Line Boundary Conditions	188
Theory for the Electromagnetic Waves, Time Explicit Interface	191
The Equations	191
In-Plane E Field or In-Plane H Field	195
Fluxes as Dirichlet Boundary Conditions	196
Absorbing Layers.	197

Chapter 5: AC/DC Interfaces

The Electrical Circuit Interface	200
Ground Node	202
Voltmeter	202
Ampère Meter.	203
Resistor	203
Capacitor.	203
Inductor	204
Voltage Source	204
Current Source	205
Voltage-Controlled Voltage Source	206
Voltage-Controlled Current Source.	207
Current-Controlled Voltage Source.	207
Current-Controlled Current Source	208

Subcircuit Definition	208
Subcircuit Instance	209
NPN BJT and PNP BJT.	209
n-Channel MOSFET and p-Channel MOSFET	210
Mutual Inductance	211
Transformer	211
Diode	212
External I vs. U	212
External U vs. I	213
External I-Terminal	214
SPICE Circuit Import	215
SPICE Circuit Export	215
Theory for the Electrical Circuit Interface	217
Electric Circuit Modeling and the Semiconductor Device Models.	217
Bipolar Transistors	218
MOSFET Transistors	221
Diode	224
Reference for the Electrical Circuit Interface	226

Chapter 6: Heat Transfer Interfaces

The Microwave Heating Interface	228
Electromagnetic Heating	231

Chapter 7: Glossary

Glossary of Terms	236
--------------------------	------------

Introduction

This guide describes the RF Module, an optional add-on package for COMSOL Multiphysics® with customized physics interfaces and functionality optimized for the analysis of electromagnetic waves.

This chapter introduces you to the capabilities of this module. 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.

- [About the RF Module](#)
- [Overview of the User's Guide](#)

About the RF Module

In this section:

- [What Can the RF Module Do?](#)
- [What Problems Can You Solve?](#)
- [The RF Module Physics Interface Guide](#)
- [Common Physics Interface and Feature Settings and Nodes](#)
- [Selecting the Study Type](#)
- [The RF Module Modeling Process](#)
- [Where Do I Access the Documentation and Application Libraries?](#)



The Physics Interfaces and Building a COMSOL Multiphysics Model in the *COMSOL Multiphysics Reference Manual*

What Can the RF Module Do?

The RF Module solves problems in the general field of electromagnetic waves, such as RF and microwave applications, optics, and photonics. The underlying equations for electromagnetics are automatically available in all of the physics interfaces — a feature unique to COMSOL Multiphysics. This also makes nonstandard modeling easily accessible.

The module is useful for component design in virtually all areas where you find electromagnetic waves, such as:

- Antennas
- Filters, couplers, and power dividers
- Planar circuits and passive devices
- RF interconnects and packages
- Waveguides and cavity resonators
- Frequency-selective surfaces
- Metamaterials

The physics interfaces cover the following types of electromagnetics field simulations and handle time-harmonic, time-dependent, and eigenfrequency/eigenmode problems:

- In-plane, axisymmetric, and full 3D electromagnetic wave propagation
- Full vector mode analysis in 2D and 3D

Material properties include inhomogeneous and fully anisotropic materials, media with gains or losses, and complex-valued material properties. In addition to the standard postprocessing features, the module supports direct computation of S-parameters and far-field radiation patterns. You can add ports with a wave excitation with specified power level and mode type, and add PMLs (perfectly matched layers) to simulate electromagnetic waves that propagate into an unbounded domain. For time-harmonic simulations, you can use the scattered wave or the total wave.

Using the multiphysics capabilities of COMSOL Multiphysics you can couple simulations with heat transfer, structural mechanics, fluid-flow formulations, and other physical phenomena.

This module also has interfaces for circuit modeling, a SPICE interface, and support for importing ECAD drawings.

What Problems Can You Solve?

QUASI-STATIC AND HIGH FREQUENCY MODELING

One major difference between quasi-static and high-frequency modeling is that the formulations depend on the *electrical size* of the structure. This dimensionless measure is the ratio between the largest distance between two points in the structure divided by the wavelength of the electromagnetic fields.

For simulations of structures with an electrical size in the range up to $1/10$, quasi-static formulations are suitable. The physical assumption of these situations is that wave propagation delays are small enough to be neglected. Thus, phase shifts or phase gradients in fields are caused by materials and/or conductor arrangements being inductive or capacitive rather than being caused by propagation delays.

For electrostatic, magnetostatic, and quasi-static electromagnetics, use the AC/DC Module, a COMSOL Multiphysics add-on module for low-frequency electromagnetics.

When propagation delays become important, it is necessary to use the full Maxwell equations for high-frequency electromagnetic waves. They are appropriate for structures of electrical size $1/100$ and larger. Thus, an overlapping range exists where you can use both the quasi-static and the full Maxwell physics interfaces.

Independently of the structure size, the module accommodates any case of nonlinear, inhomogeneous, or anisotropic media. It also handles materials with properties that vary as a function of time as well as frequency-dispersive materials.

The RF Module Physics Interface Guide

The physics interfaces in this module form a complete set of simulation tools for electromagnetic wave simulations. Add the physics interface and study type when starting to build a new model. You can add physics interfaces and studies to an existing model throughout the design process. In addition to the core physics interfaces included with the basic COMSOL Multiphysics license, the physics interfaces below are included with the RF Module and available in the indicated space dimension. All physics interfaces are available in 2D and 3D. In 2D there are in-plane formulations for problems with a planar symmetry as well as axisymmetric formulations for problems with a cylindrical symmetry. 2D mode analysis of waveguide cross sections with out-of-plane propagation is also supported.



In the *COMSOL Multiphysics Reference Manual*:

- [Studies and Solvers](#)
 - [The Physics Interfaces](#)
 - [Creating a New Model](#)
 - 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
 AC/DC				
Electrical Circuit		cir	Not space dependent	stationary; frequency domain; time dependent; small signal analysis, frequency domain
 Heat Transfer				
 Electromagnetic Heating				
Microwave Heating ¹		—	3D, 2D, 2D axisymmetric	frequency-stationary; frequency-transient; frequency-stationary, one-way coupled, electromagnetic heating; frequency-transient, one-way coupled, electromagnetic heating
 Radio Frequency				
Electromagnetic Waves, Frequency Domain		emw	3D, 2D, 2D axisymmetric	adaptive frequency sweep; boundary mode analysis; eigenfrequency; frequency domain, modal; mode analysis (2D and 2D axisymmetric models only); TEM boundary mode analysis
Electromagnetic Waves, Time Explicit		ewte	3D, 2D, 2D axisymmetric	time dependent; time dependent with FFT
Electromagnetic Waves, Transient		temw	3D, 2D, 2D axisymmetric	eigenfrequency; time dependent; time dependent, modal; time dependent with FFT

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Transmission Line		tl	3D, 2D, 1D	eigenfrequency; frequency domain
<p>¹ This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.</p>				

Common Physics Interface and Feature Settings and Nodes

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

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



In the *COMSOL Multiphysics Reference Manual* see [Table 2-4](#) 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.

Selecting the Study Type

To carry out different kinds of simulations for a given set of parameters in a physics interface, you can select, add, and change the Study Types at almost every stage of modeling.



[Studies and Solvers](#) in the *COMSOL Multiphysics Reference Manual*

COMPARING THE TIME DEPENDENT AND FREQUENCY DOMAIN STUDIES

When variations in time are present there are two main approaches to represent the time dependence. The most straightforward is to solve the problem by calculating the changes in the solution for each time step; that is, solving using the Time Dependent

study (available with the Electromagnetic Waves, Transient interface). However, this approach can be time consuming if small time steps are necessary for the desired accuracy. It is necessary when the inputs are transients like turn-on and turn-off sequences.

However, if the Frequency Domain study available with the Electromagnetic Waves, Frequency Domain interface is used, this allows you to efficiently simplify and assume that all variations in time occur as sinusoidal signals. Then the problem is time-harmonic and in the frequency domain. Thus you can formulate it as a stationary problem with complex-valued solutions. The complex value represents both the amplitude and the phase of the field, while the frequency is specified as a scalar model input, usually provided by the solver. This approach is useful because, combined with Fourier analysis, it applies to all periodic signals with the exception of nonlinear problems. Examples of typical frequency domain simulations are wave-propagation problems like waveguides and antennas.

For nonlinear problems you can apply a Frequency Domain study after a linearization of the problem, which assumes that the distortion of the sinusoidal signal is small.

Use a Time Dependent study when the nonlinear influence is strong, or if you are interested in the harmonic distortion of a sine signal. It can also be more efficient to use a Time Dependent study if you have a periodic input with many harmonics, like a square-shaped signal.

DEFAULT FREQUENCY

In the RF module, the Frequency Domain, the Frequency-Domain Modal, and the Eigenfrequency study step have GHz as the default frequency unit and a default frequency of 1[GHz].

The RF Module Modeling Process

The modeling process has these main steps, which (excluding the first step), correspond to the branches displayed in the Model Builder in the COMSOL Desktop environment.

- 1 Selecting the appropriate physics interface or predefined multiphysics coupling when adding a physics interface.
- 2 Defining component parameters and variables in the **Definitions** branch ().
- 3 Drawing or importing the component geometry in the **Geometry** branch ().
- 4 Assigning material properties to the geometry in the **Materials** branch ().

- 5 Setting up the model equations and boundary conditions in the physics interfaces branch.
- 6 Meshing in the **Mesh** branch ()
- 7 Setting up the study and computing the solution in the **Study** branch ()
- 8 Analyzing and visualizing the results in the **Results** branch ()



When using frequency related study steps, define the simulation frequency in the **Study** branch first. The frequency in the **Study** node will then be used for defining the physics-controlled mesh and in any frequency-based physics features.

Even after a model is defined, you can edit to input data, equations, boundary conditions, geometry — the equations and boundary conditions are still available through associative geometry — and mesh settings. You can restart the solver, for example, using the existing solution as the initial condition or initial guess. It is also easy to add another physics interface to account for a phenomenon not previously described in a model.



-
- [Building a COMSOL Multiphysics Model](#) in the *COMSOL Multiphysics Reference Manual*
 - [The RF Module Physics Interface Guide](#)
 - [Selecting the Study Type](#)
-

Where Do I Access the Documentation and Application Libraries?

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



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

THE DOCUMENTATION AND ONLINE HELP

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

Opening Topic-Based Help

The Help window is useful as it is connected to 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*.

	<p>To open the Help window:</p> <ul style="list-style-type: none">• 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.
---	--

 	<p>To open the Help window:</p> <ul style="list-style-type: none">• In the Model Builder or Physics Builder click a node or window and then press F1.• On the main toolbar, click the Help () button.• From the main menu, select Help>Help.
--	---

Opening the Documentation Window

	<p>To open the Documentation window:</p> <ul style="list-style-type: none">• Press Ctrl+F1.• From the File menu select Help>Documentation ().
---	---



To open the **Documentation** window:

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



THE APPLICATION LIBRARIES WINDOW

Each 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.



[The Application Libraries Window](#) in the *COMSOL Multiphysics Reference Manual*.

Opening the Application Libraries Window

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



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

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



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



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

CONTACTING COMSOL BY EMAIL

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

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 are 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/support/updates
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/community
Events	www.comsol.com/events
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

Overview of the User's Guide

The *RF Module User's Guide* gets you started with modeling using COMSOL Multiphysics. The information in this guide is specific to this module. Instructions 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.

TABLE OF CONTENTS, GLOSSARY, AND INDEX

To help you navigate through this guide, see the [Contents](#), [Glossary](#), and [Index](#).

MODELING WITH THE RF MODULE

The [RF Modeling](#) chapter familiarize you with the modeling procedures. A number of examples available through the Application Libraries window also illustrate the different aspects of the simulation process. Topics include [Preparing for RF Modeling](#), [Simplifying Geometries](#), and [Scattered Field Formulation](#).

RF THEORY

The [Electromagnetics Theory](#) chapter contains a review of the basic theory of electromagnetics, starting with [Maxwell's Equations](#), and the theory for some [Special Calculations](#): S-parameters, lumped port parameters, and far-field analysis. There is also a list of [Electromagnetic Quantities](#) with their SI units and symbols.

RADIO FREQUENCY

[Radio Frequency Interfaces](#) chapter describes:

- [The Electromagnetic Waves, Frequency Domain Interface](#), which analyzes frequency domain electromagnetic waves, and uses time-harmonic and eigenfrequency or eigenmode (2D only) studies, boundary mode analysis and frequency domain, modal.
- [The Electromagnetic Waves, Transient Interface](#), which supports the Time Dependent study type.

- [The Transmission Line Interface](#), which solves the time-harmonic transmission line equation for the electric potential.
- [The Electromagnetic Waves, Time Explicit Interface](#), which solves a transient wave equation for both the electric and magnetic fields.

The underlying theory is also included at the end of the chapter.

ELECTRICAL CIRCUIT

[AC/DC Interfaces](#) chapter describes [The Electrical Circuit Interface](#), which simulates the current in a conductive and capacitive material under the influence of an electric field. All three study types (Stationary, Frequency Domain, and Time Dependent) are available. The underlying theory is also included at the end of the chapter.

HEAT TRANSFER

[Heat Transfer Interfaces](#) chapter describes the Microwave Heating interface, which combines the physics features of an Electromagnetic Waves, Frequency Domain interface from the RF Module with the Heat Transfer interface. The predefined interaction adds the electromagnetic losses from the electromagnetic waves as a heat source and solves frequency domain (time-harmonic) electromagnetic waves in conjunction with stationary or transient heat transfer. This physics interface is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale (adiabatic assumption). The underlying theory is also included at the end of the chapter.

RF Modeling

The goal of this chapter is to familiarize you with the modeling procedure in the RF Module. A number of models available through the RF Module Applications Libraries also illustrate the different aspects of the simulation process.

In this chapter:

- [Preparing for RF Modeling](#)
- [Simplifying Geometries](#)
- [Periodic Boundary Conditions](#)
- [Scattered Field Formulation](#)
- [Modeling with Far-Field Calculations](#)
- [S-Parameters and Ports](#)
- [Lumped Ports with Voltage Input](#)
- [Jones Vectors and Polarization Plots](#)
- [Lossy Eigenvalue Calculations](#)
- [Connecting to Electrical Circuits](#)
- [SPICE Import and Export](#)

- [Reduced Order Modeling](#)
- [Part Libraries and Material Libraries](#)

Preparing for RF Modeling

Several modeling topics are described in this section that might not be found in ordinary textbooks on electromagnetic theory.

This section is intended to help answer questions such as:

- Which spatial dimension should I use: 3D, 2D axial symmetry, or 2D?
- Is my problem suited for time-dependent or frequency domain formulations?
- Can I use a quasi-static formulation or do I need wave propagation?
- What sources can I use to excite the fields?
- When do I need to resolve the thickness of thin shells and when can I use boundary conditions?
- What is the purpose of the model?
- What information do I want to extract from the model?

Increasing the complexity of a model to make it more accurate usually makes it more expensive to simulate. A complex model is also more difficult to manage and interpret than a simple one. Keep in mind that it can be more accurate and efficient to use several simple models instead of a single, complex one.



[The Physics Interfaces and Building a COMSOL Multiphysics Model](#) in the *COMSOL Multiphysics Reference Manual*

Simplifying Geometries

Most of the problems that are solved with COMSOL Multiphysics are three-dimensional (3D) in the real world. In many cases, it is sufficient to solve a two-dimensional (2D) problem that is close to or equivalent to the real problem. Furthermore, it is good practice to start a modeling project by building one or several 2D models before going to a 3D model. This is because 2D models are easier to modify and solve much faster. Thus, modeling mistakes are much easier to find when working in 2D. Once the 2D model is verified, you are in a much better position to build a 3D model.

In this section:

- [2D Models](#)
- [3D Models](#)
- [Using Efficient Boundary Conditions](#)
- [Applying Electromagnetic Sources](#)
- [Meshing and Solving](#)

2D Models

The text below is a guide to some of the common approximations made for 2D models. Remember that the modeling in 2D usually represents some 3D geometry under the assumption that nothing changes in the third dimension or that the field has a prescribed propagation component in the third dimension.

CARTESIAN COORDINATES

In this case a cross section is viewed in the xy -plane of the actual 3D geometry. The geometry is mathematically extended to infinity in both directions along the z -axis, assuming no variation along that axis or that the field has a prescribed wave vector component along that axis. All the total flows in and out of boundaries are per unit length along the z -axis. A simplified way of looking at this is to assume that the geometry is extruded one unit length from the cross section along the z -axis. The total flow out of each boundary is then from the face created by the extruded boundary (a boundary in 2D is a line).

There are usually two approaches that lead to a 2D cross-section view of a problem. The first approach is when it is known that there is no variation of the solution in one particular dimension.

This is shown in the model *H-Bend Waveguide 2D*, where the electric field only has one component in the z direction and is constant along that axis. The second approach is when there is a problem where the influence of the finite extension in the third dimension can be neglected.

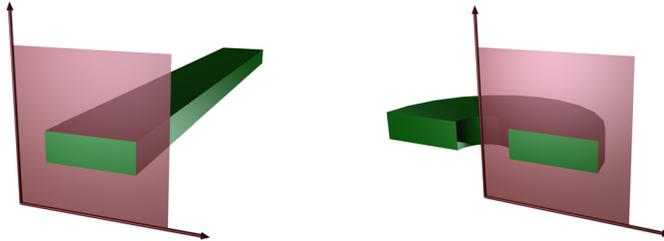


Figure 2-1: The cross sections and their real geometry for Cartesian coordinates and cylindrical coordinates (axial symmetry).



H-Bend Waveguide 2D: Application Library path **RF_Module/Transmission_Lines_and_Waveguides/h_bend_waveguide_2d**

AXIAL SYMMETRY (CYLINDRICAL COORDINATES)

If the 3D geometry can be constructed by revolving a cross section around an axis, and if no variations in any variable occur when going around the axis of revolution (or that the field has a prescribed wave vector component in the direction of revolution), then use an axisymmetric physics interface. The spatial coordinates are called r and z , where r is the radius. The flow at the boundaries is given per unit length along the third dimension. Because this dimension is a revolution all flows must be multiplied with αr , where α is the revolution angle (for example, 2π for a full turn).



Conical Antenna: Application Library path **RF_Module/Antennas/conical_antenna**



When using the axisymmetric versions, the horizontal axis represents the radial (r) direction and the vertical axis the z direction, and the geometry in the right half plane (that is, for positive r only) must be created.

POLARIZATION IN 2D

In addition to selecting 2D or 2D axisymmetry when you start building the model, the physics interfaces ([The Electromagnetic Waves, Frequency Domain Interface](#) or [The Electromagnetic Waves, Transient Interface](#)) in the Model Builder offers a choice in the Components settings section. The available choices are Out-of-plane vector, In-plane vector, and Three-component vector. This choice determines what polarizations can be handled. For example, as you are solving for the electric field, a 2D TM (out-of-plane H field) model requires choosing In-plane vector as then the electric field components are in the modeling plane.

3D Models

Although COMSOL Multiphysics fully supports arbitrary 3D geometries, it is important to simplify the problem. This is because 3D models often require more computer power, memory, and time to solve. The extra time spent on simplifying a model is probably well spent when solving it. Below are a few issues that need to be addressed before starting to implement a 3D model in this module.

- Check if it is possible to solve the problem in 2D. Given that the necessary approximations are small, the solution is more accurate in 2D, because a much denser mesh can be used.
- Look for symmetries in the geometry and model. Many problems have planes where the solution is the same on both sides of the plane. A good way to check this is to flip the geometry around the plane, for example, by turning it up-side down around the horizontal plane. Then remove the geometry below the plane if no differences are observed between the two cases regarding geometry, materials, and sources. Boundaries created by the cross section between the geometry and this plane need a symmetry boundary condition, which is available in all 3D physics interfaces.
- There are also cases when the dependence along one direction is known, and it can be replaced by an analytical function. Use this approach either to convert 3D to 2D or to convert a layer to a boundary condition.

Using Efficient Boundary Conditions

An important technique to minimize the problem size is to use efficient boundary conditions. Truncating the geometry without introducing too large errors is one of the great challenges in modeling. Below are a few suggestions of how to do this. They apply to both 2D and 3D problems.

- Many models extend to infinity or can have regions where the solution only undergoes small changes. This problem is addressed in two related steps. First, the geometry needs to be truncated in a suitable position. Second, a suitable boundary condition needs to be applied there. For static and quasi-static models, it is often possible to assume zero fields at the open boundary, provided that this is at a sufficient distance away from the sources. For radiation problems, special low-reflecting boundary conditions need to be applied. This boundary should be in the order of a few wavelengths away from any source.

A more accurate option is to use perfectly matched layers (PMLs). PMLs are layers that absorb all radiated waves with small reflections.

- Replace thin layers with boundary conditions where possible. There are several types of boundary conditions in COMSOL Multiphysics suitable for such replacements. For example, replace materials with high conductivity by the *perfect electric conductor (PEC)* boundary condition.
- Use boundary conditions for known solutions. For example, an antenna aperture can be modeled as an equivalent surface current density on a 2D face (boundary) in a 3D model.

Applying Electromagnetic Sources

Electromagnetic sources can be applied in many different ways. The typical options are boundary sources, line sources, and point sources, where point sources in 2D formulations are equivalent to line sources in 3D formulations. The way sources are imposed can have an impact on what quantities can be computed from the model. For example, a line source in an electromagnetic wave model represents a singularity and the magnetic field does not have a finite value at the position of the source. In a COMSOL Multiphysics model, the magnetic field of a line source has a finite but mesh-dependent value. In general, using volume or boundary sources is more flexible than using line sources or point sources, but the meshing of the source domains becomes more expensive.

Meshing and Solving

The finite element method approximates the solution within each element, using some elementary shape function that can be constant, linear, or of higher order. Depending on the element order in the model, a finer or coarser mesh is required to resolve the solution. In general, there are three problem-dependent factors that determine the necessary mesh resolution:

- The first is the variation in the solution due to geometrical factors. The mesh generator automatically generates a finer mesh where there is a lot of fine geometrical details. Try to remove such details if they do not influence the solution, because they produce a lot of unnecessary mesh elements.
- The second is the skin effect or the field variation due to losses. It is easy to estimate the skin depth from the conductivity, permeability, and frequency. At least two linear elements per skin depth are required to capture the variation of the fields. If the skin depth is not studied or a very accurate measure of the dissipation loss profile is not needed, replace regions with a small skin depth with a boundary condition, thereby saving elements. If it is necessary to resolve the skin depth, the boundary layer meshing technique can be a convenient way to get a dense mesh near a boundary.
- The third and last factor is the wavelength. To resolve a wave properly, it is necessary to use about 10 linear (or five 2nd order) elements per wavelength. Keep in mind that the wavelength depends on the local material properties.

SOLVERS

In most cases the solver sequence generated by COMSOL Multiphysics can be used. The choice of solver is optimized for the typical case for each physics interface and study type in this module. However, in special cases tuning the solver settings can be required. This is especially important for 3D problems because they can require a large amount of memory.



In the *COMSOL Multiphysics Reference Manual*:

- [Meshing](#)
 - [Studies and Solvers](#)
-

Periodic Boundary Conditions

The RF Module has a dedicated Periodic Condition. The periodic condition can identify simple mappings on plane source and destination boundaries of equal shape. The destination can also be rotated with respect to the source. There are three types of periodic conditions available (only the first two for transient analysis):

- *Continuity*— The tangential components of the solution variables are equal on the source and destination.
- *Antiperiodicity*— The tangential components have opposite signs.
- *Floquet periodicity*— There is a phase shift between the tangential components. The phase shift is determined by a wave vector and the distance between the source and destination. Floquet periodicity is typically used for models involving plane waves interacting with periodic structures.

Periodic boundary conditions must have compatible meshes. This can be done automatically by enabling the **Physics-control mesh** in the setting for [The Electromagnetic Waves, Frequency Domain Interface](#) or by manually setting up the correct mesh sequence



If more advanced periodic boundary conditions are required, for example, when there is a known rotation of the polarization from one boundary to another, see [Nonlocal Couplings and Coupling Operators](#) in the *COMSOL Multiphysics Reference Manual* for tools to define more general mappings between boundaries.



To learn how to use the Copy Mesh feature to ensure that the mesh on the destination boundary is identical to that on the source boundary, see *Plasmonic Wire Grating*: Application Library path **RF_Module/Tutorials/plasmonic_wire_grating**.

For an example of how to use the **Physics-controlled mesh**, see *Fresnel Equations*: Application Library path **RF_Module/Verification_Examples/fresnel_equations**.



In the *COMSOL Multiphysics Reference Manual*:

- [Periodic Condition and Destination Selection](#)
 - [Periodic Boundary Conditions](#)
-

Scattered Field Formulation

For problems where a known background field is illuminating an object in free space it is possible to use the scattered field formulation. Since the equation of the background field is known it can be entered as a model input and does not need to be computed. Starting from the frequency-domain governing equation:

$$\nabla \times (\mu^{-1} \nabla \times \mathbf{E}) - \omega^2 \epsilon_c \mathbf{E} = \mathbf{0}$$

The total electric field, \mathbf{E} , can be decomposed into two components:

$$\mathbf{E} = \mathbf{E}_{\text{total}} = \mathbf{E}_{\text{background}} + \mathbf{E}_{\text{relative}}$$

The known background field becomes a source term and the scattered field formulation thus solves for the relative electric field. A linearly polarized plane wave background field, a paraxial-approximate Gaussian beam, or a user-defined background field can be specified. When solving the scattered field formulation the total, the background, and the relative electric fields are available. The relative field is the difference between the background field and the total field. It is the relative field that contributes to the far-field calculation. For more information about the Far-Field computation, see [Far-Field Calculations Theory](#). The benefit to this approach is that if the background field is much larger in magnitude than the scattered field, the accuracy of the simulation improves if the relative field is solved for. Another advantage is that it becomes very easy to set up a perfectly matched layer surrounding the homogeneous medium modeling domain.

The drawback to this approach is that the relative field requires some careful interpretation. The relative electric field can conceptually be decomposed into:

$$\mathbf{E}_{\text{relative}} = \mathbf{E}_{\text{scattered}} + \mathbf{E}_{\text{correction}} + \mathbf{E}_{\text{cancellation}}$$

The $\mathbf{E}_{\text{scattered}}$ component is the scattered field from object. This is the field that is of interest in a scattering problem. However, the relative field may also consist of a component that represents a correction to the background field and a cancellation of the background field. The $\mathbf{E}_{\text{correction}}$ component can be nonzero when the background field does not exactly satisfy Maxwell's equations, such as when the paraxial Gaussian beam approximation is used for a tightly focused beam. For more information about the Gaussian beam theory, see [Gaussian Beams as Background Fields](#). The $\mathbf{E}_{\text{cancellation}}$ component will be nonzero and equal to $-\mathbf{E}_{\text{background}}$ wherever the total field should be zero, such as in the interior of any perfectly shielded

objects, or behind a relatively large shielding object. Note that this decomposition is conceptual only, it is only the relative field that is available.

An alternative of using the scattered-field formulation, is to use ports with the **Activate slit condition on interior port** setting enabled. Then the domain can be excited by the interior port and the outgoing field can be absorbed by perfectly matched layers. For more information about the **Port** feature and the **Activate slit condition on interior port** setting, see [Port Properties](#).

SCATTERED FIELDS SETTING

The scattered-field formulation is available for [The Electromagnetic Waves, Frequency Domain Interface](#) under the **Settings** section. The scattered field in the analysis is called the relative electric field. The total electric field is always available, and for the scattered-field formulation this is the sum of the scattered field and the incident field.



Radars Cross Section: Application Library path **RF_Module/Scattering_and_RCS/radar_cross_section**

Modeling with Far-Field Calculations

The far electromagnetic field from, for example, antennas can be calculated from the near-field solution on a boundary using far-field analysis. The antenna is located in the vicinity of the origin, while the far-field is taken at infinity but with a well-defined angular direction (θ, φ) . The far-field radiation pattern is given by evaluating the squared norm of the far-field on a sphere centered at the origin. Each coordinate on the surface of the sphere represents an angular direction.

In this section:

- [Far-Field Support in the Electromagnetic Waves, Frequency Domain Interface](#)
- [The Radiation Pattern Plots](#)



Radar Cross Section: Application Library path **RF_Module/Scattering_and_RCS/radar_cross_section**

Far-Field Support in the Electromagnetic Waves, Frequency Domain Interface

The Electromagnetic Waves, Frequency Domain interface supports far-field analysis. To define the far-field variables use the [Far-Field Calculation](#) node. Select a domain for the far-field calculation. Then select the boundaries where the algorithm integrates the near field, and enter a name for the far electric field. Also specify if symmetry planes are used in the model when calculating the far-field variable. The symmetry planes have to coincide with one of the Cartesian coordinate planes. For each of these planes it is possible to select the type of symmetry to use, which can be of either *symmetry in E (PMC)* or *symmetry in H (PEC)*. Make the choice here match the boundary condition used for the symmetry boundary. Using these settings, the parts of the geometry that are not in the model for symmetry reasons can be included in the far-field analysis.

The [Far-Field Domain](#) and the [Far-Field Calculation](#) nodes get their selections automatically, if the Perfectly Matched Layer (PML) feature has been defined before adding the [Far-Field Domain](#) feature.

For each variable name entered, the software generates functions and variables, which represent the vector components of the far electric field. The names of these variables

are constructed by appending the names of the independent variables to the name entered in the field.

For example, the name `Efar` is entered and the geometry is Cartesian with the independent variables `x`, `y`, and `z`, the generated variables get the names `Efarx`, `Efary`, and `Efarz`.

If, on the other hand, the geometry is axisymmetric with the independent variables `r`, `phi`, and `z`, the generated variables get the names `Efarr`, `Efarphi`, and `Efarz`.

In 2D, the software only generates the variables for the nonzero field components. The physics interface name also appears in front of the variable names so they can vary, but typically look something like `emw.Efarz` and so forth.

To each of the generated variables, there is a corresponding function with the same name. This function takes the vector components of the evaluated far-field direction as arguments.

The vector components also can be interpreted as a position. For example, assume that the variables `dx`, `dy`, and `dz` represent the direction in which the far electric field is evaluated.

The expression

$$Efarx(dx, dy, dz)$$

gives the value of the far electric field in this direction. To give the direction as an angle, use the expression

$$Efarx(\sin(\theta) * \cos(\phi), \sin(\theta) * \sin(\phi), \cos(\theta))$$

where the variables `theta` and `phi` are defined to represent the angular direction (θ , ϕ) in radians. The magnitude of the far field and its value in dB are also generated as the variables `normEfar` and `normdBefar`, respectively.



Far-Field Calculations Theory

The Radiation Pattern Plots

The **Radiation Pattern** plots are available with this module to plot the value of a global variable (for example, the far field norm, `normEfar` and `normdBefar`, or components of the far field variable `Efar`).

The variables are plotted for a selected number of angles on a unit circle (in 2D) or a unit sphere (in 3D). The angle interval and the number of angles can be manually specified. For 2D **Radiation Pattern** plots also the reference direction from which the angle is measured and the normal to the plane the far field is computed for can be specified. For 3D **Radiation Pattern** plots you also specify an expression for the surface color.

The main advantage with the **Radiation Pattern** plot, as compared to making a **Line Graph**, is that the unit circle/sphere that you use for defining the plot directions, is not part of your geometry for the solution. Thus, the number of plotting directions is decoupled from the discretization of the solution domain.



Default **Radiation Pattern** plots of far-field norm are automatically added to any model that uses far field calculation features.

TABLE 2-1: VARIABLES AND OPERATORS GENERATED BY FAR FIELD

	DESCRIPTION	NAME	AVAILABLE COMPONENT
Variables	Far-field norm	normEfar	2D, 2D Axisymmetric, 3D
	Far-field norm, dB	normdBefar	2D, 2D Axisymmetric, 3D
	Far-field variable, x component	Efarx	2D, 2D Axisymmetric, 3D
	Far-field variable, y component	Efary	2D, 2D Axisymmetric, 3D
	Far-field variable, z component	Efarz	2D, 2D Axisymmetric, 3D
	Effective isotropic radiated power	EIRP	3D
	Effective isotropic radiated power, dB	EIRPdB	3D
	Far-field gain	gainEfar	3D
	Far-field gain, dB	gaindBefar	3D
	Axial ratio	axialRatio	3D
	Axial ratio, dB	axialRatiODB	3D
	Bistatic radar cross section	bRCS3D	3D
	Far-field variable, phi component	Efarphi	3D

TABLE 2-1: VARIABLES AND OPERATORS GENERATED BY FAR FIELD

	DESCRIPTION	NAME	AVAILABLE COMPONENT
	Far-field variable, theta component	Efartheta	3D
	Far-field realized gain	rGainEfar	3D
	Far-field realized gain, dB	rGaindBefar	3D
	Bistatic radar cross section per unit length	bRCS2D	2D
	Maximum directivity ¹	maxD	2D Axisymmetric, 3D
	Maximum directivity, dB ¹	maxDdB	2D Axisymmetric, 3D
	Maximum gain ¹	maxGain	2D Axisymmetric, 3D
	Maximum gain, dB ¹	maxGaindB	2D Axisymmetric, 3D
	Maximum realized gain ¹	maxRGain	2D Axisymmetric, 3D
	Maximum realized gain, dB ¹	maxRGaindB	2D Axisymmetric, 3D
Functions ⁴	3D far-field norm ²	norm3DEfar	2D Axisymmetric
	3D far-field norm, dB ²	normdB3DEfar	2D Axisymmetric
	Uniform three dimensional array factor ³	af3	2D Axisymmetric, 3D
	Uniform two dimensional array factor ³	af2	2D

¹Global maximum variables require the selection for the far-field calculation feature to be spherical for 3D and circular for 2D axisymmetric components and its center to be on the origin.

²3D far-field norm functions in 2D axisymmetric geometry are available in these cases:

- Antenna models using circular port excitation with a positive azimuthal mode number
- Scattered field analysis excited by the predefined circularly polarized plane wave type

The function can be used in a 3D **Radiation Pattern** plot, where the input argument of the function must be same as the **Azimuth angle variable** in the **Evaluation** section in the settings window.

The suffix of a function name varies based on the circular port mode type, port mode number and azimuthal mode number in the physics interface. For example, when using azimuthal mode number 1 in the physics interface and transverse electric (TE)

mode with mode number 2 in the port settings, the generated operator name is norm3DEfar_TE12.

When the function is used in a radiation pattern plot under a 1D or a polar plot group, the value of input argument defines the plotting plane regardless of the normal and reference direction in the evaluation section in the settings window. For example, norm3DEfar_TE12(0) evaluates the norm of the electric far field for the TE12 mode for 0-degree azimuthal angle. This is equivalent to plotting this variable on the xz -plane. Similarly, norm3DEfar_TE12($\pi/2$) is the evaluation at 90-degree azimuthal angle, which is equivalent to plotting the variable on the yz -plane.

The 3D far-field norm, the linear superposition of the positive and negative azimuthal modes scaled by 0.5, is

$$\sqrt{|E_r \cos m\phi|^2 + |E_\phi \sin m\phi|^2 + |E_z \cos m\phi|^2},$$

where ϕ is the azimuthal angle.

³The number of input arguments depends on the dimension of model component, 2D, 2D Axisymmetric, or 3D.

TABLE 2-2: INPUT ARGUMENTS OF ARRAY FACTOR OPERATOR

ARGUMENT	DESCRIPTION	UNIT	COMPONENT
nx	Number of elements along x-axis	Dimensionless	2D, 2D Axisymmetric, 3D
ny	Number of elements along y-axis	Dimensionless	2D, 2D Axisymmetric, 3D
nz	Number of elements along z-axis	Dimensionless	2D Axisymmetric, 3D
dx	Distance between array elements along x-axis	Wavelength	2D, 2D Axisymmetric, 3D
dy	Distance between array elements along y-axis	Wavelength	2D, 2D Axisymmetric, 3D
dz	Distance between array elements along z-axis	Wavelength	2D Axisymmetric, 3D
alphax	Phase progression along x-axis	Radian	2D, 2D Axisymmetric, 3D
alphay	Phase progression along y-axis	Radian	2D, 2D Axisymmetric, 3D
alphaz	Phase progression along z-axis	Radian	2D Axisymmetric, 3D

The equation for the uniform three dimensional array factor is

$$\frac{\sin\left(\frac{n_x}{2}(2\pi d_x \sin\theta \cos\phi + \alpha_x)\right)}{\sin\left(\frac{2\pi d_x \sin\theta \cos\phi + \alpha_x}{2}\right)}, \quad \frac{\sin\left(\frac{n_y}{2}(2\pi d_y \sin\theta \cos\phi + \alpha_y)\right)}{\sin\left(\frac{2\pi d_y \sin\theta \cos\phi + \alpha_y}{2}\right)}$$

$$\frac{\sin\left(\frac{n_z}{2}(2\pi d_z \sin\theta \cos\phi + \alpha_z)\right)}{\sin\left(\frac{2\pi d_z \sin\theta \cos\phi + \alpha_z}{2}\right)},$$

where θ is the elevation angle and ϕ is the azimuthal angle.

The uniform two dimensional array factor is simpler than the three dimensional version, as the third, the z -component factor, is unity.

⁴In the postprocessing expression context menu, far-field functions are available under **Component > Definitions > Functions**.



- 2D example with a Polar Plot Group — *Radar Cross Section*: Application Library path **RF_Module/Scattering_and_RCS/radar_cross_section**
- 2D axisymmetric example with a Polar Plot Group and a 3D Plot Group — *Conical Antenna*: Application Library path **RF_Module/Antennas/conical_antenna**
- 3D example with a Polar Plot Group and a 3D Plot Group — *Radome with Double-Layered Dielectric Lens*: Application Library path **RF_Module/Antennas/radome_antenna**
- Uniform array factor operator used in a Polar Plot Group and a 3D Plot Group — *Microstrip Patch Antenna*: Application Library path **RF_Module/Antennas/microstrip_patch_antenna_inset**



- [Far-Field Support in the Electromagnetic Waves, Frequency Domain Interface](#)
- [Radiation Pattern](#) in the *COMSOL Multiphysics Reference Manual*

ANTENNA ANALYSIS USING FAR-FIELD VARIABLES

The directional properties of a radiation pattern described by variables, generated from a far-field calculation feature, help to characterize the performance of antenna devices.

Directivity from a 3D Plot

While plotting a 3D radiation pattern, the maximum directivity can be calculated by evaluating the ratio between the radiation intensity and the average value of the radiation intensity. Since the radiation intensity is a function of power, the square of the far-field norm has to be used in the **Directivity expression** in the **Radiation Pattern** settings window for the antenna directivity calculation. For other physics interfaces, such as in the Acoustics module, the expression is different.

Directivity via Global Evaluation

The maximum directivity can be computed through **Results > Derived Values > Global Evaluation**. This calculation is based on the maximum and averaged intensity values on the far-field calculation selection. It requires the selection for the far-field calculation feature to be spherical for 3D and circular for 2D axisymmetric model components, both centered at the origin.

Gain

The antenna realized gain is defined as

$$G_{\text{realized}} = \frac{4\pi U}{P_{\text{in}}} = \frac{|\text{normEfar}|^2}{60P_{\text{in}}}$$

where U is the radiation intensity, $\text{Re}(\bar{E}_{\text{far}} \times \bar{H}_{\text{far}}^*)/2 = |\text{normEfar}|^2/240\pi$ and P_{in} is the total input power.

The antenna gain is

$$G = \frac{|\text{normEfar}|^2}{60P_{\text{delivered}}}$$

where the delivered power, $P_{\text{delivered}}$ is $P_{\text{in}}(1 - |S_{11}|^2)$. The gain is available only when the S-parameter calculation is valid, that is, for the single port excitation case.

S-Parameters and Ports

In this section:

- [S-Parameters in Terms of Voltage and Electric Field](#)
- [S-Parameter Calculations](#)
- [S-Parameter Variables](#)
- [Port Sweeps/Manual Terminal Sweeps and Touchstone Export](#)

S-Parameters in Terms of Voltage and Electric Field

Scattering parameters (or S-parameters) are complex-valued, frequency dependent matrices describing the transmission and reflection of electromagnetic waves at different ports of devices like filters, antennas, waveguide transitions, and transmission lines. S-parameters originate from transmission-line theory and are defined in terms of transmitted and reflected voltage waves. All ports are assumed to be connected to matched loads/feeds, that is, there is no reflection directly at a port.

For a device with n ports, the S-parameters are

$$S = \begin{bmatrix} S_{11} & S_{12} & \cdots & S_{1n} \\ S_{21} & S_{22} & \cdots & \cdot \\ \cdot & \cdot & \cdots & \cdot \\ \cdot & \cdot & \cdots & \cdot \\ S_{n1} & \cdot & \cdots & S_{nn} \end{bmatrix}$$

where S_{11} is the voltage reflection coefficient at port 1, S_{21} is the voltage transmission coefficient from port 1 to port 2, and so on. The time average power reflection/transmission coefficients are obtained as $|S_{ij}|^2$.

Now, for high-frequency problems, voltage is not a well-defined entity, and it is necessary to define the scattering parameters in terms of the electric field.



For details on how COMSOL Multiphysics calculates the S-parameters, see [S-Parameter Calculations](#).

S-Parameter Calculations

The RF interfaces have built-in support for S-parameter calculations. Use a *Port* boundary feature for each port in the model. For connecting transmission lines and other lumped feeds, use a *lumped port* that approximates a connecting transmission lines or a voltage source with a known internal impedance. The lumped port should only be used when the port width is much smaller than the wavelength.



- For more details about lumped ports, see [Lumped Ports with Voltage Input](#).
- See [Port](#) and [Lumped Port](#) for instructions to set up a model.



For a detailed description of how to model numerical ports with a boundary mode analysis, see *Waveguide Adapter*: Application Library path **RF_Module/Transmission_Lines_and_Waveguides/waveguide_adapter**.

S-Parameter Variables

This module automatically generates variables for the S-parameters. The port names (use numbers for sweeps to work correctly) determine the variable names. If, for example, there are two ports with the numbers 1 and 2 and Port 1 is the inport, the software generates the variables **S11** and **S21**. **S11** is the S-parameter for the reflected wave and **S21** is the S-parameter for the transmitted wave. For convenience, two variables for the S-parameters on a dB scale, **S11dB** and **S21dB**, are also defined using the following relation:

$$S_{11\text{dB}} = 20\log_{10}(|S_{11}|)$$

The model and physics interface names also appear in front of the variable names so they can vary. The S-parameter variables are added to the predefined quantities in appropriate plot lists.

Port Sweeps/Manual Terminal Sweeps and Touchstone Export

The [Port Sweep Settings](#) section in the Electromagnetic Waves interface describes how to cycle through the ports, compute the entire S-matrix and export it to a Touchstone file.



H-Bend Waveguide 3D: Application Library path **RF_Module/
Transmission_Lines_and_Waveguides/h_bend_waveguide_3d**

Lumped Ports with Voltage Input

In this section:

- [About Lumped Ports](#)
- [Lumped Port Parameters](#)
- [Lumped Ports in the RF Module](#)

About Lumped Ports

The ports described in the [S-Parameters and Ports](#) section require a detailed specification of the mode, including the propagation constant and field profile. In situations with the mode being TEM, a lumped port might be a better choice. It also allows for connecting to an electrical circuit. It is not as accurate as the ordinary port in terms of calculating S-parameters, but it is easier to use. Lumped ports are used to model a connecting transmission line or as a voltage or current source applied between electrodes. For example, apply a lumped port directly to a printed circuit board or to the transmission line feed of a device. The lumped port must be applied between two metallic objects separated by a distance much smaller than the wavelength, that is a local *quasi-static approximation* must be justified. This is because the concept of port or gap voltage breaks down unless the gap is much smaller than the local wavelength.

A lumped port specified as an input port calculates the impedance, Z_{port} , and S_{11} S-parameter for that port. The parameters are directly given by the relations

$$Z_{\text{port}} = \frac{V_{\text{port}}}{I_{\text{port}}}$$
$$S_{11} = \frac{V_{\text{port}} - V_{\text{in}}}{V_{\text{in}}}$$

where V_{port} is the extracted voltage for the port given by the electric field line integral between the terminals averaged over the entire port. The current I_{port} is the averaged total current over all cross sections parallel to the terminals. Ports not specified as input ports only return the extracted voltage and current.



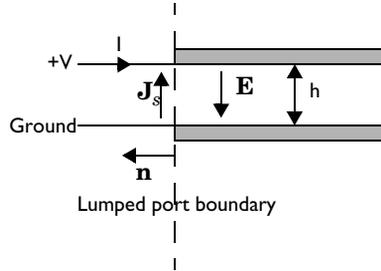
[Lumped Port Parameters](#)

Lumped Port Parameters

In transmission line theory voltages and currents are dealt with rather than electric and magnetic fields, so the lumped port provides an interface between them. The requirement on a lumped port is that the feed point must be similar to a transmission line feed, so its gap must be much less than the wavelength. It is then possible to define the electric field from the voltage as

$$V = \int_h \mathbf{E} \cdot d\mathbf{l} = \int_h (\mathbf{E} \cdot \mathbf{a}_h) dl$$

where h is a line between the terminals at the beginning of the transmission line, and the integration is going from positive (phase) V to ground. The current is positive going into the terminal at positive V .



The transmission line current can be represented with a surface current at the lumped port boundary directed opposite to the electric field.

The impedance of a transmission line is defined as

$$Z = \frac{V}{I}$$

and in analogy to this an equivalent surface impedance is defined at the lumped port boundary

$$\eta = \frac{\mathbf{E} \cdot \mathbf{a}_h}{\mathbf{J}_s \cdot (-\mathbf{a}_h)}$$

To calculate the surface current density from the current, integrate along the width, w , of the transmission line

$$I = \int_w (\mathbf{n} \times \mathbf{J}_s) \cdot d\mathbf{l} = -\int_w (\mathbf{J}_s \cdot \mathbf{a}_h) dl$$

where the integration is taken in the direction of $\mathbf{a}_h \times \mathbf{n}$. This gives the following relation between the transmission line impedance and the surface impedance

$$Z = \frac{V}{I} = \frac{\int (\mathbf{E} \cdot \mathbf{a}_h) dl}{-\int_w (\mathbf{J}_s \cdot \mathbf{a}_h) dl} = \eta \frac{\int (\mathbf{E} \cdot \mathbf{a}_h) dl}{\int_w (\mathbf{E} \cdot \mathbf{a}_h) dl} \approx \eta \frac{h}{w} \Rightarrow$$

$$\eta = Z \frac{w}{h}$$

where the last approximation assumed that the electric field is constant over the integrations. A similar relationship can be derived for coaxial cables

$$\eta = Z \frac{2\pi}{\ln \frac{b}{a}}$$

The transfer equations above are used in an impedance type boundary condition, relating surface current density to tangential electric field via the surface impedance.

$$\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) + \frac{1}{\eta} \mathbf{n} \times (\mathbf{E} \times \mathbf{n}) = 2 \frac{1}{\eta} \mathbf{n} \times (\mathbf{E}_0 \times \mathbf{n})$$

where \mathbf{E} is the total field and \mathbf{E}_0 the incident field, corresponding to the total voltage, V , and incident voltage, V_0 , at the port.



When using the lumped port as a circuit port, the port voltage is fed as input to the circuit and the current computed by the circuit is applied as a uniform current density, that is as a surface current condition. Thus, an open (unconnected) circuit port is just a continuity condition.

Lumped Ports in the RF Module

Not all models can use lumped ports due to the polarization of the fields and how sources are specified. For the physics interfaces and study types that support the lumped port, the Lumped Port is available as a boundary feature. See [Lumped Port](#) for instructions to set up this feature.

LUMPED PORT VARIABLES

Each lumped port generates variables that are accessible to the user. Apart from the S-parameter, a lumped port condition also generates the following variables.

NAME	DESCRIPTION
Vport	Extracted port voltage
Iport	Port current
Zport	Port impedance

For example, a lumped port with port number 1, defined in the first geometry, for the Electromagnetic Waves interface with the tag `emw`, defines the port impedance variable `emw.Zport_1`.



RF Coil: Application Library path **RF_Module/Passive_Devices/rf_coil**

Jones Vectors and Polarization Plots

In this section:

- [Jones Vectors for Polarization Analysis](#)
- [Polarization Plots](#)
- [Jones Vector Variables](#)

Jones Vectors for Polarization Analysis

Periodic ports and Diffraction order ports launch and absorb plane waves propagating in homogeneous domains (adjacent the port boundary). For a plane wave propagating with the wave vector \mathbf{k} , the polarization must be orthogonal to the wave vector. For each wave vector, there are two possible orthogonal polarizations. We can select one such set of orthogonal polarizations by first defining the out-of-plane polarization as the field in the direction $\mathbf{e}_1 = \mathbf{k} \times \mathbf{n}$, where \mathbf{n} is the normal direction to the port. Then the in-plane polarization direction is defined as $\mathbf{e}_2 = \mathbf{e}_1 \times \mathbf{k}$.

Assuming now that \mathbf{e}_1 and \mathbf{e}_2 are normalized base vectors, the electric field can be expanded as

$$\mathbf{E} = E_1 \mathbf{e}_1 + E_2 \mathbf{e}_2,$$

where E_1 and E_2 are the elements of the Jones vector

$$\begin{pmatrix} E_1 \\ E_2 \end{pmatrix}.$$



In COMSOL, the Jones vector elements are complex numbers and the Jones vectors are not normalized.

If the Jones vector elements have the same phase or a π phase difference, the Jones vector represents a linear polarization state. A phase difference of $\pm\pi/2$ between the

two Jones vector elements defines a circular polarization state. For other phase differences, the Jones vector represents elliptic polarization states.



For more details about the different types of periodic ports, see

- [Periodic](#)
- [Diffraction Order](#)
- [Orthogonal Polarization](#)

Polarization Plots

Default Polarization plots are automatically generated for [Periodic](#) ports in 3D and in 2D, if the **Electric field components for** setting in the **Components** section for the physics interface is set to **Three-component vector**. The Polarization plot includes polarization ellipses for each diffraction order. The polarization ellipse line graphs are generated by plotting the in-plane Jones vector element versus the out-of-plane Jones vector element for a phase change of 2π .



For an example model including a Polarization plot, see *Hexagonal Grating*: Application Library path **RF_Module/Tutorials/hexagonal_grating**.

Jones Vector Variables

This module automatically generates variables for the Jones vector elements. As for [Polarization Plots](#), the variables are created for [Periodic](#) ports in 3D and in 2D, if the **Electric field components for** setting in the **Components** section for the physics interface is set to **Three-component vector**. The variables are available for post-processing as global variables, with names based on what port boundary the variable is applicable for, the polarization direction, and the mode number. The context above is encoded in a variable name of the form `tag.JXYYY_Z`, where

- `tag` is the physics interface tag, for example `emw`.
- `X` is `R` if the port is located on the same side as the exciting port or otherwise `T`.
- `YYY` is `OOP` or `IP` for the out-of-plane and in-plane modes, respectively.
- `Z` is the mode number that in 2D is provided as a single signed integer and in 3D as two signed integers separated by an underscore. The signed integers use the prefix `p` for positive values and `n` for negative values.

Thus, the variable `emw.JR00P_0_n1` represents the Jones vector element for the out-of-plane mode in 3D with mode numbers $m = 0$ and $n = -1$ for a port located on the same boundary as the exciting Periodic port.

There are also variables for the norm of the Jones vector, named as `tag.normJX_Z`, where X and Z represents the boundary location and mode number, respectively. Thus, the variable `emw.normJR_0_n1` represents the Jones vector norm in 3D with mode numbers $m = 0$ and $n = -1$ for a port located on the same boundary as the exciting Periodic port.

The base vectors are available for evaluation on the port boundaries, with variable names like `tag.eJXYYY[xyz]_Z`, where `tag`, X , YYY , and Z have the same meanings as for the variables discussed above and `[xyz]` is one of the Cartesian components. Thus, `emw.eJTIPx_p1`, is the x -component of the in-plane base vector for a 2D port, with mode number $m = +1$, that is not located on the same boundary as the exciting port.

Lossy Eigenvalue Calculations

In mode analysis and eigenfrequency analysis, it is usually the primary goal to find a propagation constant or an eigenfrequency. These quantities are often real-valued although it is not necessary. If the analysis involves some lossy part, like a nonzero conductivity or an open boundary, the eigenvalue is complex. In such situations, the eigenvalue is interpreted as two parts (1) the propagation constant or eigenfrequency and (2) the damping in space and time.

In this section:

- [Eigenfrequency Analysis](#)
- [Mode Analysis](#)



Lossy Circular Waveguide: Application Library path **RF_Module/Transmission_Lines_and_Waveguides/lossy_circular_waveguide**

Eigenfrequency Analysis

The eigenfrequency analysis solves for the eigenfrequency of a model. The time-harmonic representation of the fields is more general and includes a complex parameter in the phase

$$\mathbf{E}(\mathbf{r}, t) = \text{Re}(\tilde{\mathbf{E}}(\mathbf{r}_T)e^{j\omega t}) = \text{Re}(\tilde{\mathbf{E}}(\mathbf{r})e^{-\lambda t})$$

where the eigenvalue, $(-\lambda) = -\delta + j\omega$, has an imaginary part representing the eigenfrequency, and a real part responsible for the damping. It is often more common to use the *quality factor* or *Q-factor*, which is derived from the eigenfrequency and damping

$$Q_{\text{fact}} = \frac{\omega}{2|\delta|}$$

VARIABLES AFFECTED BY EIGENFREQUENCY ANALYSIS

The following list shows the variables that the eigenfrequency analysis affects:

NAME	EXPRESSION	CAN BE COMPLEX	DESCRIPTION
omega	imag(-lambda)	No	Angular frequency
damp	real(lambda)	No	Damping in time
Qfact	0.5*omega/abs(damp)	No	Quality factor
nu	omega/(2*pi)	No	Frequency

NONLINEAR EIGENFREQUENCY PROBLEMS

For some combinations of formulation, material parameters, and boundary conditions, the eigenfrequency problem can be nonlinear, which means that the eigenvalue enters the equations in another form than the expected second-order polynomial form. The following table lists those combinations:

SOLVE FOR	CRITERION	BOUNDARY CONDITION
E	Nonzero conductivity	Impedance boundary condition
E	Nonzero conductivity at adjacent domain	Scattering boundary condition
E	Analytical ports	Port boundary condition

These situations may require special treatment, especially since it can lead to “singular matrix” or “undefined value” messages if not treated correctly. Under normal circumstances, the automatically generated solver settings should handle the cases described in the table above. However, the following discussion provide some background to the problem of defining the eigenvalue linearization point. The complication is not only the nonlinearity itself, it is also the way it enters the equations. For example the impedance boundary conditions with nonzero boundary conductivity has the term

$$-(-\lambda) \frac{\sqrt{\epsilon_0 \mu_0} \sqrt{\mu_{rbnd}}}{\sqrt{\epsilon_{rbnd} + \frac{\sigma_{bnd}}{(-\lambda)\epsilon_0}}} (\mathbf{n} \times (\mathbf{n} \times \mathbf{H}))$$

where $(-\lambda) = -\delta + j\omega$. When the solver starts to solve the eigenfrequency problem it linearizes the entire formulation with respect to the eigenvalue around a certain linearization point. By default this linearization point is set to the value provided to the **Search for eigenvalues around** field, for the three cases listed in the table above. Normally, this should be a good value for the linearization point. For instance, for the

impedance boundary condition, this avoids setting the eigenvalue λ to zero in the denominator in the equation above. For other cases than those listed in the table above, the default linearization point is zero.

If the default values for the linearization point is not suitable for your particular problem, you can manually provide a “good” linearization point for the eigenvalue solver. Do this in the **Eigenvalue** node (not the Eigenfrequency node) under the **Solver Sequence** node in the **Study** branch of the Model Builder. A solver sequence can be generated first. In the **Linearization Point** section, select the **Transform point** check box and enter a suitable value in the **Point** field. For example, if it is known that the eigenfrequency is close to 1 GHz, enter the eigenvalue 1 [GHz] in the field.

In many cases it is enough to specify a good linearization point and then solve the problem once. If a more accurate eigenvalue is needed, an iterative scheme is necessary:

- 1 Specify that the eigenvalue solver only searches for one eigenvalue. Do this either for an existing solver sequence in the **Eigenvalue** node or, before generating a solver sequence, in the **Eigenfrequency** node.
- 2 Solve the problem with a “good” linearization point. As the eigenvalue shifts, use the same value with the real part removed from the eigenvalue or, equivalently, use the real part of the eigenfrequency.
- 3 Extract the eigenvalue from the solution and update the linearization point and the shift.
- 4 Repeat until the eigenvalue does not change more than a desired tolerance.



- For a list of the studies available by physics interface, see [The RF Module Physics Interface Guide](#)
- [Studies and Solvers](#) in the *COMSOL Multiphysics Reference Manual*

Mode Analysis

In mode analysis and boundary mode analysis COMSOL Multiphysics solves for the propagation constant. The time-harmonic representation is almost the same as for the eigenfrequency analysis, but with a known propagation in the out-of-plane direction

$$\mathbf{E}(\mathbf{r}, t) = \text{Re}(\tilde{\mathbf{E}}(\mathbf{r}_T)e^{j\omega t - j\beta z}) = \text{Re}(\tilde{\mathbf{E}}(\mathbf{r})e^{j\omega t - \alpha z})$$

The spatial parameter, $\alpha = \delta_z + j\beta = -\lambda$, can have a real part and an imaginary part. The propagation constant is equal to the imaginary part, and the real part, δ_z , represents the damping along the propagation direction.

VARIABLES INFLUENCED BY MODE ANALYSIS

The following table lists the variables that are influenced by the mode analysis:

NAME	EXPRESSION	CAN BE COMPLEX	DESCRIPTION
beta	<code>imag(-lambda)</code>	No	Propagation constant
dampz	<code>real(-lambda)</code>	No	Attenuation constant
dampzdB	<code>20*log10(exp(1))*dampz</code>	No	Attenuation per meter in dB
neff	<code>j*lambda/k0</code>	Yes	Effective mode index



For an example of Boundary Mode Analysis, see the model *Polarized Circular Ports*: Application Library path **RF_Module/Tutorials/polarized_circular_ports**.



- For a list of the studies available by physics interface, see [The RF Module Physics Interface Guide](#)
- [Studies and Solvers](#) in the *COMSOL Multiphysics Reference Manual*

Connecting to Electrical Circuits

In this section:

- [About Connecting Electrical Circuits to Physics Interfaces](#)
- [Connecting Electrical Circuits Using Predefined Couplings](#)
- [Connecting Electrical Circuits by User-Defined Couplings](#)
- [Solving](#)
- [Postprocessing](#)



Connecting a 3D Electromagnetic Wave Model to an Electrical Circuit: Application Library path **RF_Module/Transmission_Lines_and_Waveguides/coaxial_cable_circuit**

About Connecting Electrical Circuits to Physics Interfaces

This section describes the various ways electrical circuits can be connected to other physics interfaces in COMSOL Multiphysics. If you are not familiar with circuit modeling, it is recommended that you review the [Theory for the Electrical Circuit Interface](#).

In general electrical circuits connect to other physics interfaces via one or more of three special circuit features:

- [External I vs. U](#)
- [External U vs. I](#)
- [External I-Terminal](#)

These features either accept a voltage measurement from the connecting noncircuit physics interface and return a current from a or the Electrical Circuit interface or the other way around.



The “External” features are considered “ideal” current or voltage sources by the Electrical Circuit interface. Hence, you cannot connect them directly in parallel (voltage sources) or in series (current sources) with other ideal sources. This results in the error message *The DAE is structurally inconsistent*. A workaround is to provide a suitable parallel or series resistor, which can be tuned to minimize its influence on the results.

Connecting Electrical Circuits Using Predefined Couplings

In addition to these circuit features, interfaces in the AC/DC Module, RF Module, MEMS Module, Plasma Module, and Semiconductor Module (the modules that include the Electrical Circuit interface) also contain features that provide couplings to the Electrical Circuit interface by accepting a voltage or a current from one of the specific circuit features ([External I vs. U](#), [External U vs. I](#), and [External I-Terminal](#)).

This coupling is typically activated when:

- A choice is made in the **Settings** window for the noncircuit physics interface feature, which then announces (that is, includes) the coupling to the Electrical Circuit interface. Its voltage or current is then included to make it visible to the connecting circuit feature.
- A voltage or current that has been announced (that is, included) is selected in a feature node’s **Settings** window.

These circuit connections are supported in Lumped Ports.

Connecting Electrical Circuits by User-Defined Couplings

A more general way to connect a physics interface to the Electrical Circuit interface is to:

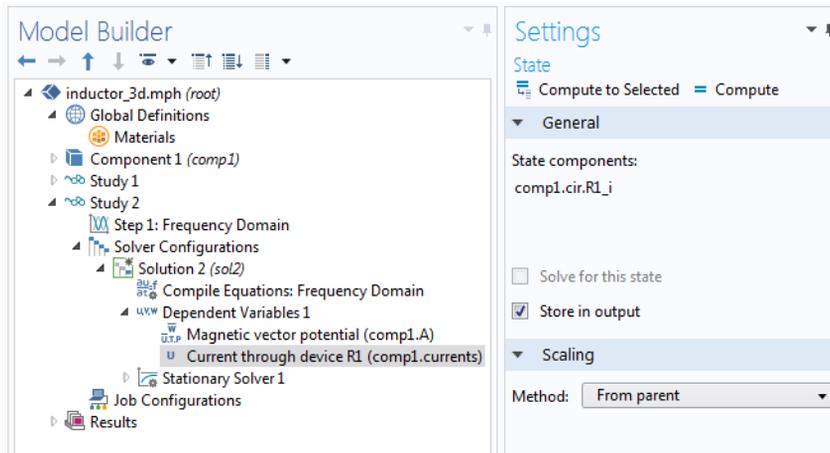
- Apply the voltage or current from the connecting “External” circuit feature as an excitation in the noncircuit physics interface.

- Define your own voltage or current measurement in the noncircuit physics interface using variables, coupling operators and so forth.
- In the **Settings** window for the Electrical Circuit interface feature, selecting the User-defined option and entering the name of the variable or expression using coupling operators defined in the previous step.

DETERMINING A CURRENT OR VOLTAGE VARIABLE NAME

To determine a current or voltage variable name, look at the **Dependent Variables** node under the **Study** node. To do this:

- 1 In the **Model Builder**, right-click the **Study** node and select **Show Default Solver**.
- 2 Expand the **Solver>Dependent Variables** node and click the state node, in this example, **Current through device R1 (comp1.currents)**. The variable name is shown in the **Settings** window for **State**.



Typically, voltage variables are named `cir.Xn_v` and current variables `cir.Xn_i`, where n is the “External” device number — 1, 2, and so on.

Solving



Some modeling errors lead to the error message *The DAE is structurally inconsistent* being displayed when solving. This error typically occurs from having an open current loop, from connecting voltage sources in parallel, or connecting current sources in series.

In this respect, the predefined coupling features are also treated as (ideal) voltage or current sources. The remedy is to close current loops and to connect resistors in series with voltage sources or in parallel with current sources.

Postprocessing

The Electrical Circuits interface, unlike most of the other physics interfaces, solves for a relatively large number of global dependent variables (such as voltages and currents), instead of solving for a few space-varying fields (such as temperature or displacement). For this reason, the Electrical Circuit interface does not provide default plots when computing a study.

The physics interface defines a number of variables that can be used in postprocessing. All variables defined by the Electrical Circuit interface are of a global scope, and can be evaluated in a **Global Evaluation** node (under **Derived Values**). In addition, the time evolution or dependency on a parameter can be plotted in a **Global** plot (under a **ID Plot Group** node).

The physics interface defines a Node voltage variable for each electrical node in the circuit, with name `cir.v_name`, where `cir` is the physics interface Label and `<name>` is the node Name. For each two-pin component, the physics interface also defines variables containing the voltage across it and the current flowing through it.



In the *COMSOL Multiphysics Reference Manual*:

- [Derived Values, Evaluation Groups, and Tables and Global Evaluation](#)
 - [Plot Groups and Plots and Global](#)
-

SPICE Import and Export

SPICE Import

The circuit definition in COMSOL Multiphysics adheres to the SPICE format developed at the University of California, Berkeley (Ref. 1). SPICE netlists can be imported and the corresponding circuit nodes are generated in the COMSOL Multiphysics model. Most circuit simulators can export to this format or some version of it.

The Electrical Circuit interface supports the following device models:

TABLE 2-3: SUPPORTED SPICE DEVICE MODELS

STATEMENT	DEVICE MODEL
R	Resistor
C	Capacitor
L	Inductor
V	Voltage Source
I	Current Source
E	Voltage-Controlled Voltage Source
F	Current-Controlled Current Source
G	Voltage-Controlled Current Source
H	Current-Controlled Voltage Source
D	Diode
Q	NPN BJT and PNP BJT
M	n-Channel MOSFET and p-Channel MOSFET
X	Subcircuit Instance

Statements corresponding to multiple devices are resolved by parsing the associated `.model` statement. The physics interface also supports the `.subckt` statement, which is represented in COMSOL by a [Subcircuit Definition](#) node, and the `.include` statement. SPICE commands are interpreted case-insensitively. The statement defining each device is also interpreted as the **Device name**.

According to SPICE specification, the first line in the netlist file is assumed to be the title of the netlist and it is ignored by the parser.

SPICE Export

The **SPICE Export** functionality creates a SPICE netlist file containing a description of the circuit represented by the physics interface. This functionality can be accessed from the physics interface context menu (right-click the physics interface node and select **Export SPICE Netlist**). After specifying a file name, the circuit is exported and messages from the export process display in the **Messages** window. During the export process, a series of operations are performed:

- In order to avoid conflicts, each component must be identified by a unique **Device name**. If one or more components have the same device name, the export operation fails and an error message is displayed. All characters in a **Device name** that are not letters, digits or underscores are replaced by underscores.
- According to the SPICE specification, each circuit must have a node with name 0, which is assumed to be the only ground node. When exporting a circuit, any node with name 0 that is not connected to a **Ground** component is exported with a different node name. All nodes that are connected to a Ground components are exported as a merged node with name 0. The Messages window shows a log message if these operations are performed, showing the name of the renamed or merged nodes.
- All characters in node names that are not letters, digits or underscores are replaced by underscores.
- Some components (most notably, the **External** components used to couple to other physics interfaces) cannot be exported to a SPICE netlist. These components are ignored during the export process, and a message is shown in the Messages window. Note that this can change the exported circuit, since some components are then missing.
- Subcircuit definitions are added as `.subckt` statements in the netlist. Semiconductor devices (such as MOSFETs, BJTs, and diodes) are exported as a SPICE device with a corresponding `.model` statement.

The title of the exported netlist file is the model's filename, and the time, date, and version of COMSOL Multiphysics is added as a comment in the netlist file.

Reference

1. <http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE/>

Reduced Order Modeling

When designing bandpass-filter type high-Q devices in the frequency domain, it may be necessary to apply many frequency samples to describe the passband accurately. The reduced order modeling technique can help accelerate the modeling of such devices as a fine frequency resolution can be used for a modest simulation time.

Two simulation methods: the asymptotic waveform evaluation (AWE) and frequency-domain modal methods, both are designed to help overcome the conventional issue of a longer simulation time when using a very fine frequency resolution or running a very wideband simulation. The AWE is efficient when it comes to describing smooth frequency responses with a single resonance or no resonance at all. The frequency-domain modal method, meanwhile, is useful for quickly analyzing multistage filters or filters of a high number of elements that have multiple resonances in a target passband.

In this section:

- [Adaptive Frequency Sweep Using Asymptotic Waveform Evaluation \(AWE\) Method](#)
- [Frequency Domain, Modal Method](#)



[Adaptive Frequency Sweep and Frequency Domain, Modal in the COMSOL Multiphysics Reference Manual](#)

Adaptive Frequency Sweep Using Asymptotic Waveform Evaluation (AWE) Method

The AWE method is very useful when simulating resonant circuits, especially single-resonance bandpass-filter type devices with many frequency points. When using the **Adaptive Frequency Sweep** study, the simulation time with a much finer frequency resolution can be almost the same as a coarse resolution regular **Frequency Domain** simulation.

CHOOSING APPROPRIATE ASYMPTOTIC WAVEFORM EVALUATION (AWE) EXPRESSIONS

The simulation time may vary depending on the user input for the **AWE expressions**. Any model variable works as an AWE expression, so long as it has a smooth curve shape like a Gaussian pulse as a function of frequency. The absolute value of S_{21} , `abs(comp1.emw.S21)`, often works as the input for the AWE expression in the case of two-port bandpass filters. For one-port devices like antennas, the absolute value of S_{11} is a good choice. If the frequency response of the AWE expression contains an infinite gradient — the case for the S_{11} value of an antenna with excellent impedance matching at a single frequency point — the simulation will take longer to complete because it requires many data points to describe the sharp dip. When the loss in a one-port device is negligible, an alternative expression such as `sqrt(1 - abs(comp1.emw.S11)^2)` may work more efficiently than using `abs(comp1.emw.S11)` directly.

DATA MANAGEMENT

With a very fine frequency step simulation, the solutions contain a lot of data. As a result, the model file size will increase tremendously when it is saved. By selecting the **Store fields in output** check box in the **Values of Dependent Variables** section of the **Frequency Domain** study step settings, it is possible to define for what part of the model the computed solution should be saved. When only S-parameters are of interest, it is not necessary to store all of the field solutions. Instead, only store the field on the selections for the port and lumped port boundaries, as those will be used for the S-parameter calculations.

In the **Values of Dependent Variables** section, change the selection in the **Store fields in output** combo box from **All** to **For selections** and then add the explicit selections that include the port and lumped port boundaries. The explicit selection can be easily created from the port and lumped port feature by clicking **Create Selection** icon in the **Boundary Selection** settings once the selection is specified.

AWE CONFIGURATION IN FREQUENCY DOMAIN STUDY STEP SETTINGS

The solver performs fast adaptive frequency sweeping using an AWE method. To trigger the AWE in a **Frequency Domain** study, the following steps are required:

- Expand **Study Extensions** section in **Frequency Domain** study step settings.

- Click the **Use asymptotic waveform evaluation** check box in the **Study Extensions** section.
- Specify the user input for the **AWE expressions**.



Evanescent Mode Cylindrical Cavity Filter: Application Library path **RF_Module/Filters/cylindrical_cavity_filter_evanescent**

RF Coil: Application Library path **RF_Module/Passive_Devices/rf_coil**

Frequency Domain, Modal Method

Bandpass-frequency responses of a passive circuit often result from a combination of multiple resonances. **Eigenfrequency** analysis is used for capturing the resonance frequencies of a device. In a subsequent step, the information from the **Eigenfrequency** solutions can be reused in a **Frequency Domain, Modal** study to generate a solution for the driven problem.

EIGENFREQUENCY STUDY STEP SETTINGS

To perform a **Frequency Domain, Modal** analysis, it is necessary to configure the **Eigenfrequency** study step properly. As the output of the **Eigenfrequency** study may include unphysical results (so called spurious modes), appropriate settings help refine the **Eigenfrequency** study results.

- Set **Eigenfrequency search method** to **Manual**
- Adjust **Desired number of eigenfrequencies** if necessary.
- Set **Search for eigenfrequencies around** to the estimate of the lowest passband frequency.
- Set **Eigenfrequency search method around shift** to **Larger real part**.

DATA MANAGEMENT

The **Store fields in output** check box in the **Values of Dependent Variables** section can be applied to the **Frequency Domain, Modal** study — if you are interested only in

S-parameters. By storing solutions only on port or lumped port boundaries, the saved model file size will decrease a lot.



Note that the phase of the computed S-parameters in the **Frequency Domain, Modal** study can be different from that of the regular frequency sweep model due to that all relevant eigenmodes might not be included in the simulation. It is recommended to perform an initial eigenfrequency investigation, to find all relevant eigenfrequencies contributing to the frequency response. If not all relevant eigenfrequencies are included in the simulation, the results are compatible only for phase-independent S-parameter values, such as dB-scaled, absolute value, reflectivity, or transmittivity.



Cascaded Rectangular Cavity Filter: Application Library path **RF_Module/Filters/cascaded_cavity_filter**

Coupled-Line Bandpass Filter: Application Library path **RF_Module/Filters/coupled_line_filter**

Coplanar Waveguide Bandpass Filter: Application Library path **RF_Module/Filters/cpw_bandpass_filter**

Part Libraries

In RF simulations, it is often necessary to set up identical geometry sequences, such as connectors, multiple times. This can be conveniently accomplished using the Part Library for the RF Module.

The Part Library contains complex shapes, frequently required for RF simulations, including the following parts:

- Connectors
- Surface mount devices
- Waveguides

The parts are built from partially parameterized sequences of geometry instructions. For example, you can load the `connector_sma_flange2` part into a model and then specify the outer and inner radii of the coaxial structure. RF parts usually include selections for the conductive boundary that make it easy to apply PEC boundary conditions while setting up the physics.



The following RF tutorials use the Part Library to create their geometry sequences:

- *Branch-Line Coupler*: Application Library path **RF_Module/Couplers_and_Power_Dividers/branch_line_coupler**
- *SMA Connectorized Wilkinson Power Divider*: Application Library path **RF_Module/Couplers_and_Power_Dividers/wilkinson_power_divider**



[Part Libraries](#) in the *COMSOL Multiphysics Reference Manual*.

Material Libraries

The RF Module features a Material Library with material properties for substrate materials to assist in modeling RF, microwave, and millimeter-wave circuit boards.

The RF Material Library contains the material property data from following companies' products:

ROGERS CORPORATION

RO4000® Laminates, RT/duroid® Laminates, RO3000® Laminates, XT/duroid® Copper Clad Laminates, TMM® Laminates, TC™ Series Laminates, Kappa™ 438 Laminates, CuClad® & IsoClad® Series Laminate, DiClad® Series Laminates, CLTE Series™ Materials, AD Series™ Laminates, and Bondply/Prepreg

See <https://www.rogerscorp.com> for more information.

ISOLA GROUP

185HR, 254, 370 TURBO®, 370HR, A11, Astra® MT77, DE104, ED130UV, FR402, FR406, FR406N, FR408, FR408HR, G200, I-Speed®, I-Tera® MT40, I-Tera® MT40 (RF/MW), IS300MD, IS400, IS400HR, IS410, IS415, IS420, IS680, IS680 AG, IS680 AG -348, P25N, P95/P25, P96/P26, Tachyon® 100G, TerraGreen®, and TerraGreen® (RF/MW)

See <https://www.isola-group.com> for more information.

PREMIX GROUP

PREPERM® Standard Grades, PREPERM® Radome Grades, PREPERM® High-temperature Grades, PREPERM® Flexible Grades, and PREPERM® H-series Grades

The PREPERM® materials are Premix group products, see <https://www.premixgroup.com> for more information.

Electromagnetics Theory

This chapter contains a review of the basic theory of electromagnetics, starting with Maxwell's equations, and the theory for some special calculations: S-parameters, lumped port parameters, and far-field analysis. There is also a list of electromagnetic quantities with their SI units and symbols.

In this chapter:

- [Maxwell's Equations](#)
- [Special Calculations](#)
- [Electromagnetic Quantities](#)

See also:

- [Theory for the Electromagnetic Waves Interfaces](#)
- [Theory for the Electrical Circuit Interface](#)
- [Theory for Heat Transfer](#) in the *COMSOL Multiphysics Reference Manual*

Maxwell's Equations

In this section:

- [Introduction to Maxwell's Equations](#)
- [Constitutive Relations](#)
- [Potentials](#)
- [Electromagnetic Energy](#)
- [Material Properties](#)
- [About the RF Materials Database](#)
- [Boundary and Interface Conditions](#)
- [Phasors](#)

Introduction to Maxwell's Equations

Electromagnetic analysis on a macroscopic level involves solving *Maxwell's equations* subject to certain boundary conditions. Maxwell's equations are a set of equations, written in differential or integral form, stating the relationships between the fundamental electromagnetic quantities. These quantities are the:

- Electric field intensity **E**
- Electric displacement or electric flux density **D**
- Magnetic field intensity **H**
- Magnetic flux density **B**
- Current density **J**
- Electric charge density ρ

The equations can be formulated in differential or integral form. The differential form are presented here, because it leads to differential equations that the finite element method can handle. For general time-varying fields, Maxwell's equations can be written as

$$\begin{aligned}\nabla \times \mathbf{H} &= \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \\ \nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t} \\ \nabla \cdot \mathbf{D} &= \rho \\ \nabla \cdot \mathbf{B} &= 0\end{aligned}$$

The first two equations are also referred to as *Maxwell-Ampère's law* and *Faraday's law*, respectively. Equation three and four are two forms of *Gauss' law*, the electric and magnetic form, respectively.

Another fundamental equation is the *equation of continuity*, which can be written as

$$\nabla \cdot \mathbf{J} = -\frac{\partial \rho}{\partial t}$$

Out of the five equations mentioned, only three are independent. The first two combined with either the electric form of Gauss' law or the equation of continuity form such an independent system.

Constitutive Relations

To obtain a closed system, the *constitutive relations* describing the macroscopic properties of the medium, are included. They are given as

$$\begin{aligned}\mathbf{D} &= \epsilon_0 \mathbf{E} + \mathbf{P} \\ \mathbf{B} &= \mu_0 (\mathbf{H} + \mathbf{M}) \\ \mathbf{J} &= \sigma \mathbf{E}\end{aligned}$$

Here ϵ_0 is the permittivity of vacuum, μ_0 is the permeability of vacuum, and σ the electrical conductivity. In the SI system, the permeability of a vacuum is chosen to be $4\pi \cdot 10^{-7}$ H/m. The velocity of an electromagnetic wave in a vacuum is given as c_0 and the permittivity of a vacuum is derived from the relation

$$\epsilon_0 = \frac{1}{2} \frac{1}{c_0 \mu_0} = 8.854 \cdot 10^{-12} \text{ F/m} \approx \frac{1}{36\pi} \cdot 10^{-9} \text{ F/m}$$

The *electric polarization vector* \mathbf{P} describes how the material is polarized when an electric field \mathbf{E} is present. It can be interpreted as the volume density of *electric dipole* moments. \mathbf{P} is generally a function of \mathbf{E} . Some materials can have a nonzero \mathbf{P} also when there is no electric field present.

The *magnetization vector* \mathbf{M} similarly describes how the material is magnetized when a magnetic field \mathbf{H} is present. It can be interpreted as the volume density of *magnetic dipole* moments. \mathbf{M} is generally a function of \mathbf{H} . Permanent magnets, however, have a nonzero \mathbf{M} also when there is no magnetic field present.

For linear materials, the polarization is directly proportional to the electric field, $\mathbf{P} = \epsilon_0 \chi_e \mathbf{E}$, where χ_e is the *electric susceptibility*. Similarly in linear materials, the magnetization is directly proportional to the magnetic field, $\mathbf{M} = \chi_m \mathbf{H}$, where χ_m is the magnetic susceptibility. For such materials, the constitutive relations can be written

$$\begin{aligned}\mathbf{D} &= \epsilon_0(1 + \chi_e)\mathbf{E} = \epsilon_0 \epsilon_r \mathbf{E} = \epsilon \mathbf{E} \\ \mathbf{B} &= \mu_0(1 + \chi_m)\mathbf{H} = \mu_0 \mu_r \mathbf{H} = \mu \mathbf{H}\end{aligned}$$

The parameter ϵ_r is the relative permittivity and μ_r is the relative permeability of the material. These are usually scalar properties but they can, for a general anisotropic material, be 3-by-3 tensors. The properties ϵ and μ (without subscripts) are the permittivity and permeability of the material.

GENERALIZED CONSTITUTIVE RELATIONS

Generalized forms of the constitutive relations are well suited for modeling nonlinear materials. The relation used for the electric fields is

$$\mathbf{D} = \epsilon_0 \epsilon_r \mathbf{E} + \mathbf{D}_r$$

The field \mathbf{D}_r is the *remanent displacement*, which is the displacement when no electric field is present.

Similarly, a generalized form of the constitutive relation for the magnetic field is

$$\mathbf{B} = \mu_0 \mu_r \mathbf{H} + \mathbf{B}_r$$

where \mathbf{B}_r is the *remanent magnetic flux density*, which is the magnetic flux density when no magnetic field is present.

The relation defining the current density is generalized by introducing an externally generated current \mathbf{J}^e . The resulting constitutive relation is

$$\mathbf{J} = \sigma \mathbf{E} + \mathbf{J}^e$$

Potentials

Under certain circumstances it can be helpful to formulate the problems in terms of the electric scalar potential V and the magnetic vector potential \mathbf{A} . They are given by the equalities

$$\begin{aligned}\mathbf{B} &= \nabla \times \mathbf{A} \\ \mathbf{E} &= -\nabla V - \frac{\partial \mathbf{A}}{\partial t}\end{aligned}$$

The defining equation for the magnetic vector potential is a direct consequence of the magnetic Gauss' law. The electric potential results from Faraday's law.

Electromagnetic Energy

The electric and magnetic energies are defined as

$$\begin{aligned}W_e &= \int_V \left(\int_0^D \mathbf{E} \cdot d\mathbf{D} \right) dV = \int_V \left(\int_0^T \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} dt \right) dV \\ W_m &= \int_V \left(\int_0^B \mathbf{H} \cdot d\mathbf{B} \right) dV = \int_V \left(\int_0^T \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dt \right) dV\end{aligned}$$

The time derivatives of these expressions are the electric and magnetic power

$$\begin{aligned}P_e &= \int_V \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} dV \\ P_m &= \int_V \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dV\end{aligned}$$

These quantities are related to the resistive and radiative energy, or energy loss, through Poynting's theorem ([Ref. 3](#))

$$-\int_V \left(\mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} + \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} \right) dV = \int_V \mathbf{J} \cdot \mathbf{E} dV + \oint_S (\mathbf{E} \times \mathbf{H}) \cdot \mathbf{n} dS$$

where V is the computation domain and S is the closed boundary of V .

The first term on the right-hand side represents the resistive losses,

$$P_h = \int_V \mathbf{J} \cdot \mathbf{E} dV$$

which result in heat dissipation in the material. (The current density \mathbf{J} in this expression is the one appearing in Maxwell-Ampère's law.)

The second term on the right-hand side of Poynting's theorem represents the radiative losses,

$$P_r = \oint_S (\mathbf{E} \times \mathbf{H}) \cdot \mathbf{n} dS$$

The quantity $\mathbf{S} = \mathbf{E} \times \mathbf{H}$ is called the Poynting vector.

Under the assumption the material is linear and isotropic, it holds that

$$\begin{aligned} \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} &= \epsilon \mathbf{E} \cdot \frac{\partial \mathbf{E}}{\partial t} = \frac{\partial}{\partial t} \left(\frac{1}{2} \epsilon \mathbf{E} \cdot \mathbf{E} \right) \\ \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} &= \frac{1}{\mu} \mathbf{B} \cdot \frac{\partial \mathbf{B}}{\partial t} = \frac{\partial}{\partial t} \left(\frac{1}{2\mu} \mathbf{B} \cdot \mathbf{B} \right) \end{aligned}$$

By interchanging the order of differentiation and integration (justified by the fact that the volume is constant and the assumption that the fields are continuous in time), this equation results:

$$-\frac{\partial}{\partial t} \int_V \left(\frac{1}{2} \epsilon \mathbf{E} \cdot \mathbf{E} + \frac{1}{2\mu} \mathbf{B} \cdot \mathbf{B} \right) dV = \int_V \mathbf{J} \cdot \mathbf{E} dV + \oint_S (\mathbf{E} \times \mathbf{H}) \cdot \mathbf{n} dS$$

The integrand of the left-hand side is the total electromagnetic energy density

$$w = w_e + w_m = \frac{1}{2} \epsilon \mathbf{E} \cdot \mathbf{E} + \frac{1}{2\mu} \mathbf{B} \cdot \mathbf{B}$$

Material Properties

Until now, there has only been a formal introduction of the constitutive relations. These seemingly simple relations can be quite complicated at times. There are four main groups of materials where they require some considerations. A given material can belong to one or more of these groups. The groups are:

- Inhomogeneous materials
- Anisotropic materials
- Nonlinear materials
- Dispersive materials

The least complicated of the groups above is that of the inhomogeneous materials. An inhomogeneous medium is one where the constitutive parameters vary with the space coordinates, so that different field properties prevail at different parts of the material structure.

For anisotropic materials, the field relations at any point are different for different directions of propagation. This means that a 3-by-3 tensor is required to properly define the constitutive relations. If this tensor is symmetric, the material is often referred to as *reciprocal*. In these cases, the coordinate system can be rotated in such a way that a diagonal matrix is obtained. If two of the diagonal entries are equal, the material is *uniaxially anisotropic*. If none of the elements have the same value, the material is *biaxially anisotropic* (Ref. 2). An example where anisotropic parameters are used is for the permittivity in crystals (Ref. 2).

Nonlinearity is the effect of variations in permittivity or permeability with the intensity of the electromagnetic field. This also includes hysteresis effects, where not only the current field intensities influence the physical properties of the material, but also the history of the field distribution.

Finally, dispersion describes changes in the velocity of the wave with wavelength. In the frequency domain, dispersion is expressed by a frequency dependence in the constitutive laws.

MATERIAL PROPERTIES AND THE MATERIAL BROWSER

All interfaces in the RF Module support the use of the COMSOL Multiphysics material database libraries. The electromagnetic material properties that can be stored in the materials database are:

- The electrical conductivity
- The relative permittivity
- The relative permeability
- The refractive index

The physics-specific domain material properties are by default taken from the material specification. The material properties are inputs to material laws or constitutive relations that are defined on the feature level below the physics interface node in the model tree. There is one editable default domain feature (wave equation) that initially represents a linear isotropic material. Domains with different material laws are specified by adding additional features. Some of the domain parameters can either be a scalar or a matrix (tensor) depending on whether the material is isotropic or anisotropic.

In a similar way, boundary, edge, and point settings are specified by adding the corresponding features. A certain feature might require one or several fields to be specified, while others generate the conditions without user-specified fields.

About the RF Materials Database



For detailed information about [Materials](#) and [Modeling Anisotropic Materials](#), including the [RF Materials Database](#), see the *COMSOL Multiphysics Reference Manual*.

The RF database contains a number of substrate materials that assist in modeling RF components. Dielectric properties, like the real part of the relative permittivity and the loss tangent, including the frequency dependence, are included.

Boundary and Interface Conditions

To get a full description of an electromagnetic problem, specify boundary conditions at material interfaces and physical boundaries. At interfaces between two media, the boundary conditions can be expressed mathematically as

$$\begin{aligned}\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) &= \mathbf{0} \\ \mathbf{n}_2 \cdot (\mathbf{D}_1 - \mathbf{D}_2) &= \rho_s \\ \mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) &= \mathbf{J}_s \\ \mathbf{n}_2 \cdot (\mathbf{B}_1 - \mathbf{B}_2) &= 0\end{aligned}$$

where ρ_s and \mathbf{J}_s denote surface charge density and *surface current density*, respectively, and \mathbf{n}_2 is the outward normal from medium 2. Of these four conditions, only two are independent. One of the first and the fourth equations, together with one of the second and third equations, form a set of two independent conditions.

A consequence of the above is the interface condition for the current density,

$$\mathbf{n}_2 \cdot (\mathbf{J}_1 - \mathbf{J}_2) = -\frac{\partial \rho_s}{\partial t}$$

INTERFACE BETWEEN A DIELECTRIC AND A PERFECT CONDUCTOR

A perfect conductor has infinite electrical conductivity and thus no internal electric field. Otherwise, it would produce an infinite current density according to the third

fundamental constitutive relation. At an interface between a dielectric and a perfect conductor, the boundary conditions for the \mathbf{E} and \mathbf{D} fields are simplified. If, say, subscript 1 corresponds to the perfect conductor, then $\mathbf{D}_1 = \mathbf{0}$ and $\mathbf{E}_1 = \mathbf{0}$ in the relations above. For the general time-varying case, it holds that $\mathbf{B}_1 = \mathbf{0}$ and $\mathbf{H}_1 = \mathbf{0}$ as well (as a consequence of Maxwell's equations). What remains is the following set of boundary conditions for time-varying fields in the dielectric medium.

$$\begin{aligned} -\mathbf{n}_2 \times \mathbf{E}_2 &= \mathbf{0} \\ -\mathbf{n}_2 \times \mathbf{H}_2 &= \mathbf{J}_s \\ -\mathbf{n}_2 \cdot \mathbf{D}_2 &= \rho_s \\ -\mathbf{n}_2 \cdot \mathbf{B}_2 &= \mathbf{0} \end{aligned}$$

Phasors

Whenever a problem is time-harmonic the fields can be written in the form

$$\mathbf{E}(\mathbf{r}, t) = \hat{\mathbf{E}}(\mathbf{r}) \cos(\omega t + \phi)$$

Instead of using a cosine function for the time dependence, it is more convenient to use an exponential function, by writing the field as

$$\mathbf{E}(\mathbf{r}, t) = \hat{\mathbf{E}}(\mathbf{r}) \cos(\omega t + \phi) = \text{Re}(\hat{\mathbf{E}}(\mathbf{r}) e^{j\phi} e^{j\omega t}) = \text{Re}(\tilde{\mathbf{E}}(\mathbf{r}) e^{j\omega t})$$

The field $\tilde{\mathbf{E}}(\mathbf{r})$ is a *phasor* (phase vector), which contains amplitude and phase information of the field but is independent of t . One thing that makes the use of phasors suitable is that a time derivative corresponds to a multiplication by $j\omega$,

$$\frac{\partial \mathbf{E}}{\partial t} = \text{Re}(j\omega \tilde{\mathbf{E}}(\mathbf{r}) e^{j\omega t})$$

This means that an equation for the phasor can be derived from a time-dependent equation by replacing the time derivatives by a factor $j\omega$. All time-harmonic equations in this module are expressed as equations for the phasors. (The tilde is dropped from the variable denoting the phasor.)



When looking at the solution of a time-harmonic equation, it is important to remember that the field that has been calculated is a phasor and not a physical field.

For example, all plot functions visualize

$$\text{Re}(\tilde{\mathbf{E}}(\mathbf{r}))$$

by default, which is \mathbf{E} at time $t = 0$. To obtain the solution at a given time, specify a phase factor when evaluating and visualizing the results.

Special Calculations

In this section:

- [S-Parameter Calculations](#)
- [Far-Field Calculations Theory](#)
- [References](#)



Lumped Ports with Voltage Input

S-Parameter Calculations

For high-frequency problems, voltage is not a well-defined entity, and it is necessary to define the scattering parameters (S-parameter) in terms of the electric field. To convert an electric field pattern on a port to a scalar complex number corresponding to the voltage in transmission line theory an eigenmode expansion of the electromagnetic fields on the ports needs to be performed. Assume that an eigenmode analysis has been performed on the ports 1, 2, 3, ... and that the electric field patterns $\mathbf{E}_1, \mathbf{E}_2, \mathbf{E}_3, \dots$ of the fundamental modes on these ports are known. Further, assume that the fields are normalized with respect to the integral of the power flow across each port cross section, respectively. This normalization is frequency dependent unless TEM modes are being dealt with. The port excitation is applied using the fundamental eigenmode, the mode with subscript 1. The computed electric field \mathbf{E}_c on the port consists of the excitation plus the reflected field. That is, on the port boundary where there is an incident wave, the computed field can be expanded in terms of the mode fields as

$$\mathbf{E}_c = \mathbf{E}_1 + \sum_{i=1} S_{i1} \mathbf{E}_i,$$

whereas on all other port boundaries, the computed field is given by

$$\mathbf{E}_c = \sum_{i=1} S_{i1} \mathbf{E}_i$$

The S-parameter for the mode with index k is then given by multiplying with the conjugate of the mode field for mode k and integrating over the port boundary. Since the mode fields for the different modes are orthogonal, the following relations are obtained for the S-parameters

$$S_{11} = \frac{\int_{\text{port 1}} ((\mathbf{E}_c - \mathbf{E}_1) \cdot \mathbf{E}_1^*) dA_1}{\int_{\text{port 1}} (\mathbf{E}_1 \cdot \mathbf{E}_1^*) dA_1}$$

$$S_{21} = \frac{\int_{\text{port 2}} (\mathbf{E}_c \cdot \mathbf{E}_2^*) dA_2}{\int_{\text{port 2}} (\mathbf{E}_2 \cdot \mathbf{E}_2^*) dA_2}$$

$$S_{31} = \frac{\int_{\text{port 3}} (\mathbf{E}_c \cdot \mathbf{E}_3^*) dA_3}{\int_{\text{port 3}} (\mathbf{E}_3 \cdot \mathbf{E}_3^*) dA_3}$$

and so on. To get S_{22} and S_{12} , excite port number 2 in the same way.

POWER FLOW NORMALIZATION

The fields \mathbf{E}_1 , \mathbf{E}_2 , \mathbf{E}_3 , and so on, should be normalized such that they represent the same power flow through the respective ports. The power flow is given by the time-average Poynting vector,

$$\mathbf{S}_{\text{av}} = \frac{1}{2} \text{Re}(\mathbf{E} \times \mathbf{H}^*)$$

The amount of power flowing out of a port is given by the normal component of the Poynting vector,

$$\mathbf{n} \cdot \mathbf{S}_{\text{av}} = \mathbf{n} \cdot \frac{1}{2} \text{Re}(\mathbf{E} \times \mathbf{H}^*)$$

Below the *cutoff frequency* the power flow is zero, which implies that it is not possible to normalize the field with respect to the power flow below the cutoff frequency. But in this region the S-parameters are trivial and do not need to be calculated.

In the following subsections the power flow is expressed directly in terms of the electric field for TE, TM, and TEM waves.

TE Waves

For TE waves it holds that

$$\mathbf{E} = -Z_{\text{TE}}(\mathbf{n} \times \mathbf{H})$$

where Z_{TE} is the wave impedance

$$Z_{\text{TE}} = \frac{\omega\mu}{\beta}$$

ω is the angular frequency of the wave, μ the permeability, and β the propagation constant. The power flow then becomes

$$\mathbf{n} \cdot \mathbf{S}_{\text{av}} = \frac{1}{2} \mathbf{n} \cdot \text{Re}(\mathbf{E} \times \mathbf{H}^*) = -\frac{1}{2} \text{Re}(\mathbf{E} \cdot (\mathbf{n} \times \mathbf{H}^*)) = \frac{1}{2Z_{\text{TE}}} |\mathbf{E}|^2$$

TM Waves

For TM waves it holds that

$$\mathbf{H} = \frac{1}{Z_{\text{TM}}}(\mathbf{n} \times \mathbf{E})$$

where Z_{TM} is the wave impedance

$$Z_{\text{TM}} = \frac{\beta}{\omega\epsilon}$$

and ϵ is the permittivity. The power flow then becomes

$$\begin{aligned} \mathbf{n} \cdot \mathbf{S}_{\text{av}} &= \frac{1}{2} \mathbf{n} \cdot \text{Re}(\mathbf{E} \times \mathbf{H}^*) = \frac{1}{2Z_{\text{TM}}} (\mathbf{n} \cdot \text{Re}(\mathbf{E} \times (\mathbf{n} \times \mathbf{E}^*))) \\ &= \frac{1}{2Z_{\text{TM}}} |\mathbf{n} \times \mathbf{E}|^2 \end{aligned}$$

TEM Waves

For TEM waves it holds that

$$\mathbf{H} = \frac{1}{Z_{\text{TEM}}}(\mathbf{n} \times \mathbf{E})$$

where Z_{TEM} is the wave impedance

$$Z_{\text{TEM}} = \sqrt{\frac{\mu}{\epsilon}}$$

The power flow then becomes

$$\mathbf{n} \cdot \mathbf{S}_{\text{av}} = \frac{1}{2} \mathbf{n} \cdot \text{Re}(\mathbf{E} \times \mathbf{H}^*) = \frac{1}{2Z_{\text{TEM}}} |\mathbf{n} \times \mathbf{E}|^2 = \frac{1}{2Z_{\text{TEM}}} |\mathbf{E}|^2$$

where the last equality holds because the electric field is tangential to the port.

Far-Field Calculations Theory

The far field from, for example, antennas can be calculated from the near field using the Stratton-Chu formula. In 3D, this is:

$$\mathbf{E}_p = \frac{jk}{4\pi} \mathbf{r}_0 \times \int [\mathbf{n} \times \mathbf{E} - \eta \mathbf{r}_0 \times (\mathbf{n} \times \mathbf{H})] \exp(jk \mathbf{r} \cdot \mathbf{r}_0) dS$$

and in 2D it looks slightly different:

$$\mathbf{E}_p = \sqrt{\lambda} \frac{jk}{4\pi} \mathbf{r}_0 \times \int [\mathbf{n} \times \mathbf{E} - \eta \mathbf{r}_0 \times (\mathbf{n} \times \mathbf{H})] \exp(jk \mathbf{r} \cdot \mathbf{r}_0) dS$$

In both cases the integration is performed on a closed boundary. In the scattered field formulation, where the total electric field is the sum of the background field and the scattered field, the far-field only gets contributions from the scattered field, since the contributions from the background field cancel out when integrated over all parts of the closed boundary.

For scattering problems, the far field in COMSOL Multiphysics is identical to what in physics is known as the “scattering amplitude”.

The antenna is located in the vicinity of the origin, while the far-field point p is taken at infinity but with a well-defined angular position (θ, φ) .

In the above formulas,

- \mathbf{E} and \mathbf{H} are the fields on the “aperture” — the surface S enclosing the antenna.
- \mathbf{r}_0 is the unit vector pointing from the origin to the field point p . If the field points lie on a spherical surface S' , \mathbf{r}_0 is the unit normal to S' .
- \mathbf{n} is the unit normal to the surface S .
- η is the impedance:

$$\eta = \sqrt{\mu/\epsilon}$$

- k is the wave number.

- λ is the wavelength.
- \mathbf{r} is the radius vector (not a unit vector) of the surface S .
- \mathbf{E}_p is the calculated far field in the direction from the origin towards point p .

Thus the unit vector \mathbf{r}_0 can be interpreted as the direction defined by the angular position (θ, φ) and \mathbf{E}_p is the far field in this direction.

Because the far field is calculated in free space, the magnetic field at the far-field point is given by

$$\mathbf{H}_p = \frac{\mathbf{r}_0 \times \mathbf{E}_p}{\eta_0}$$

The Poynting vector gives the power flow of the far field:

$$\mathbf{r}_0 \cdot \mathbf{S} = \mathbf{r}_0 \cdot \text{Re}(\mathbf{E}_p \times \mathbf{H}_p^*) \sim |\mathbf{E}_p|^2$$

Thus the relative far-field radiation pattern is given by plotting $|\mathbf{E}_p|^2$.

References

1. D.K. Cheng, *Field and Wave Electromagnetics*, 2nd ed., Addison-Wesley, 1991.
2. Jianming Jin, *The Finite Element Method in Electromagnetics*, 2nd ed., Wiley-IEEE Press, 2002.
3. A. Kovetz, *The Principles of Electromagnetic Theory*, Cambridge University Press, 1990.
4. R.K. Wangsness, *Electromagnetic Fields*, 2nd ed., John Wiley & Sons, 1986.

Electromagnetic Quantities

Table 3-1 shows the symbol and SI unit for most of the physical quantities that are included with this module.

TABLE 3-1: ELECTROMAGNETIC QUANTITIES

QUANTITY	SYMBOL	UNIT	ABBREVIATION
Angular frequency	ω	radian/second	rad/s
Attenuation constant	α	meter ⁻¹	m ⁻¹
Capacitance	C	farad	F
Charge	q	coulomb	C
Charge density (surface)	ρ_s	coulomb/meter ²	C/m ²
Charge density (volume)	ρ	coulomb/meter ³	C/m ³
Current	I	ampere	A
Current density (surface)	\mathbf{J}_s	ampere/meter	A/m
Current density (volume)	\mathbf{J}	ampere/meter ²	A/m ²
Electric displacement	\mathbf{D}	coulomb/meter ²	C/m ²
Electric field	\mathbf{E}	volt/meter	V/m
Electric potential	V	volt	V
Electric susceptibility	χ_e	(dimensionless)	–
Electrical conductivity	σ	siemens/meter	S/m
Energy density	W	joule/meter ³	J/m ³
Force	\mathbf{F}	newton	N
Frequency	ν	hertz	Hz
Impedance	Z, η	ohm	Ω
Inductance	L	henry	H
Magnetic field	\mathbf{H}	ampere/meter	A/m
Magnetic flux	Φ	weber	Wb
Magnetic flux density	\mathbf{B}	tesla	T
Magnetic potential (scalar)	V_m	ampere	A
Magnetic potential (vector)	\mathbf{A}	weber/meter	Wb/m
Magnetic susceptibility	χ_m	(dimensionless)	–
Magnetization	\mathbf{M}	ampere/meter	A/m

TABLE 3-1: ELECTROMAGNETIC QUANTITIES

QUANTITY	SYMBOL	UNIT	ABBREVIATION
Permeability	μ	henry/meter	H/m
Permittivity	ϵ	farad/meter	F/m
Polarization	\mathbf{P}	coulomb/meter ²	C/m ²
Poynting vector	\mathbf{S}	watt/meter ²	W/m ²
Propagation constant	β	radian/meter	rad/m
Reactance	X	ohm	Ω
Relative permeability	μ_r	(dimensionless)	–
Relative permittivity	ϵ_r	(dimensionless)	–
Resistance	R	ohm	W
Resistive loss	Q	watt/meter ³	W/m ³
Torque	T	newton-meter	Nm
Velocity	\mathbf{v}	meter/second	m/s
Wavelength	λ	meter	m
Wave number	k	radian/meter	rad/m

Radio Frequency Interfaces

This chapter discusses the physics interfaces found under the **Radio Frequency** branch () .

In this chapter:

- [The Electromagnetic Waves, Frequency Domain Interface](#)
- [The Electromagnetic Waves, Transient Interface](#)
- [The Transmission Line Interface](#)
- [The Electromagnetic Waves, Time Explicit Interface](#)
- [Theory for the Electromagnetic Waves Interfaces](#)
- [Theory for the Transmission Line Interface](#)
- [Theory for the Electromagnetic Waves, Time Explicit Interface](#)

The Electromagnetic Waves, Frequency Domain Interface

The **Electromagnetic Waves, Frequency Domain (emw)** interface () , found under the **Radio Frequency** branch () when adding a physics interface, is used to solve for time-harmonic electromagnetic field distributions.

For this physics interface, the maximum mesh element size should be limited to a fraction of the wavelength. The domain size that can be simulated thus scales with the amount of available computer memory and the wavelength. The physics interface supports the Frequency Domain, Eigenfrequency, Mode Analysis, and Boundary Mode Analysis study types. The Frequency Domain study type is used for source driven simulations for a single frequency or a sequence of frequencies. The Eigenfrequency study type is used to find resonance frequencies and their associated eigenmodes in resonant cavities.

This physics interface solves the time-harmonic wave equation for the electric field.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Wave Equation, Electric, Perfect Electric Conductor**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Electromagnetic Waves, Frequency Domain** to select physics features from the context menu.

The Mode analysis study type is applicable only for 2D and 2D axisymmetric cross-sections of waveguides and transmission lines where it is used to find allowed propagating modes. Boundary mode analysis is used for the same purpose in 2D, 2D axisymmetry, and 3D and applies to boundaries representing waveguide ports.

Physics-Controlled Mesh

The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). In the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the check box in the **Use** column on the same row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

When the **Use** check box for the physics interface is selected, this invokes a parameter for the maximum mesh element size in free space. The physics-controlled mesh automatically scales the maximum mesh element size as the wavelength changes in

different dielectric and magnetic regions. If the model is configured by any periodic conditions, identical meshes are generated on each pair of periodic boundaries. Perfectly matched layers are built with a structured mesh, specifically, a swept mesh in 3D and a mapped mesh in 2D.

When the **Use** check box is selected for the physics interface, in the section for the physics interface below the table, choose one of the four options for the **Maximum mesh element size control parameter** — **From study** (the default), **User defined**, **Frequency**, or **Wavelength**. When **From study** is selected, $1/5$ of the vacuum wavelength from the highest frequency defined in study step is used for the maximum mesh element size. For the option **User defined**, enter a suitable **Maximum element size in free space**. For example, $1/5$ of the vacuum wavelength or smaller. When **Frequency** is selected, enter the highest frequency intended to be used during the simulation. The maximum mesh element size in free space is $1/5$ of the vacuum wavelength for the entered frequency. For the **Wavelength** option, enter the smallest vacuum wavelength intended to be used during the simulation. The maximum mesh element size in free space is $1/5$ of the entered wavelength.

When **Resolve wave in lossy media** is selected, the outer boundaries of lossy media domains are meshed with a maximum mesh element size in free space given by the minimum value of half a skin depth and $1/5$ of the vacuum wavelength.

The maximum mesh element size in dielectric media is that in free space divided by the square root of the product of the relative permittivity and permeability.



In the *COMSOL Multiphysics Reference Manual* see the [Physics-Controlled Mesh](#) section for more information about how to define the physics-controlled mesh.

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 `emw`.

SETTINGS

From the **Formulation** list, select whether to solve for the **Full field** (the default) or the **Scattered field**.

For **Scattered field** select a **Background wave type** according to the following table:

TABLE 4-1: BACKGROUND WAVE TYPE BASED ON COMPONENT DIMENSION

COMPONENT	BACKGROUND WAVE TYPE
2D	User defined (default), Gaussian beam
2D Axisymmetric	User defined (default), Circularly polarized plane wave
3D	User defined (default), Gaussian beam, Linearly polarized plane wave

User Defined

Enter the component expressions for the **Background electric field** \mathbf{E}_b (SI unit: V/m). The entered expressions must be differentiable.



Notice that expressions including coupling operators are not differentiable and cannot be used as background fields.

Gaussian Beam

For **Gaussian beam** select the **Gaussian beam type** — **Paraxial approximation** (the default) or **Plane wave expansion**.

When selecting **Paraxial approximation**, the Gaussian beam background field is a solution to the paraxial wave equation, which is an approximation to the Helmholtz equation solved for by the **Electromagnetic Waves, Frequency Domain (emw)** interface. The approximation is valid for Gaussian beams that have a beam radius that is much larger than the wavelength. Since the paraxial Gaussian beam background field is an approximation to the Helmholtz equation, for tightly focused beams, you can get a nonzero scattered field solution, even if you do not have any scatterers. The option **Plane wave expansion** means that the electric field for the Gaussian beam is approximated by an expansion of the electric field into a number of plane waves. Since each plane wave is a solution to Helmholtz equation, the plane wave expansion of the electric field is also a solution to Helmholtz equation. Thus, this option can be used also for tightly focused Gaussian beams.

If the beam spot radius is smaller than the wavelength, evanescent plane waves need to be included in the expansion. The evanescent waves decay exponentially in the propagation direction, why it only makes sense to model such tightly focused beams if the focal plane coincides with the input boundary. If the focal plane is located inside

the modeled domain, the field can be dominated by the exponentially decaying evanescent waves. Those waves can have a very high field strength before the focal plane even though they only provide a small contribution to the field at the focal plane.

For **Plane wave expansion** select **Wave vector distribution type** — **Automatic** (the default) or **User defined**. For **Automatic** also check **Allow evanescent waves**, to include evanescent waves in the plane wave expansion. For **User defined** also enter values for the **Wave vector count** $N_{\mathbf{k}}$ (the default value is 13) and **Maximum transverse wave number** $k_{t,\max}$ (SI unit: rad/m, default value is $(2 \cdot (\sqrt{2 \cdot \log(10)})) / \text{emw.w0}$). Use an odd number for the **Wave vector count** $N_{\mathbf{k}}$ to make sure that a wave vector pointing in the main propagation direction is included in the plane-wave expansion. The **Wave vector count** $N_{\mathbf{k}}$ specifies the number of wave vectors that will be included per transverse dimension. So for 3D the total number of wave vectors will be $N_{\mathbf{k}} \cdot N_{\mathbf{k}} \cdot N_{\mathbf{k}}$.



Evanescent waves are included in the plane wave expansion if the **Maximum transverse wave number** $k_{t,\max}$ is larger than the specified **Wave number** k . When the **Wave vector distribution type** is set to **Automatic**, evanescent waves are included in the expansion if the **Allow evanescent waves** check box is selected.

A plane wave expansion with a finite number of plane waves included will make the field periodic in the plane orthogonal to the main propagation direction. If the separation between the transverse wave vector components, given by $2k_{t,\max} / (N_{\mathbf{k}} - 1)$, is too small, replicas of the Gaussian beam background field can appear. To avoid that, increase the value for the **Wave vector count** $N_{\mathbf{k}}$.

The number of plane waves included in the expansion can be quite large, especially for 3D. For instance, using the default settings, $2 \cdot 13 \cdot 13 = 338$ plane waves will be included (the factor 2 accounts for the two possible polarizations for each wave vector). Thus, initializing the plane-wave expansion for the Gaussian beam background field can take some time in 3D.

For more information about the Gaussian beam theory, see [Gaussian Beams as Background Fields](#).

- Select a **Beam orientation**: **Along the x-axis** (the default), **Along the y-axis**, or for 3D components, **Along the z-axis**.
- Enter a **Beam radius** w_0 (SI unit: m). The default is $20\pi / \text{emw.k}_0$ m (10 vacuum wavelengths).
- Enter a **Focal plane along the axis** p_0 (SI unit: m). The default is 0 m.

- Enter the component expressions for the **Background electric field amplitude, Gaussian beam \mathbf{E}_{bg0}** (SI unit: V/m).
- Enter a **Wave number k** (SI unit: rad/m). The default is emw.k_0 rad/m. The wave number must evaluate to a value that is the same for all the domains the scattered field is applied to. Setting the **Wave number k** to a positive value, means that the wave is propagating in the positive x -, y -, or z -axis direction, whereas setting the **Wave number k** to a negative value means that the wave is propagating in the negative x -, y -, or z -axis direction.

Linearly Polarized Plane Wave

The initial background wave is predefined as $\mathbf{E}_0 = \exp(-jk_x x)\mathbf{z}$. This field is transformed by three successive rotations along the roll, pitch, and yaw angles, in that order. For a graphic representation of the initial background field and the definition of the three rotations; compare with [Figure 4-1](#) below.

- Enter an **Electric field amplitude E_0** (SI unit: V/m). The default is 1 V/m.
- Enter a **Roll angle** (SI unit: rad), which is a right-handed rotation with respect to the $+x$ direction. The default is 0 rad, corresponding to polarization along the $+z$ direction.
- Enter a **Pitch angle** (SI unit: rad), which is a right-handed rotation with respect to the $+y$ direction. The default is 0 rad, corresponding to the initial direction of propagation pointing in the $+x$ direction.

- Enter a **Yaw angle** (SI unit: rad), which is a right-handed rotation with respect to the +z direction.
- Enter a **Wave number** k (SI unit: rad/m). The default is emw.k_0 rad/m. The wave number must evaluate to a value that is the same for the domains the scattered field is applied to.

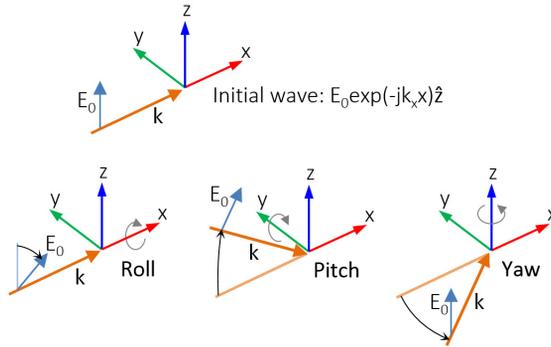


Figure 4-1: Schematic of the directions for the wave vector \mathbf{k} , the electric field \mathbf{E}_0 , and the roll, pitch and yaw rotations. The top image represents an initial wave propagating in the x direction with a polarization along the z direction.

Circularly Polarized Plane Wave

The background wave is defined as

$$\mathbf{E}_b(r, \varphi, z) = \tilde{\mathbf{E}}_b(r, z) e^{-jm\varphi},$$

where

$$\tilde{\mathbf{E}}_b(r, z) = E_0(\hat{\mathbf{r}} - jm\hat{\boldsymbol{\phi}}) e^{-jkz},$$

m is the azimuthal mode number (+1 or -1) varying depending on the **Circular polarization type** and **Direction of propagation** settings, and $\hat{\mathbf{r}}$ and $\hat{\boldsymbol{\phi}}$ are the unit vectors in the r and φ directions, respectively.

- Select the **Circular polarization type** — **Right handed** or **Left handed**.
- Select the **Direction of propagation** — **+z** or **-z**.
- Enter an **Electric field amplitude** E_0 (SI unit: V/m). The default is 1 V/m.
- Enter an **Wave number** k (SI unit: rad/m). The default is emw.k_0 rad/m.

COMPONENTS

This section is available for 2D and 2D axisymmetric components.

Select the **Electric field components solved for** — **Three-component vector**, **Out-of-plane vector**, or **In-plane vector**. Select:

- **Three-component vector** (the default) to solve using a full three-component vector for the electric field \mathbf{E} .
- **Out-of-plane vector** to solve for the electric field vector component perpendicular to the modeling plane, assuming that there is no electric field in the plane.
- **In-plane vector** to solve for the electric field vector components in the modeling plane assuming that there is no electric field perpendicular to the plane.

OUT-OF-PLANE WAVE NUMBER

This section is available for 2D and 2D axisymmetric components, when solving for **Three-component vector** or **In-plane vector**.

For 2D components, assign a wave vector component to the **Out-of-plane wave number** field. For 2D axisymmetric components, assign an integer constant or an integer parameter expression to the **Azimuthal mode number** field.

ANALYSIS METHODOLOGY

From the **Methodology options** list, select one of three solver configurations: **Robust**, **Intermediate**, or **Fast** (the default).

The settings of each methodology option are found in **Solver Configurations** and the subsidiary nodes.

PORT SWEEP SETTINGS

Select the **Use port sweep** check box to enable the port sweep. When selected, this invokes a parametric sweep over the ports in addition to the frequency sweep already added. The generated lumped parameters are in the form of an S-parameter matrix.

For **Use port sweep** enter a **Sweep parameter name** to assign a specific name to the parameter that controls the port number solved for during the sweep. Before making the port sweep, the parameter must also have been added to the list of parameters in the **Parameters** section of the **Parameters** node under the **Global Definitions** node. This process can be automated by clicking the **Configure Sweep Settings** button. The **Configure Sweep Settings** button helps add a necessary port sweep parameter and a **Parametric Sweep** study step in the last study node. If there is already a **Parametric Sweep** study step, the sweep settings are adjusted for the port sweep.

Select **Export Touchstone file** and the S-parameters are subject to **Touchstone file export**. Click **Browse** to locate the file, or enter a file name and path. Select an **Parameter format (value pair)**: **Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

Enter a **Reference impedance for Touchstone file export** Z_{ref} (SI unit: Ω) that is used only for the header in the exported Touchstone file. The default is 50 Ω .

DEPENDENT VARIABLES

The dependent variables (field variables) are for the **Electric field \mathbf{E}** and its components (in the **Electric field components** fields). The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

Select the shape order for the **Electric field** dependent variable — **Linear**, **Quadratic** (the default), or **Cubic**. For more information about the **Discretization** section, see [Settings for the Discretization Sections](#) in the *COMSOL Multiphysics Reference Manual*.



- [Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Frequency Domain Interface](#)
- [Theory for the Electromagnetic Waves Interfaces](#)



H-Bend Waveguide 3D: Application Library path **RF_Module/Transmission_Lines_and_Waveguides/h_bend_waveguide_3d** demonstrates how to set up a port sweep.

Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Frequency Domain Interface

The [Electromagnetic Waves, Frequency Domain Interface](#) has these domain, boundary, edge, point, and pair nodes and subnodes, listed in alphabetical order, 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.

DOMAIN

- Archie's Law
- Divergence Constraint
- External Current Density
- Far-Field Calculation
- Specific Absorption Rate
- Far-Field Domain
- Initial Values
- Effective Medium
- Wave Equation, Electric

BOUNDARY CONDITIONS

With no surface currents present the boundary conditions

$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{0}$$

$$\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$

need to be fulfilled. Because \mathbf{E} is being solved for, the tangential component of the electric field is always continuous, and thus the first condition is automatically fulfilled. The second condition is equivalent to the natural boundary condition

$$-\mathbf{n} \times [(\mu_r^{-1} \nabla \times \mathbf{E})_1 - (\mu_r^{-1} \nabla \times \mathbf{E})_2] = \mathbf{n} \times j\omega\mu_0(\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$

and is therefore also fulfilled. These conditions are available (listed in alphabetical order):

- Diffraction Order
- Electric Field
- Impedance Boundary Condition
- Lumped Element
- Lumped Port
- Magnetic Field
- Orthogonal Polarization
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Condition
- Port
- Scattering Boundary Condition
- Surface Current Density
- Surface Magnetic Current Density
- Surface Roughness
- Transition Boundary Condition

EDGE, POINT, AND PAIR

- Circular Port Reference Axis
- Edge Current
- Electric Field
- Electric Point Dipole
- Electric Potential
- Ground
- Integration Line for Current
- Integration Line for Voltage
- Line Current (Out-of-Plane)
- Magnetic Current
- Magnetic Point Dipole
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Port Reference Point
- Reference Point
- Surface Current Density
- Surface Magnetic Current Density



For 2D 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](#) 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.

Wave Equation, Electric

Wave Equation, Electric is the main feature node for this physics interface. The governing equation can be written in the form

$$\nabla \times (\mu_{\text{r}}^{-1} \nabla \times \mathbf{E}) - k_0^2 \epsilon_{\text{rc}} \mathbf{E} = \mathbf{0}$$

for the time-harmonic and eigenfrequency problems. The wave number of free space k_0 is defined as

$$k_0 = \omega \sqrt{\epsilon_0 \mu_0} = \frac{\omega}{c_0}$$

where c_0 is the speed of light in vacuum.

In 2D the electric field varies with the out-of-plane wave number k_z as

$$\mathbf{E}(x, y, z) = \tilde{\mathbf{E}}(x, y) \exp(-ik_z z).$$

The wave equation is thereby rewritten as

$$(\nabla - ik_z \mathbf{z}) \times [\mu_r^{-1} (\nabla - ik_z \mathbf{z}) \times \tilde{\mathbf{E}}] - k_0^2 \epsilon_{rc} \tilde{\mathbf{E}} = \mathbf{0},$$

where \mathbf{z} is the unit vector in the out-of-plane z direction.

Notice that the ansatz above just explains how the wave equation is modified when the out-of-plane wave vector component k_z is not zero. As an example, for a plane wave with a nonzero out-of-plane wave vector component, the electric field is of course given by

$$\mathbf{E}(x, y, z) = \tilde{\mathbf{E}}(x, y) \exp(-ik_z z) = \mathbf{A} \exp(-i(k_x x + k_y y + k_z z)),$$

where \mathbf{A} is a constant amplitude and k_x , k_y , and k_z are the wave vector components.

In 2D axisymmetry, the electric field varies with the azimuthal mode number m as

$$\mathbf{E}(r, \varphi, z) = \tilde{\mathbf{E}}(r, z) \exp(-im\varphi).$$

For this case, the wave equation is rewritten as

$$\left(\nabla - i \frac{m}{r} \boldsymbol{\varphi} \right) \times \left[\mu_r^{-1} \left(\nabla - i \frac{m}{r} \boldsymbol{\varphi} \right) \times \tilde{\mathbf{E}} \right] - k_0^2 \epsilon_{rc} \tilde{\mathbf{E}} = \mathbf{0},$$

where $\boldsymbol{\varphi}$ is the unit vector in the out-of-plane φ direction.

When solving the equations as an eigenfrequency problem the eigenvalue is the complex eigenfrequency $\lambda = -j\omega + \delta$, where δ is the damping of the solution. The Q-factor is given from the eigenvalue by the formula

$$Q_{\text{fact}} = \frac{\omega}{2|\delta|}$$

Using the relation $\epsilon_r = n^2$, where n is the refractive index, the equation can alternatively be written

$$\nabla \times (\nabla \times \mathbf{E}) - k_0^2 n^2 \mathbf{E} = \mathbf{0}$$

When the equation is written using the refractive index, the assumption is that $\mu_r = 1$ and $\sigma = 0$ and only the constitutive relations for linear materials are available. When solving for the scattered field the same equations are used but $\mathbf{E} = \mathbf{E}_{sc} + \mathbf{E}_i$ and \mathbf{E}_{sc} is the dependent variable.

The [Divergence Constraint](#) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

ELECTRIC DISPLACEMENT FIELD

Select an **Electric displacement field model** — **Relative permittivity** (the default), **Refractive index**, **Loss tangent**, **loss angle**, **Loss tangent, dissipation factor**, **Dielectric loss**, **Drude-Lorentz dispersion model**, or **Debye dispersion model**.

Relative Permittivity

When **Relative permittivity** is selected, the default **Relative permittivity** ϵ_r takes values **From material**. Select **Effective medium** to add an [Effective Medium](#) subnode, or for **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix.

Refractive Index

When **Refractive index** is selected, the default **Refractive index** n and **Refractive index, imaginary part** k take the values **From material**. To specify the real and imaginary parts of the refractive index and assume a relative permeability of unity and zero conductivity, for one or both of the options, select **User defined** then choose **Isotropic**, **Diagonal**, **Symmetric**, or **Full**. Enter values or expressions in the field or matrix.



The diagonal components of the input refractive index matrix correspond to the semi-axes of the so called index ellipsoid. You can orient the index ellipsoid by first creating a suitably oriented coordinate system below the **Definitions** node for the model component. Then select the created coordinate system in the **Coordinate system** setting in the **Coordinate System Selection** section in the settings for the **Wave Equation, Electric** feature.



Beware of the time-harmonic sign convention requiring a lossy material having a negative imaginary part of the refractive index (see [Introducing Losses in the Frequency Domain](#)).

Loss Tangent, Loss Angle

When **Loss tangent, loss angle** is selected, the default **Relative permittivity (real part) ϵ'** and **Loss tangent, loss angle δ** take values **From material**. For **User defined** select **Isotropic, Diagonal, Symmetric**, or **Full** and enter values or expressions in the field or matrix. Then if **User defined** is selected for **Loss tangent, loss angle δ** , enter a value to specify a loss angle for dielectric losses. This assumes zero conductivity.

Loss Tangent, Dissipation Factor

When **Loss tangent, dissipation factor** is selected, the default **Relative permittivity (real part) ϵ'** and **Loss tangent, dissipation factor $\tan\delta$** take values **From material**. For **User defined** select **Isotropic, Diagonal, Symmetric**, or **Full** and enter values or expressions in the field or matrix. Then if **User defined** is selected for **Loss tangent, dissipation factor $\tan\delta$** , enter a value to specify a dissipation for dielectric losses. This assumes zero conductivity.

Dielectric Loss

When **Dielectric loss** is selected, the default **Relative permittivity ϵ'** and **Relative permittivity (imaginary part) ϵ''** take values **From material**. For **User defined** select **Isotropic, Diagonal, Symmetric**, or **Full** and enter values or expressions in the field or matrix.



Beware of the time-harmonic sign convention requiring a lossy material having a negative imaginary part of the relative permittivity (see [Introducing Losses in the Frequency Domain](#)).

Drude-Lorentz Dispersion Model

The **Drude-Lorentz dispersion model** is defined by the equation

$$\epsilon_r(\omega) = \epsilon_\infty + \sum_{j=1}^M \frac{f_j \omega_p^2}{\omega_{0j}^2 - \omega^2 + i\Gamma_j \omega}$$

where ϵ_∞ is the high-frequency contribution to the relative permittivity, ω_p is the plasma frequency, f_j is the oscillator strength, ω_{0j} is the resonance frequency, and Γ_j is the damping coefficient.

When **Drude-Lorentz dispersion model** is selected, the default **Relative permittivity, high frequency ϵ_∞** (dimensionless) takes its value **From material**. For **User defined** select **Isotropic, Diagonal, Symmetric**, or **Full** and enter a value or expression in the field or matrix.

Enter a **Plasma frequency** ω_∞ (SI unit: rad/s). The default is 0 rad/s.

In the table, enter values or expressions in the columns for the **Oscillator strength**, **Resonance frequency (rad/s)**, and **Damping in time (rad/s)**.

Debye Dispersion Model

The **Debye dispersion model** is given by

$$\epsilon(\omega) = \epsilon_\infty + \sum_k \frac{\Delta\epsilon_k}{1 + i\omega\tau_k}$$

where ϵ_∞ is the high-frequency contribution to the relative permittivity, $\Delta\epsilon_k$ is the contribution to the relative permittivity, and τ_k is the relaxation time.

When **Debye dispersion model** is selected, the default **Relative permittivity, high frequency** ϵ_∞ (dimensionless) takes its value **From material**. For **User defined** select **Isotropic, Diagonal, Symmetric**, or **Full** and enter a value or expression in the field or matrix.

In the table, enter values or expressions in the columns for the **Relative permittivity contribution** and **Relaxation time (s)**.

MAGNETIC FIELD

Select the **Constitutive relation** — **Relative permeability** (the default) or **Magnetic losses**.



For magnetic losses, beware of the time-harmonic sign convention requiring a lossy material having a negative imaginary part of the relative permeability (see [Introducing Losses in the Frequency Domain](#)).

- For **Relative permeability** the relative permeability μ_r uses values **From material**. For **User defined** select **Isotropic, Diagonal, Symmetric**, or **Full** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix. If **Effective medium** is selected, the [Effective Medium](#) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.
- For **Magnetic losses** the default values for **Relative permeability (real part)** μ' and **Relative permeability (imaginary part)** μ'' are taken **From material**. For **User defined** enter different values.

CONDUCTION CURRENT

By default, the **Electrical conductivity** σ (SI unit: S/m) uses values **From material**.

- For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the current and enter values or expressions in the field or matrix.
- For **Linearized resistivity** the default values for the **Reference temperature** T_{ref} (SI unit: K), **Resistivity temperature coefficient** α (SI unit: 1/K), and **Reference resistivity** ρ_0 (SI unit: $\Omega \cdot \text{m}$) are taken **From material**. For **User defined** enter other values or expressions for any of these variables.
- When **Effective medium** is selected, the **Effective Medium** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.
- When **Archie's law** is selected, the **Archie's Law** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Divergence Constraint

The **Divergence Constraint** subnode is available from the context menu (right-click the **Wave Equation**, **Electric** parent node) or from the **Physics** toolbar, **Attributes** menu. It is used for numerical stabilization when the frequency is low enough for the total electric current density related term in the wave equation to become numerically insignificant. For **The Electromagnetic Waves**, **Frequency Domain Interface** and **Heat Transfer Interfaces** the divergence condition is given by

$$\nabla \cdot \mathbf{J} = 0$$

and for **The Electromagnetic Waves**, **Transient Interface** it is

$$\nabla \cdot (\sigma \mathbf{A}) = 0$$

DIVERGENCE CONSTRAINT

Enter a value or expression for the **Divergence condition variable scaling** ψ_0 .

For the **Electromagnetic Waves**, **Frequency Domain** and **Microwave Heating** interfaces, the SI unit is $\text{kg}/(\text{m} \cdot \text{s}^3 \cdot \text{A})$. The default is $1 \text{ kg}/(\text{m} \cdot \text{s}^3 \cdot \text{A})$.

For the **Electromagnetic Waves**, **Transient interface** (and the **Microwave Plasma** interface available with the **Plasma Module**) the SI unit is A/m and the default is 1 A/m.

Initial Values

The **Initial Values** node adds an initial value for the electric field that can serve as an initial guess for a nonlinear solver. Add additional **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter values or expressions for the initial values of the components of the **Electric field** \mathbf{E} (SI unit: V/m). The default values are 0 V/m.

External Current Density

The **External Current Density** node adds an externally generated current density \mathbf{J}_e , which appears in Ohm's law

$$\mathbf{J} = \sigma \mathbf{E} + \mathbf{J}_e$$

and in the equation that the physics interface defines.

EXTERNAL CURRENT DENSITY

Based on space dimension, enter the components (\mathbf{x} , \mathbf{y} , and \mathbf{z} for 3D components for example) of the **External current density** \mathbf{J}_e (SI unit: A/m²).

Far-Field Domain

To set up a far-field calculation, add a **Far-Field Domain** node and specify the far-field domains in its Settings window. Use [Far-Field Calculation](#) subnodes (one is added by default) to specify all other settings needed to define the far-field calculation. If a **Perfectly Matched Layer** (PML) node has been added before adding the **Far-Field Domain**, all of the domains in the Electromagnetic Waves, Frequency Domain interface adjacent to the PML are automatically selected by default. If there is no PML, all of the domains are selected. The selection can be modified. In that case, select only a homogeneous domain or domain group that is outside of all radiating and scattering objects and which has the material settings of the far-field medium.



[Far-Field Support in the Electromagnetic Waves, Frequency Domain Interface](#)



Radar Cross Section: Application Library path **RF_Module/Scattering_and_RCS/radar_cross_section**

Biconical Antenna: Application Library path **RF_Module/Antennas/biconical_antenna**

Far-Field Calculation

A **Far-Field Calculation** subnode is added by default to the **Far-Field Domain** node and is used to select boundaries corresponding to a single closed surface surrounding all radiating and scattering objects. By default, all exterior boundaries of the **Far-Field Domain** are selected. If a **Perfectly Matched Layer (PML)** node has been added before adding the **Far-Field Domain**, all exterior boundaries of the **Far-Field Domain** adjacent to the PML are selected. Symmetry reduction of the geometry makes it relevant to select boundaries defining a nonclosed surface. Also use this feature to indicate symmetry planes and symmetry cuts applied to the geometry, and whether the selected boundaries are defining the inside or outside of the far field domain; that is, to say whether they are facing away from infinity or toward infinity.

FAR-FIELD CALCULATION

Enter a **Far-field variable name**. The default is Efar.

Select as needed the **Symmetry in the x=0 plane**, **Symmetry in the y=0 plane**, or **Symmetry in the z=0 plane** check boxes to use it your model when calculating the far-field variable. The symmetry planes have to coincide with one of the Cartesian coordinate planes.

When a check box is selected, also choose the type of symmetry to use from the **Symmetry type** list that appears — **Symmetry in E (PMC)** or **Symmetry in H (PEC)**. The selection should match the boundary condition used for the symmetry boundary. Using these settings, include the parts of the geometry that are not in the model for symmetry reasons in the far-field analysis.

From the **Boundary relative to domain** list, select **Inside** or **Outside** (the default) to define if the selected boundaries are defining the inside or outside of the far-field domain (that is, whether facing away from infinity or toward infinity).



If perfectly matched layers are added to the model after the [Far-Field Domain](#) is configured, then it is necessary to press the **Reset Far-Field Boundaries** button to reassign all exterior boundaries.



Dielectric Resonator Antenna: Application Library path **RF_Module/Antennas/dielectric_resonator_antenna**

Specific Absorption Rate

To calculate specific absorption rate (SAR), add a **Specific Absorption Rate** node and specify the material density in its Settings window. The specific absorption rate is calculated as

$$SAR = \frac{\sigma |\mathbf{E}|^2}{2\rho}.$$

Density

When using **From material**, ρ is defined from the material node where the domain selection is common to this feature. Select **User defined** to enter the value directly.



Specific Absorption Rate (SAR) in the Human Brain: Application Library path **RF_Module/Microwave_Heating/sar_in_human_head**

Perfect Electric Conductor

The **Perfect Electric Conductor** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{0}$$

is a special case of the electric field boundary condition that sets the tangential component of the electric field to zero. It is used for modeling of a lossless metallic surface (for example, a ground plane) or as a symmetry type boundary condition. It imposes symmetry for magnetic fields and “magnetic currents” and antisymmetry for electric fields and electric currents. It supports induced electric surface currents and

thus any prescribed or induced electric currents (volume, surface, or edge currents) flowing into a *perfect electric conductor* boundary is automatically balanced by induced surface currents.

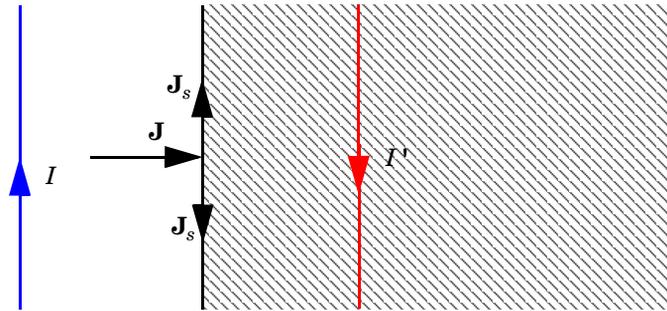


Figure 4-2: The perfect electric conductor boundary condition is used on exterior and interior boundaries representing the surface of a lossless metallic conductor or (on exterior boundaries) representing a symmetry cut. The shaded (metallic) region is not part of the model but still carries effective mirror images of the sources. Note also that any current flowing into the boundary is perfectly balanced by induced surface currents. The tangential electric field vanishes 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 box. For information about the **Constraint Settings** section, see [Constraint Settings](#) in the *COMSOL Multiphysics Reference Manual*.



RF Coil: Application Library path **RF_Module/Passive_Devices/rf_coil**

Perfect Magnetic Conductor

The **Perfect Magnetic Conductor** boundary condition

$$\mathbf{n} \times \mathbf{H} = \mathbf{0}$$

is a special case of the surface current boundary condition that sets the tangential component of the magnetic field and thus also the surface current density to zero. On external boundaries, this can be interpreted as a “high surface impedance” boundary condition or used as a symmetry type boundary condition. It imposes symmetry for electric fields and electric currents. Electric currents (volume, surface, or edge currents) are not allowed to flow into a *perfect magnetic conductor* boundary as that

would violate current conservation. On interior boundaries, the perfect magnetic conductor boundary condition literally sets the tangential magnetic field to zero, which in addition to setting the surface current density to zero also makes the tangential electric field discontinuous.

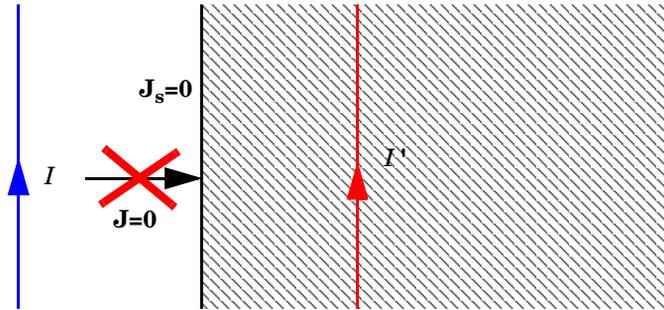


Figure 4-3: The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut. The shaded (high impedance) region is not part of the model but nevertheless carries effective mirror images of the sources. Note also that any electric current flowing into the boundary is forbidden as it cannot be balanced by induced electric surface currents. The tangential magnetic field vanishes at the boundary. On interior boundaries, the perfect magnetic conductor boundary condition literally sets the tangential magnetic field to zero which in addition to setting the surface current density to zero also makes the tangential electric field (and in dynamics the tangential electric field) discontinuous.



Magnetic Frill: Application Library path **RF_Module/Antennas/magnetic_frrill**

Port

Use the **Port** node where electromagnetic energy enters or exits the model. A port can launch and absorb specific modes. Use the boundary condition to specify wave type ports. Ports support S-parameter calculations but can be used just for exciting the model. This node is not available with the Electromagnetic Waves, Transient interface.

A port assumes that the cross section's geometry and material is constant in the normal direction. Furthermore, the port boundary is assumed to be flat, resulting in a constant normal across the boundary.

In 3D, the following subnodes are available from the context menu, by right-clicking the **Port** node, or from the **Physics** toolbar, **Attributes** menu:

- **Circular Port Reference Axis** to determine a reference direction for the modes. This subnode is selected from the **Points** submenu when **Circular** is selected as the type of port.
- **Ground** to define zero potential boundaries for the port mode field calculation in **TEM Boundary Mode Analysis**. This subnode is available in the context menu when **Transverse electromagnetic (TEM)** is selected as the type of port.
- **Electric Potential** to define positive or negative 1 V potential boundaries for the port mode field calculation in **TEM Boundary Mode Analysis**. This subnode is available in the context menu when **Transverse electromagnetic (TEM)** is selected as the type of port.
- **Periodic Port Reference Point** to uniquely determine reciprocal lattice vectors. This subnode is selected from the **Points** submenu when **Periodic** is selected as the type of port.

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 and numeric port names are also required for port sweeps and Touchstone file export.

Select the **Type of Port** — **User defined**, **Numeric**, **Transverse electromagnetic (TEM)**, **Rectangular**, **Coaxial**, **Circular**, or **Periodic**.

Periodic ports are available in 3D and 2D. **Circular** and **Coaxial** ports are available in 3D and 2D axisymmetry.

Numeric ports require a **Boundary Mode Analysis** study type. It should appear before the frequency domain study node in the study branch of the model tree. If more than one numeric port is needed, use one Boundary Mode Analysis node per port and assign each to the appropriate port. Then, it is best to add all the studies; Boundary Mode Analysis 1, Boundary Mode Analysis 2, ..., Frequency Domain 1, manually. **Numeric**

ports are by default computed for the deformed mesh whereas other types of ports compute the mode shape using geometry information.

The Boundary Mode Analysis study step stores the frequency f_{ref} and propagation constant β_{ref} for which it was run. For a TE, TM, or TEM mode, the propagation constant β for an arbitrary frequency f is given by

$$\beta^2 = \beta_{\text{ref}}^2 + k^2(1 - (f_{\text{ref}}/f)^2).$$



In addition, for TE, TM, and TEM modes, the mode field shape is independent of the frequency. Thus, when making a frequency sweep including only TE, TM, and TEM modes, the Boundary Mode Analysis study steps can be done for just one frequency, with the propagation constants obtained from the expression above for the other frequencies. For waveguides consisting of multiple dielectric materials, like optical fibers, where there are no TE, TM, or, TEM modes, the Boundary Mode Analysis steps must be recomputed for each frequency.



It is only possible to excite one port at a time if the purpose is to compute S-parameters. In other cases (for example, when studying microwave heating) more than one inport 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.

Transverse electromagnetic (TEM) ports are only available in 3D models. It requires a **TEM Boundary Mode Analysis** study type. It should appear before the frequency domain study node in the study branch of the model tree. Add Ground and Electric Potential subnodes to compute the port mode field and impedance. These subnodes are available from the context menu (right-click the Port parent node) or from the **Physics** toolbar, **Attributes** menu. Enter a **Characteristic impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω . The characteristic impedance of a port is calculated using the square root of the ratio of the inductance and capacitance. The electric mode field on a port boundary is scaled by the ratio between the computed mode characteristic impedance and Z_{ref} .

Wave Excitation at this Port

To set whether it is an inport or a listener port, select **On** or **Off** from the **Wave excitation at this port** list. If **On** is selected, enter a **Port input power** P_{in} (SI unit: W) or a **Deposited power** P_{dep} (SI unit: W) if the **Enable active port feedback** check box is marked. When

a **Deposited power** is specified, the input power is adjusted so that the power that is not reflected equals the specified **Deposited power**. If **Enable active port feedback** is marked this must be the only inport.



The [Port Sweep Settings](#) section in the Electromagnetic Waves, Frequency Domain interface cycles through the ports, computes the entire S-matrix and exports it to a Touchstone file. When using port sweeps, the local setting for **Wave excitation at this port** is overridden by the solver so only one port at a time is excited.

Activate Slit Condition

Select the **Activate slit condition on interior port** check box to use the **Port** boundary condition on interior boundaries.

Then select a **Slit type** — **PEC-backed** (the default) or **Domain-backed**. The **PEC-backed** type makes the port on interior boundaries perform as it does on exterior boundaries. The **Domain-backed** type can be combined with perfectly matched layers to absorb the excited mode from a source port and other higher order modes.

Click **Toggle Power Flow Direction** button to define the power flow for the port. For an excited port, the power flow should point in to the excited domain and for a listener port the power flow should point out from the excited domain. The power flow direction is visualized with a red arrow on the port boundary in the Graphics window.



When the **Slit type** is set to **Domain-backed**, there must be no waves reflected from the domain backing the port. Thus, the backing domain must have homogeneous material and geometric properties and it should be truncated by a PML domain or a nonreflecting boundary condition.



When an active slit port is adjacent to a periodic condition, the periodic condition needs to be split at the active port, as the field is discontinuous at the point (in 2D) or edge (in 3D) where the slit port and the periodic condition meet.

Analyze as a TEM Field

This check box is available for 3D components and when the **Type of port** is **Numeric**.

Select the **Analyze as a TEM field** check box to add [Integration Line for Current](#) and [Integration Line for Voltage](#) subnodes. These subnodes are available from the context menu (right-click the Port parent node) or from the **Physics** toolbar, **Attributes** menu.

Enter a reference **Characteristic impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω . The characteristic impedance of a port is calculated using the ratio of the voltage and current using **Boundary Mode Analysis**. The mode field on a port boundary is scaled by the ratio between the computed characteristic impedance and user-specified Z_{ref} .



Notch Filter Using a Split Ring Resonator: Application Library path
RF_Module/Filters/notch_filter_srr

PORT MODE SETTINGS

The input is based on the **Type of Port** selected above — [User Defined](#), [Rectangular](#), [Circular](#), or [Periodic](#). No entry is required if **Numeric** or **Coaxial** are selected.

Set the **Mode phase** θ_{in} (SI unit: rad) for the port mode field. The default is 0 radians. For instance, if the inspected port mode field is polarized in the opposite direction compared to the expected direction, a **Mode phase** of π (enter pi in the field) can be used for polarizing the mode field in the expected direction. Notice that a change of the **Mode phase**, either on the exciting or the listener port, changes also the S-parameter coupling the exciting and the listener port. However, a change of the **Mode phase** on the exciting port does not modify the reflection coefficient (normally denoted S_{11}) associated with the exciting port.

User Defined

For **User defined** specify the eigenmode of the incoming wave at the port. Even if **Wave excitation at this port** is set to **Off**, the mode field should represent the incoming wave that corresponds to the actual outgoing wave. The mode field can be entered with an arbitrary amplitude and is normalized internally.

- Enter the components of the **Electric mode field** \mathbf{E}_0 (SI unit: V/m) or the **Magnetic mode field** \mathbf{H}_0 (SI unit: A/m). The entered expressions must be differentiable.
- Enter the **Propagation constant** β (SI unit: rad/m). This is frequency dependent for all but TEM modes and a correct frequency-dependent expression must be used.



Notice that expressions including coupling operators are not differentiable and cannot be used as mode fields.

Rectangular

For **Rectangular** specify a unique rectangular mode.

For 3D components, select a **Mode type** — **Transverse electric (TE)** or **Transverse magnetic (TM)**. Enter the **Mode number**, for example, 10 for a TE₁₀ mode, or 11 for a TM₁₁ mode.

For 2D components, to excite the fundamental mode, select the mode type **Transverse electromagnetic (TEM)**, since the rectangular port represents a parallel-plate waveguide port that can support a TEM mode. Only TE modes are possible when solving for the out-of-plane vector component, and only TM and TEM modes are possible when solving for the in-plane vector components. There is only a single mode number, which is selected from a list.

Coaxial

In 2D axisymmetry, **Coaxial** does not support nonzero azimuthal mode number. The **Azimuthal mode number** in the **Physics interface** should be defined as zero.

Circular

For **Circular** specify a unique circular mode.

- Select a **Mode type** — **Transverse electric (TE)** or **Transverse magnetic (TM)**.
- Select the **Mode number** from the list.

For 3D components, enter the **Mode number**, for example, 11 for a TE₁₁ mode, or 01 for a TM₀₁ mode. When **Circular** is selected as the type of port in 3D, the **Circular Port Reference Axis** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. It defines the orientation of fields on a port boundary.

For 2D axisymmetry components, select whether the **Azimuthal mode number** is defined in the **Physics interface** or if it is **User defined**. For **User defined** define an integer constant or an integer parameter expression for the **Azimuthal mode number**. Note that the absolute value of the **Azimuthal mode number** must be less than 11.

Periodic

For **Periodic** specify parameters for the mode field. When **Periodic** is selected, the **Diffraction Order** port subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Select a **Input quantity** — **Electric field** or **Magnetic field** and define the mode field amplitude for the incoming wave at the port. Even if **Wave excitation at this port** is set

to **Off**, the mode field amplitude should represent the incoming wave that corresponds to the actual outgoing wave.

- For 2D components and if the **Input quantity** is set to **Electric field**, define the **Electric mode field amplitude**. For example, for a TE wave set the x , y , and z components to 0, 0, 1. Similarly, if the **Input quantity** is set to **Magnetic field**, define the **Magnetic mode field amplitude**. For a TM wave set the x , y , and z components to 0, 0, and 1.
- Define the **Angle of incidence**, if **Wave excitation at this port** is **On**.

For 3D components, if **Wave excitation at this port** is **On**, define the **Elevation angle of incidence** and **Azimuth angle of incidence**. The **Elevation angle of incidence** α_1 and **Azimuth angle of incidence** α_2 are used in the relations

$$\mathbf{k} = \mathbf{k}_{\text{parallel}} + \mathbf{k}_{\text{perpendicular}}$$

$$\mathbf{k}_{\text{parallel}} = \mathbf{k}_F = |k| \sin \alpha_1 (\vec{\mathbf{a}}_1 \cos \alpha_2 + \mathbf{n} \times \vec{\mathbf{a}}_1 \sin \alpha_2)$$

where \mathbf{k} is the wave vector, $\mathbf{k}_{\text{parallel}}$ is the projection of \mathbf{k} onto the port, \mathbf{k}_F is the k -vector for Floquet periodicity, \mathbf{n} is the outward unit normal vector to the boundary, and $\vec{\mathbf{a}}_1$ is one of the normalized primitive unit cell vectors from the periodic structure defined from [Periodic Port Reference Point](#).

The **Elevation angle of incidence** α_1 is the angle between \mathbf{n} and \mathbf{k} .

The **Azimuth angle of incidence** is the counterclockwise rotating angle from the primitive vector \mathbf{a}_1 around the axis built with [Periodic Port Reference Point](#) and \mathbf{n} .

For periodic ports with hexagonal port boundaries, the definition of the vector \mathbf{a}_1 is slightly different from the default definition. In this case, the unit cell is actually a rhomboid, with primitive vectors pointing in other directions than the side vectors of the hexagon. Thus, for a hexagonal periodic port, the vector \mathbf{a}_1 is defined along one of the sides of the hexagon, and it is not one of the primitive vectors of the hexagonal point lattice. The **Azimuth angle of incidence** α_2 is still measured from the vector \mathbf{a}_1 , even though this vector now refers to a side vector of the hexagonal port boundary and not a primitive vector.



Hexagonal Grating: Application Library path **RF_Module/Tutorials/hexagonal_grating**

For 2D components define the **Angle of incidence**. The **Angle of incidence** α is defined by the relation

$$\mathbf{k} \times \mathbf{n} = k \sin \alpha \mathbf{z}$$

where \mathbf{k} is the projection of the wave vector in the xy -plane, \mathbf{n} is the normalized normal vector to the boundary, k is the magnitude of the projected wave vector in the xy -plane, and \mathbf{z} is the unit vector in the z direction.



Notice that the mode field defined for the Periodic port assumes homogeneous isotropic material properties in the domain adjacent to the selected port boundary.



The propagation directions for listener **Periodic** ports are deduced from the angle setting(s) for the source **Periodic** port and the refractive indices defined for the source and the listener ports. Thus, adding source **Periodic** ports with different propagation angles will give ambiguous propagation directions for the listener **Periodic** ports.

Default Polarization plots are automatically generated for **Periodic** ports in 3D and in 2D, if the **Electric field components solved for** setting in the **Components** section for the physics interface is set to **Three-component vector**. The Polarization plot includes polarization ellipses for each diffraction order. The polarization ellipse line graphs are generated by plotting the in-plane Jones vector element versus the out-of-plane Jones vector element.



For more information about Jones Vectors and Polarization plots, see [Jones Vectors and Polarization Plots](#).

AUTOMATIC DIFFRACTION ORDER CALCULATION

This section is only available for **Periodic** ports to provide parameter settings that are used when automatically adding [Diffraction Order](#) subnodes to **Periodic** ports.

- Select the **Include in automatic diffraction order calculation** check box to add **Diffraction Order** subnodes to the selected **Periodic** port, when the **Add Diffraction Orders** button is clicked from the exciting **Periodic** port.
- Define the **Refractive index, real part** at the boundary.
- Define the **Maximum frequency** — **From study** (the default) or **User defined**. When **From study** is selected, the **Maximum frequency** is taken from the study step associated with the physics interface. For **User defined**, enter the maximum frequency f_{\max} (SI

unit: Hz). The default value is 0 Hz. If a single frequency is used, insert the frequency, or if a frequency sweep is performed, insert the maximum frequency of the sweep. This parameter is only available when **Wave excitation at this port** is **On**.

When all parameters are defined, click the **Add Diffraction Orders** button from the exciting **Periodic** port to automatically create **Diffraction Order** ports as subnodes to all **Periodic** ports having the **Include in automatic diffraction order calculation** check box selected.

CALCULATOR

This utility is available for **Rectangular** and **Circular** port types. Enter a **Relative permittivity** (default is 1) of the material fully filled in a waveguide and then click the **Compute Waveguide Cutoff Frequency** button to compute the cutoff frequency for the particular waveguide geometry and the selected **Mode type**. The result is displayed in the **Messages** window.

DE-EMBEDDING PORT

Set the **Port Offset** d_{offset} (SI unit: m) for the de-embedded S-parameter calculation. The default is 0 m. The phase of the de-embedded S-parameters is adjusted from the calculated S-parameters with the propagation constant and the value of d_{offset} .

$$\text{De-embedded } S_{nm} = S_{nm} \exp(j\beta_n d_n) \exp(j\beta_m d_m)$$

where m is the source port name, n is the listener port name, β is propagation constant, and d is offset distance from the port boundary.

The de-embedding functionality is triggered when d_{offset} is set to a nonzero value. It is assumed that the domain between the port boundary and the boundary projected by the d_{offset} is straight, while maintaining a constant cross-sectional shape.

CONSTRAINT SETTINGS

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



- [S-Parameters and Ports](#)
 - [S-Parameter Variables](#)
-



3D model with numeric ports — *Waveguide Adapter*: Application Library path **RF_Module/Transmission_Lines_and_Waveguides/waveguide_adapter**

2D model with rectangular ports — *Three-Port Ferrite Circulator*: Application Library path **RF_Module/Ferrimagnetic_Devices/circulator**

2D model with periodic ports — *Plasmonic Wire Grating*: Application Library path **RF_Module/Tutorials/plasmonic_wire_grating**

3D model using slit conditions — *Frequency Selective Surface, Periodic Complementary Split Ring Resonator*: Application Library path **RF_Module/EMI_EMG_Applications/frequency_selective_surface_csrr**

Integration Line for Current

The **Integration Line for Current** is available only in 3D from the context menu (right-click the **Port** node) when the **Analyze as a TEM field** check box is selected under the **Port Properties** section for the **Port** node.

Integration Line for Voltage

The **Integration Line for Voltage** is available only in 3D from the context menu (right-click the **Port** node) when the **Analyze as a TEM field** check box is selected under the **Port Properties** section for the **Port** node. The characteristic impedance of a **Numeric** port is defined by the ratio between the voltage and current.

Ground

The **Ground** subnode is available from the context menu (right-click the **Port** node) when **Transverse electromagnetic (TEM)** is selected as the type of port. Zero potential is assigned on the boundary selection in the port mode field calculation.

Electric Potential

The **Electric Potential** subnode is available from the context menu (right-click the **Port** node) when **Transverse electromagnetic (TEM)** is selected as the type of port. Either positive or negative 1 V is assigned on the boundary selection in the port mode field calculation.

ELECTRIC POTENTIAL

Select the **Electric mode potential** — **Positive electric potential** (the default) or **Negative electric potential** to specify positive or negative 1 V on the boundary selection.

Circular Port Reference Axis

The **Circular Port Reference Axis** is available only in 3D. When the **Type of port** is set to **Circular** under **Port Properties**, the **Circular Port Reference Axis** subnode is available from the context menu (right-click the **Port** parent node) or from the **Physics** toolbar, **Attributes** menu. Two points are used to define the orientation of fields on a port boundary. If there are more than two points on the selection list, the first and last points are used. For the fundamental TE₁₁ mode, the direction of the reference axis corresponds to the polarization of the electric field at the port center.

Diffraction Order

The **Diffraction Order** port is available in 3D and 2D. When the **Type of Port** is set to **Periodic** under **Port Properties**, this subnode is available from the context menu (right-click the **Port** parent node) or from the **Physics** toolbar, **Attributes** menu.

Use the **Diffraction Order** port to define diffraction orders from a periodic structure. Normally a **Diffraction Order** node is added automatically during the **Periodic** port setup. Additional **Diffraction Order** ports subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

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 and numeric port names are also required for port sweeps and Touchstone file export.

The Diffraction Order port is a listener port feature.

PORT MODE SETTINGS

These settings define the diffracted plane wave.

Components

Select the **Components** setting for the port — **In-plane vector** (the default) or **Out-of-plane vector**. In 3D, when the diffraction order m and n are both set to 0, a third option is **Orthogonal polarization**, which represents a polarization orthogonal to the polarization of the incident wave.

Diffraction Order

Specify an integer constant or an integer parameter expression for the **Diffraction order** m (the default is 0) and in 3D n (the default is 0).

Note that **In-plane vector** and **Out-of-plane vector** are based on the plane of diffraction which is constructed with the diffraction wave vector and the outward normal vector of the port boundary. The diffraction wave vector is defined by

$$\mathbf{k}_{\text{diffraction,parallel}} = \mathbf{k}_F + M\mathbf{G}_1 + N\mathbf{G}_2$$

$$\mathbf{k}_{\text{diffraction}} = \mathbf{k}_{\text{diffraction,parallel}} - \mathbf{n}k_{\text{diffraction,perpendicular}}$$

$$k_{\text{diffraction,perpendicular}} = \sqrt{k^2 - k_{\text{diffraction,parallel}}^2}$$

where M and N are diffraction orders, $k \geq k_{\text{diffraction,parallel}}$, k is the magnitude of the wave vector and $k_{\text{diffraction,parallel}}$ is the magnitude of $\mathbf{k}_{\text{diffraction,parallel}}$. Reciprocal lattice vectors, \mathbf{G}_1 and \mathbf{G}_2 are defined from [Periodic Port Reference Point](#).

In-plane vector lies on the plane of diffraction while **Out-of-plane vector** is normal to the plane of diffraction.

For a 2D component, **In-plane vector** is available when the settings for the physics interface is set to either **In-plane vector** or **Three-component vector** under [components](#). **Out-of-plane vector** is available when the settings for the physics interface is set to either **Out-of-plane vector** or **Three-component vector** under [components](#).

Enter a value or expression for the **Mode phase** θ_{in} (SI unit: rad). The default is 0 radians. The **Mode phase** setting is further discussed for the [Port](#) feature.



Notice that the mode field defined for the Diffraction Order feature assumes isotropic material properties in the domain adjacent to the selected feature boundary.



- [S-Parameters and Ports](#)
- [S-Parameter Variables](#)



Plasmonic Wire Grating: Application Library path **RF_Module/Tutorials/plasmonic_wire_grating**

Orthogonal Polarization

The **Orthogonal Polarization** port is available in 3D and 2D. When the **Type of Port** is set to **Periodic** under [Port Properties](#), this subnode is available from the context menu (right-click the [Port](#) parent node) or from the **Physics** toolbar, **Attributes** menu.

Use the **Orthogonal Polarization** port to define port with a mode that is orthogonal to the mode of the parent **Periodic Port**. Normally a **Orthogonal Polarization** node is added automatically during the **Periodic** port setup, but the **Orthogonal Polarization** port subnode is also available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. Only one **Orthogonal Polarization** node can be added per parent **Periodic Port**.

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 and numeric port names are also required for port sweeps and Touchstone file export.

The **Orthogonal Polarization** port is a listener port feature.

PORT MODE SETTINGS

Diffraction Order

The **Orthogonal Polarization** port represent a zero-order mode (same as assumed for the parent Periodic port). Thus, the **Diffraction order** settings are just for information and cannot be edited.

Enter a value or expression for the **Mode phase** θ_m (SI unit: rad). The default is 0 radians. The **Mode phase** setting is further discussed for the [Port](#) feature.



Notice that the mode field defined for the **Orthogonal Polarization** feature assumes isotropic material properties in the domain adjacent to the selected feature boundary.



- [S-Parameters and Ports](#)
 - [S-Parameter Variables](#)
-

Periodic Port Reference Point

The **Periodic Port Reference Point** subnode is available only in 3D. When the **Type of Port** is set to **Periodic** under **Port Properties**, this subnode is available from the context menu (right-click the **Port** parent node) or from the **Physics** toolbar, **Attributes** menu.

The **Periodic Port Reference Point** is used to uniquely identify two primitive unit cell vectors, \mathbf{a}_1 and \mathbf{a}_2 , and two reciprocal lattice vectors, \mathbf{G}_1 and \mathbf{G}_2 . These reciprocal vectors are defined in terms of the unit cell vectors, \mathbf{a}_1 and \mathbf{a}_2 , tangent to the edges shared between the port and the adjacent periodic boundary conditions. \mathbf{G}_1 and \mathbf{G}_2 are defined by the relation

$$\frac{\mathbf{a}_1 \times \mathbf{a}_2}{|\mathbf{a}_1 \times \mathbf{a}_2|} = \mathbf{n}$$
$$\mathbf{G}_1 = 2\pi \frac{\mathbf{a}_2 \times \mathbf{n}}{\mathbf{a}_1 \cdot \mathbf{a}_2 \times \mathbf{n}} \quad \text{and} \quad \mathbf{G}_2 = 2\pi \frac{\mathbf{n} \times \mathbf{a}_1}{\mathbf{a}_1 \cdot \mathbf{a}_2 \times \mathbf{n}}$$

where \mathbf{n} is the outward unit normal vector to the port boundary. If there are multiple points defined in the selection list, only the last point is used.

POINT SELECTION

The primitive unit cell vectors, \mathbf{a}_1 and \mathbf{a}_2 are defined from two edges sharing the **Periodic Port Reference Point** on a port boundary. The two vectors can have unequal lengths and are not necessarily orthogonal. They start from the **Periodic Port Reference Point**.

For listener (passive, observation, and not excited) ports, if the outward normal vector on the listener port boundary is opposite to that of the source port, the listener port reference point needs to be mirrored from the source port reference point based on the center coordinate of the model domain. For example, if the source port reference point is at $\{-1,-1,1\}$ in a cubic domain around the origin, the mirrored listener port reference point is $\{1,1,-1\}$. In this case, if the **Azimuth angle of incidence** at the source port boundary is α_2 , the **Azimuth angle of incidence** at the listener port boundary $\pi/2 - \alpha_2$ and the signs of the diffraction order on the source and listener ports are opposite. See also **Periodic** for the angle definition.

If the lattice vectors are collinear with two Cartesian axes, then the lattice vectors can be defined without the **Periodic Port Reference Point**. For the port where \mathbf{n} points along a positive Cartesian direction, \mathbf{a}_1 and \mathbf{a}_2 are also assigned to point along positive Cartesian directions. Conversely, for the port where \mathbf{n} points along a negative

Cartesian direction, \mathbf{a}_1 and \mathbf{a}_2 are assigned to point along negative Cartesian directions. The condition $\mathbf{a}_1 \times \mathbf{a}_2 \parallel \mathbf{n}$ is true on both ports. For example, if $\mathbf{n} = \mathbf{z}$, then $\mathbf{a}_1/|\mathbf{a}_1| = \mathbf{x}$ and $\mathbf{a}_2/|\mathbf{a}_2| = \mathbf{y}$ and if $\mathbf{n} = -\mathbf{z}$, then $\mathbf{a}_1/|\mathbf{a}_1| = -\mathbf{x}$ and $\mathbf{a}_2/|\mathbf{a}_2| = -\mathbf{y}$.



Plasmonic Wire Grating: Application Library path **RF_Module/Tutorials/plasmonic_wire_grating**

Lumped Port

Use the **Lumped Port** node to apply a voltage or current excitation of a model or to connect to a circuit. A lumped port is a simplification of the port boundary condition.

A **Lumped Port** condition can only be applied on boundaries that extend between two metallic boundaries — that is, boundaries where **Perfect Electric Conductor**, **Impedance Boundary**, or **Transition Boundary** (Electromagnetic Waves, Frequency Domain interface only) conditions apply — separated by a distance much smaller than the wavelength.

LUMPED PORT PROPERTIES

Enter a unique **Lumped port name**. It is recommended to use a numeric name as it is used to define the elements of the S-parameter matrix and numeric port names are also required for port sweeps and Touchstone file export (for the Electromagnetic Waves, Frequency Domain interface).

Type of Lumped Port

Select a **Type of lumped port** — **Coaxial**, **Multielement uniform**, **Uniform**, **User defined**, or **Via**.

Select **User defined** for nonuniform ports, for example, a curved port and enter values or expressions in the fields — **Height of lumped port** h_{port} (SI unit: m), **Width of lumped port** w_{port} (SI unit: m), and **Direction between lumped port terminals** \mathbf{a}_h . In 2D axisymmetry, **Coaxial** does not support a nonzero azimuthal mode number. The **Azimuthal mode number** should be set to zero in the **Out-of-Plane Wave Number** section

in the settings for [The Electromagnetic Waves, Frequency Domain Interface](#). When it is modeled in 3D, its geometry has to be a full coaxial shape.



Notice that the input field for **Direction between lumped port terminals** a_h is not shown in 2D, when **Electric field components solved for** is set to **Out-of-plane vector** in the **Components** section for the physics interface. In this case the **Direction between lumped port terminals** a_h is defined to be in the out-of-plane direction.

Select **Multielement uniform** for multi-excitation or -termination of, for example, a coplanar waveguide port or a differential port. The direction of the field in each sub-element of the **Multielement uniform** lumped port is defined by the subnodes, **Uniform element**.

Select **Via** for excitation or termination of a cylindrical shape of a structure used in a metalized via that is a plated through hole.

Terminal Type

Select a **Terminal type** — a **Cable** port for a voltage driven transmission line and S-parameter calculation, a **Current** driven port, or a **Circuit** port.

For **Cable** select **On** or **Off** from the **Wave excitation at this port** list to set whether it is an input or a listener port. If **On** is selected, enter a **Voltage** V_0 (SI unit: V), and **Port phase** θ_{in} (SI unit: rad).

Note it is only possible to excite one **Cable** port at a time if the purpose is to compute S-parameters. In other cases, for example, when studying microwave heating, more than one input might be wanted, but the S-parameter variables cannot be correctly computed so if several ports are excited, the S-parameter output is turned off.

For the Electromagnetic Waves, Frequency Domain and Microwave Heating interfaces, the [Port Sweep Settings](#) cycles through the ports, computes the entire S-matrix, and exports it to a Touchstone file. When using port sweeps, the local setting for **Wave excitation at this port** is overridden by the solver so only one port at a time is excited.

SETTINGS

No entry is required if a **Circuit** terminal type is selected above.

- For a **Cable** terminal type enter the **Characteristic impedance** Z_{ref} (SI unit: Ω).
- For a **Current** terminal type enter a **Terminal current** I_0 (SI unit: A).

CALCULATOR

Enter a **Relative permittivity** (default is 1) for the dielectric material between the coaxial inner and outer conductor and then click the **Compute Coaxial Line Impedance** button to compute the impedance for the coaxial lumped port. The result is displayed in the **Messages** window.

CALCULATE S-PARAMETER

The **Calculate S-parameter** check box needs to be activated for S-parameter calculation with the Electromagnetic Waves, Transient and Electromagnetic Waves, Time Explicit interfaces, while the **Cable** port in the Electromagnetic Wave, Frequency Domain calculates S-parameters automatically.

VOLTAGE SOURCE TYPE

When **Calculate S-parameter** is checked, select **Voltage source type** from the list (default **Modulated Gaussian pulse**). The **Modulated Gaussian pulse** is defined as

$$\frac{2f_0}{\sqrt{2\pi}} e^{-\frac{(x-2/f_0)^2}{2/(2f_0)^2}} \cdot \sin(2\pi \cdot (1 + \eta_f) \cdot f_0)$$

where the **Center frequency** f_0 defines the location as $2/f_0$, the standard deviation as $1/(2f_0)$, and the modulation frequency f_0 . η_f is the upshift ratio for the sinusoidal modulation frequency.

The modulating sinusoidal function can be slightly shifted from the center frequency f_0 by a factor of $1+\eta_f$ to improve the frequency responses.

CENTER FREQUENCY

When **Calculate S-parameter** is checked, enter a **Center frequency** f_0 for the input modulated Gaussian pulse (SI unit: Hz).

MODULATION FREQUENCY UPSHIFT RATIO

When **Calculate S-parameter** is checked, enter a **Modulation frequency upshift ratio** η_f for the input modulated Gaussian pulse.



- [S-Parameters and Ports](#)
 - [Lumped Ports with Voltage Input](#)
-



As a multiport device example, *Branch-Line Coupler*: Application Library path **RF_Module/Couplers_and_Power_Dividers/branch_line_coupler**

For example of how to use the **Multielement uniform** lumped port, *Coplanar Waveguide Bandpass Filter*: Application Library path **RF_Module/Filters/cpw_bandpass_filter**

Lumped Element

Use a **Lumped Element** node to mimic the insertion of a capacitor, inductor, or general impedance between two metallic boundaries. A **Lumped Element** condition is a passive lumped port boundary condition which cannot be used as a source. Unlike a **Lumped Port**, it does not generate S-parameters. The sign of the current and power of a **Lumped Element** is opposite to that of a **Lumped Port**.

It can only be applied on boundaries that extend between two metallic boundaries — that is, boundaries where **Perfect Electric Conductor**, **Impedance Boundary**, or **Transition Boundary** (Electromagnetic Waves, Frequency Domain interface only) conditions apply — separated by a distance much smaller than the wavelength.

LUMPED ELEMENT PROPERTIES

Enter a unique **Lumped element name**. See [Lumped Port](#) for the rest of the settings.

SETTINGS

Select a **Lumped element device** — **User defined** (the default), **Inductor**, **Capacitor**, **Parallel LC**, **Series LC**, **Parallel RLC**, or **Series RLC**. **Inductor**, **Capacitor**, **Parallel LC**, **Series LC**, **Parallel RLC**, and **Series RLC** are available only in the frequency domain study type.

- For **User defined** enter a **Lumped element impedance** Z_{element} (SI unit: Ω). The default is 50 Ω .
- For **Inductor**, **Parallel LC**, **Series LC**, **Parallel RLC**, or **Series RLC** enter a **Lumped element inductance** L_{element} (SI unit: H). The default is 1 nH.
- For **Capacitor**, **Parallel LC**, **Series LC**, **Parallel RLC**, or **Series RLC** enter a **Lumped element capacitance** C_{element} (SI unit: F). The default is 1 pF.
- For **Parallel RLC**, or **Series RLC** enter a **Lumped element resistance** R_{element} (SI unit: Ω). The default is 50 Ω .

CALCULATOR

Enter a **Relative permittivity** (default is 1) for the dielectric material between the coaxial inner and outer conductor and then click the **Compute Coaxial Line Impedance** button to compute the impedance for the coaxial lumped element. The result is displayed in the **Messages** window.



SMA Connectorized Wilkinson Power Divider: Application Library path
RF_Module/Couplers_and_Power_Dividers/wilkinson_power_divider

Uniform Element

The **Uniform Element** is available in 3D. When the **Type of lumped port** is set to **Multielement uniform** under [Lumped Port Properties](#), this subnode is available from the context menu (right-click the [Lumped Port](#) parent node) or from the **Physics** toolbar, **Attributes** menu.

PORT PROPERTIES

Enter a unique **Uniform element name**. Enter values or expressions in the fields — **Direction between uniform element terminals** \mathbf{a}_h . The **Direction between uniform element terminals** defines the electric potential polarity, as \mathbf{a}_h points in the direction of the electric field.

Electric Field

The **Electric Field** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{n} \times \mathbf{E}_0$$

specifies the tangential component of the electric field. It should in general not be used to excite a model. Consider using the [Port](#), [Lumped Port](#), or [Scattering Boundary Condition](#) instead. It is provided mainly for completeness and for advanced users who can recognize the special modeling situations when it is appropriate to use. The commonly used special case of zero tangential electric field is described in the [Perfect Electric Conductor](#) section.

ELECTRIC FIELD

Enter the value or expression for the components of the **Electric field** \mathbf{E}_0 (SI unit: V/m).

CONSTRAINT SETTINGS

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

Magnetic Field

The **Magnetic Field** node adds a boundary condition for specifying the tangential component of the magnetic field at the boundary:

$$\mathbf{n} \times \mathbf{H} = \mathbf{n} \times \mathbf{H}_0$$

MAGNETIC FIELD

Enter the value or expression for the components of the **Magnetic field** \mathbf{H}_0 (SI unit: A/m).

Scattering Boundary Condition

Use the **Scattering Boundary Condition** to make a boundary transparent for a scattered wave. The boundary condition is also transparent for an incoming plane wave. The scattered (outgoing) wave types for which the boundary condition is perfectly transparent are

$\mathbf{E} = \mathbf{E}_{sc} e^{-jk(\mathbf{n} \cdot \mathbf{r})} + \mathbf{E}_0 e^{-jk(\mathbf{k} \cdot \mathbf{r})}$	Plane scattered wave
$\mathbf{E} = \mathbf{E}_{sc} \frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{\sqrt{r}} + \mathbf{E}_0 e^{-jk(\mathbf{k} \cdot \mathbf{r})}$	Cylindrical scattered wave
$\mathbf{E} = \mathbf{E}_{sc} \frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{r_s} + \mathbf{E}_0 e^{-jk(\mathbf{k} \cdot \mathbf{r})}$	Spherical scattered wave

The field \mathbf{E}_0 is the incident plane wave that travels in the direction \mathbf{k} . The boundary condition is transparent for incoming (but not outgoing) plane waves with any angle of incidence. In addition, to an incident plane wave, \mathbf{E}_0 can also be the electric field distribution for a Gaussian beam that propagates in the direction \mathbf{k} .

If there is an incident field, a [Reference Point](#) subnode can be added by right-clicking the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. The [Reference Point](#) subnode redefines the incident field to be expressed as

$$\mathbf{E}_0 e^{-jk(\mathbf{k} \cdot (\mathbf{r} - \mathbf{r}_{ref}))},$$

where \mathbf{r}_{ref} is a reference point determined as the average point from the point selection in the [Reference Point](#) subnode.

The boundary is only perfectly transparent for scattered (outgoing) waves of the selected type at normal incidence to the boundary. That is, a plane wave at oblique incidence is partially reflected and so is a cylindrical wave or spherical wave unless the wave fronts are parallel to the boundary. For the Electromagnetic Waves, Frequency Domain interface, the Perfectly Matched Layer feature is available as a general way of modeling an open boundary.

- For cylindrical waves, specify around which cylinder axis the waves are cylindrical. Do this by specifying one point at the cylinder axis and the axis direction.
- For spherical waves, specify the center of the sphere around which the wave is spherical.

The domain material adjacent to the boundary where the Scattering Boundary Condition is applied can be lossy.

If the problem is solved for the eigenfrequency or the scattered field, the boundary condition does not include the incident wave.

$$\mathbf{E}_{sc} = \mathbf{E}_{sc} e^{-jk(\mathbf{n} \cdot \mathbf{r})} \quad \text{Plane scattered wave}$$

$$\mathbf{E}_{sc} = \mathbf{E}_{sc} \frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{\sqrt{r}} \quad \text{Cylindrical scattered wave}$$

$$\mathbf{E}_{sc} = \mathbf{E}_{sc} \frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{r_s} \quad \text{Spherical scattered wave}$$

SCATTERING BOUNDARY CONDITION

Select an **Incident field** — **No incident field** (the default), **Wave given by E field**, **Wave given by H field**, or **Gaussian beam**. Enter the expressions for the components for the **Incident electric field** \mathbf{E}_0 , **Incident magnetic field** \mathbf{H}_0 , or **Gaussian beam electric field amplitude** \mathbf{E}_{g0} .

If **Incident field** is set to **Gaussian beam**, edit the **Beam radius** w_0 (SI unit: m) and the **Distance to focal plane** p_0 (SI unit: m). The default values are $((10*2)*\pi)/\text{emw.k0}$ and 0 m, respectively. The optical axis for the Gaussian beam is defined by a line including a reference point on the feature selection with a direction specified by the

Incident wave direction (see below). By default, the reference point is the average position for the feature selection. However, by adding a [Reference Point](#) subnode any available point (or the average of several selected points) on the feature selection can be used as the reference point. The focal plane for the Gaussian beam is located the **Distance to focal plane** p_0 from the reference point in the **Incident wave direction**.

If the **Incident field** is not set to **No incident field**, edit the **Incident wave direction** \mathbf{k}_{dir} for the vector coordinates. The default direction is in the opposite direction to the boundary normal. For 2D axisymmetry, the **Incident wave direction** \mathbf{k}_{dir} should be parallel or anti-parallel to the symmetry axis.

Select a **Scattered wave type** for which the boundary is absorbing — **Plane wave** (the default), **Spherical wave**, or **Cylindrical wave**.

- For the Electromagnetic Waves, Frequency Domain interface, select an **Order** — **First order** (the default) or **Second order**.
- For **Cylindrical wave** also enter coordinates for the **Source point** \mathbf{r}_0 (SI unit: m) and **Source axis direction** \mathbf{r}_{axis} (dimensionless). For 2D the **Source axis direction** is assumed to be in the z direction, whereas in 2D axisymmetry it is assumed to be along the axis of rotation.
- For **Spherical wave** enter coordinates for the **Source point** \mathbf{r}_0 (SI unit: m).



For more information about the Gaussian beam theory, see [Gaussian Beams as Background Fields](#).



Conical Antenna: Application Library path **RF_Module/Antennas/conical_antenna**

INITIAL VALUES FOR INCIDENT WAVE

For the Electromagnetic Waves, Transient interface enter the components for the initial value of the **Magnetic vector potential** \mathbf{A}_0 (SI unit: Wb/m).

DISPERSION AND ABSORPTION

This section is only available for the Electromagnetic Waves, Transient interface. To display it, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.

Select the **Dispersion and absorption model** that will be used when calculating the wave number and attenuation constant for the incident and scattered waves — **Low loss approximation** (the default), or **High loss**. For **High loss** also enter a **Carrier frequency** f_0 (SI unit: Hz). The default is 1 GHz.

When the **Dispersion and absorption model** is set to **Low loss approximation** the refractive index is calculated from the relative permittivity and the relative permeability as

$$n = \sqrt{\epsilon_r \mu_r}.$$

Similarly, the absorption coefficient is calculated as

$$\gamma = \frac{1}{2} \sigma \sqrt{\frac{\mu_0 \mu_r}{\epsilon_0 \epsilon_r}} = \frac{1}{2} \sigma Z_c,$$

where Z_c is the characteristic impedance.



When the **Dispersion and absorption model** is set to **High loss**, the real and the imaginary parts of the complex refractive index is solved for from the real and the imaginary parts of the relative permittivity, using the relations

$$n^2 - \kappa^2 = \epsilon'_r \mu_r$$

and

$$2n\kappa = \epsilon''_r \mu_r = \frac{\sigma \mu_r}{\omega \epsilon_0}.$$

The absorption coefficient is then given by

$$\gamma = \frac{\omega}{c} \kappa.$$

Reference Point

The **Reference Point** subnode is available only when there is an available incident field defined in the parent node. Then this subnode is available from the context menu (right-click the **Scattering Boundary Condition** parent node) or from the **Physics** toolbar, **Attributes** menu.

The **Reference Point** subnode defines a reference position \mathbf{r}_{ref} that is calculated as the average position from the point selection in the **Reference Point** subnode.

In the parent node, the incident field is then defined using the reference position:

$$\mathbf{E}_0 e^{-jk(\mathbf{k} \cdot (\mathbf{r} - \mathbf{r}_{\text{ref}}))}.$$

POINT SELECTION

Select the points that should be used when calculating the reference position. The reference position is calculated as the average position of the selected points.

Impedance Boundary Condition

The **Impedance Boundary Condition**

$$\sqrt{\frac{\mu_0 \mu_r}{\epsilon_c}} \mathbf{n} \times \mathbf{H} + \mathbf{E} - (\mathbf{n} \cdot \mathbf{E}) \mathbf{n} = (\mathbf{n} \cdot \mathbf{E}_s) \mathbf{n} - \mathbf{E}_s$$

is used at boundaries where the field is known to penetrate only a short distance outside the boundary. This penetration is approximated by a boundary condition to avoid the need to include another domain in the model. Although the equation is identical to the one in the low-reflecting boundary condition, it has a different interpretation. The material properties are for the domain outside the boundary and not inside, as for low-reflecting boundaries. A requirement for this boundary condition to be a valid approximation is that the magnitude of the complex refractive index

$$N = \sqrt{\frac{\mu \epsilon_c}{\mu_1 \epsilon_1}}$$

where μ_1 and ϵ_1 are the material properties of the inner domain, is large; that is, $|N| \gg 1$.

The source electric field \mathbf{E}_s can be used to specify a source surface current on the boundary.

The [Surface Roughness](#) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

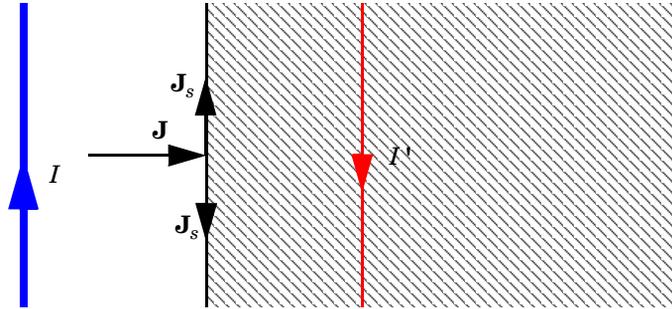


Figure 4-4: The impedance boundary condition is used on exterior boundaries representing the surface of a lossy domain. The shaded (lossy) region is not part of the model. The effective induced image currents are of reduced magnitude due to losses. Any current flowing into the boundary is perfectly balanced by induced surface currents as for the perfect electric conductor boundary condition. The tangential electric field is generally small but nonzero at the boundary.

IMPEDANCE BOUNDARY CONDITION

Select an **Electric displacement field model** — **Relative permittivity** (the default), **Refractive index**, **Loss tangent**, **loss angle**, **Loss tangent, dissipation factor**, **Dielectric loss**, **Drude-Lorentz dispersion model**, or **Debye dispersion model**. See the [Wave Equation](#), [Electric](#) node, [Electric Displacement Field](#) section, for all settings.

SOURCE ELECTRIC FIELD

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

Enter a **Source electric field** \mathbf{E}_s (SI unit: V/m). The default is 0 V/m.



Coaxial to Waveguide Coupling: Application Library path **RF_Module/Transmission_Lines_and_Waveguides/coaxial_waveguide_coupling**

Computing Q-Factors and Resonant Frequencies of Cavity Resonators: Application Library path **RF_Module/Verification_Examples/cavity_resonators**

Surface Current Density

The **Surface Current Density** boundary condition

$$-\mathbf{n} \times \mathbf{H} = \mathbf{J}_s$$

$$\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{J}_s$$

specifies a surface current density at both exterior and interior boundaries, respectively. The current density is specified as a three-dimensional vector, but because it needs to flow along the boundary surface, COMSOL Multiphysics projects it onto the boundary surface and neglects its normal component. This makes it easier to specify the current density and avoids unexpected results when a current density with a component normal to the surface is given.

For [Transition Boundary Condition](#), the **Surface Current Density** subnode as an one-sided surface current density is available from the context menu (right-click the Transition Boundary Condition parent node) or from the **Physics** toolbar, **Attributes** menu.

SURFACE CURRENT DENSITY

Enter values or expressions for the components of the **Surface current density** \mathbf{J}_{s0} (SI unit: A/m).

For the **Surface Current Density** subnode, select **Side** — **Upside** (the default) or **Downside** to define on which side the **Surface Current Density** is applied. The red arrow visualized on the selected boundaries always indicates the upside.

Surface Magnetic Current Density

The **Surface Magnetic Current Density** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{J}_{ms}$$

$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = -\mathbf{J}_{ms}$$

specifies a surface magnetic current density at both exterior and interior boundaries, respectively. The magnetic current density is specified as a three-dimensional vector, but because it needs to flow along the boundary surface, COMSOL Multiphysics projects it onto the boundary surface and neglects its normal component. This makes it easier to specify the magnetic current density and avoids unexpected results when a magnetic current density with a component normal to the surface is given.

SURFACE MAGNETIC CURRENT DENSITY

Enter values or expressions for the components of the **Surface magnetic current density** \mathbf{J}_{ms0} (SI unit: V/m).

CONSTRAINT SETTINGS

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

Surface Roughness

The **Surface Roughness** subnode is available from the context menu (right-click the [Impedance Boundary Condition](#) or [Transition Boundary Condition](#) parent node) or from the **Physics** toolbar, **Attributes** menu.

SURFACE ROUGHNESS

Select a **Surface roughness model** — **Sawtooth** (the default) or **Snowball**. For **Sawtooth**, enter a value or expression for the **Surface roughness** Δ_{RMS} (SI unit: m). For **Snowball**, enter a **Snowball hexagon cell area** A_{hex} (SI unit: m^2). Then enter values for the **Snowball radius** (SI unit: m) and **Number of snowballs** in the table, adding as many rows as you need.

Select the **Skin depth type** to be **Physics-controlled** (the default) or **User defined**. For **User defined**, enter the **Skin depth** δ (SI unit: m).

The **Surface Roughness** increases the impedance, and consequently it decreases the surface current density of [Impedance Boundary Condition](#) or [Transition Boundary Condition](#), proportional to the ratio between the impedance of a rough surface and that of a smooth surface:

SAWTOOTH MODEL

$$Z_{\text{rough}} = Z_{\text{smooth}} \cdot \left[1 + \frac{2}{\pi} \text{atan} \left((1.4) \left(\frac{\Delta_{\text{RMS}}}{\delta} \right)^2 \right) \right]$$

where Δ_{RMS} is the root mean square of the surface roughness, and δ is the skin depth of the material (Ref. 1).

SNOWBALL MODEL

$$Z_{\text{rough}} = Z_{\text{smooth}} \cdot \left[1 + \frac{3}{2} \sum_i \left(\frac{N_i \cdot 4\pi a_i^2}{A_{\text{hex}}} \right) \sqrt{1 + \frac{\delta}{a_i} + \frac{\delta^2}{a_i^2}} \right]$$

where A_{hex} is the hexagonal area of a unit cell, N_i is the number of snowballs, a_i is the radius of a snowball, and δ is the skip depth of the material (Ref. 2).

For Transition Boundary Condition, select **Side** — **Upside** (the default) or **Downside** to define on which side the **Surface Roughness** is applied. The red arrow visualized on the selected boundaries always indicates the upside.

References

1. E. Hammerstad, O. Jensen, “Accurate Models for Microstrip Computer-Aided Design”, *Microwave symposium Digest, 1980 IEEE MTT-S International*, pp. 407–409, May 1980
2. P.G. Huray, *The Foundation of Signal Integrity*, Wiley-IEEE Press, 2010

Transition Boundary Condition

The **Transition Boundary Condition** is used on interior boundaries to model a sheet of a medium that should be geometrically thin but does not have to be electrically thin. It represents a discontinuity in the tangential electric field. Mathematically it is described by a relation between the electric field discontinuity and the induced surface current density:

$$\mathbf{J}_{s1} = \frac{(Z_S \mathbf{E}_{t1} - Z_T \mathbf{E}_{t2})}{Z_S^2 - Z_T^2}$$

$$\mathbf{J}_{s2} = \frac{(Z_S \mathbf{E}_{t2} - Z_T \mathbf{E}_{t1})}{Z_S^2 - Z_T^2}$$

$$Z_S = \frac{-j\omega\mu}{k} \frac{1}{\tan(kd)}$$

$$Z_T = \frac{-j\omega\mu}{k} \frac{1}{\sin(kd)}$$

$$k = \omega\sqrt{(\epsilon + (\sigma/(j\omega)))\mu}$$

where indices 1 and 2 refer to the different sides of the layer. This feature is not available with the Electromagnetic Waves, Transient interface.



The **Transition Boundary Condition** is based on the assumption that the wave propagates in the normal direction in the thin layer. Thus, the wave could be incident in the normal direction or the wave could be refracted to propagate in a direction close to the normal direction. The latter condition is fulfilled for a good conductor.

The thickness of the layer should also be less than the radius of curvature for the boundary.

A consequence of the normal direction propagation assumption is that the **Transition Boundary Condition** is not compatible with mode analysis, as for mode analysis it is assumed that the wave predominantly propagates in the out-of-plane direction whereas the normal to the boundary is in an in-plane direction.

The [Surface Roughness](#) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

TRANSITION BOUNDARY CONDITION

Select an **Electric displacement field model** — **Relative permittivity** (the default), **Refractive index**, **Loss tangent**, **loss angle**, **Loss tangent**, **dissipation factor**, **Dielectric loss**, **Drude-Lorentz dispersion model**, or **Debye dispersion model**. See the [Wave Equation](#), [Electric](#) node, [Electric Displacement Field](#) section, for all settings.

Select the **Electrically thick layer** check box (unselected by default) to make the two domains adjacent to the boundary uncoupled. When the **Electrically thick layer** check box is unselected, enter a **Thickness** d (SI unit: m). The default is 0.01 m.

Periodic Condition

The **Periodic Condition** sets up a periodicity between the selected boundaries. The **Destination Selection** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

BOUNDARY SELECTION

The software automatically identifies the boundaries as either source boundaries or destination boundaries. This works fine for cases like opposing parallel boundaries. To

control the destination, add a **Destination Selection** subnode. By default it contains the selection that COMSOL Multiphysics has identified.

PERIODICITY SETTINGS

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

- **Continuity** to make the electric field periodic (equal on the source and destination),
- **Antiperiodicity** to make it antiperiodic, or
- **Floquet periodicity** ([The Electromagnetic Waves, Frequency Domain Interface](#) only) to use a Floquet periodicity (Bloch-Floquet periodicity).
 - For **Floquet periodicity** also enter the source for the **k-vector for Floquet periodicity**.
 - For **User defined** specify the components of the **k-vector for Floquet periodicity** \mathbf{k}_F (SI unit: rad/m).
 - For **From periodic port** the **k-vector for Floquet periodicity** \mathbf{k}_F is obtained from the **Periodic Port** settings.

CONSTRAINT SETTINGS

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

ORIENTATION OF SOURCE

For information about the **Orientation of Source** section, see [Orientation of Source and Destination](#) in the *COMSOL Multiphysics Reference Manual*.

	Periodic Boundary Conditions
	<ul style="list-style-type: none"> • <i>Fresnel Equations</i>: Application Library path RF_Module/Verification_Examples/fresnel_equations • <i>Plasmonic Wire Grating</i>: Application Library path: RF_Module/Tutorials/plasmonic_wire_grating

Magnetic Current

The **Magnetic Current** node specifies a magnetic line current along one or more edges. For a single **Magnetic Current** source, the electric field is orthogonal to both the line and the distance vector from the line to the field point. For 2D and 2D axisymmetric models the **Magnetic Current** node is applied to **Points**, representing magnetic currents directed out of the model plane. For 3D models, the **Magnetic Current** is applied to **Edges**.

MAGNETIC CURRENT

Enter a value for the **Magnetic current** I_m (SI unit: V).

Two-Port Network

Use the **Two-Port Network** node to characterize the response of a two-port network system such as reflection and transmission using S-parameters.

A **Two-Port Network** can only be applied on boundaries that extend between two metallic boundaries — that is, boundaries where **Perfect Electric Conductor**, **Impedance Boundary**, or **Transition Boundary** conditions apply — separated by a distance much smaller than the wavelength.

TWO-PORT NETWORK PROPERTIES

Type of Port

Select a **Type of Port** — **Coaxial** or **Uniform**.

Coaxial does not support nonzero azimuthal mode number. The **Azimuthal mode number** in the **Physics interface** should be defined as zero.

Type of S-parameter Definition

Select a **Type of S-parameter Definition** — **Matrix** or **Touchstone file**.

Matrix defines the S-parameter by a constant matrix input. **Touchstone file** imports a two-port Touchstone file to characterize the two-port boundaries as a function of frequency.

SETTINGS

Enter the **Characteristic impedance** Z_{ref} (SI unit: Ω) for **Matrix**.

INTERPOLATION AND EXTRAPOLATION

Select **Interpolation** and **Extrapolation** types to obtain S-parameter values from the imported Touchstone file corresponding to the simulation frequencies.

Interpolation

Select a **Interpolation** — **Nearest neighbor**, **Linear**, **Piecewise cubic**, or **Cubic spline** (the default).

The S-parameter values are interpolated within the frequency range specified in the Touchstone file.

Extrapolation

Select a **Extrapolation** — **Constant** (the default), **Linear**, **Nearest function**, or **Specific value**.

The S-parameter values are extrapolated outside the frequency range specified in the Touchstone file.

Two-Port Network Port

A pair of **Two-Port Network Port** subnodes is added by default to the **Two-Port Network** node and is used to select boundaries corresponding to port 1 and port 2 in the S-parameter input, respectively.

Mixed Mode S-Parameters

To calculate the mixed mode S-parameters of a four-port network, add a **Mixed Mode S-parameters** global feature and specify port names for each balanced pair. The mixed mode S-parameter calculation also requires to perform a parametric port sweep.

BALANCED PORT

Specify **Port name for port a in balanced pair 1**, **Port name for port b in balanced pair 1**, **Port name for port c in balanced pair 2**, and **Port name for port d in balanced pair 2**. These must be matched to the port names specified in the (lumped) port features of the four-port system.

Edge Current

The **Edge Current** node specifies an electric line current along one or more edges.

EDGE CURRENT

Enter an **Edge current** I_0 (SI unit: A).

Electric Point Dipole

Add **Electric Point Dipole** nodes to 3D and 2D models. This represents the limiting case of when the length d of a current filament carrying uniform current I approaches zero while maintaining the product between I and d . The dipole moment is a vector entity with the positive direction set by the current flow.

DIPOLE SPECIFICATION

Select a **Dipole specification** — **Magnitude and direction** or **Dipole moment**.

DIPOLE PARAMETERS

Based on the **Dipole specification** selection:

- For **Magnitude and direction** enter coordinates for the **Electric current dipole moment direction** \mathbf{n}_p and **Electric current dipole moment, magnitude** p (SI unit: A·m).
- For **Dipole moment** enter coordinates for the **Electric current dipole moment** \mathbf{p} (SI unit: A·m).

Magnetic Point Dipole

Add a **Magnetic Point Dipole** to 3D and 2D models. The point dipole source represents a small circular current loop I in the limit of zero loop area a at a fixed product $I \cdot a$.

DIPOLE SPECIFICATION

Select a **Dipole specification** — **Magnitude and direction** or **Dipole moment**.

DIPOLE PARAMETERS

Based on the **Dipole specification** selection:

- For **Magnitude and direction** enter coordinates for the **Magnetic dipole moment direction** \mathbf{n}_m and **Magnetic dipole moment, magnitude** m (SI unit: $\text{m}^2 \cdot \text{A}$).
- For **Dipole moment** enter coordinates for the **Magnetic dipole moment** \mathbf{m} (SI unit: $\text{m}^2 \cdot \text{A}$).

Line Current (Out-of-Plane)

Add a **Line Current (Out-of-Plane)** node to 2D or 2D axisymmetric models. This specifies a line current out of the modeling plane. In axially symmetric geometries this is the rotational direction, in 2D geometries it is the z direction.

LINE CURRENT (OUT-OF-PLANE)

Enter an **Out-of-plane current** I_0 (SI unit: A).

Archie's Law

This subfeature is available only when **Archie's law** is selected as the **Electrical conductivity** material parameter in the parent feature (for example, the **Wave Equation**, **Electric** node). Then the subnodes are made available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu.

Use the **Archie's Law** subnode to provide an electrical conductivity computed using Archie's Law. This subnode can be used to model nonconductive effective medium saturated (or variably saturated) by conductive liquids, using the relation:

$$\sigma = s_L^n \epsilon_p^m \sigma_L$$



Archie's Law Theory

CONDUCTION CURRENTS

By default, the **Electrical conductivity** σ_L (SI unit: S/m) for the fluid is defined **From material**. This uses the value of the conductivity of the material domain.

For **User defined** enter a value or expression. If another type of temperature dependence is used other than a linear temperature relation, enter any expression for the conductivity as a function of temperature.

Enter these dimensionless parameters as needed:

- **Cementation exponent** m
- **Saturation exponent** n

- **Fluid saturation** S_L
- **Porosity** ε_p to set up the volume fraction of the fluid.

Effective Medium

This subfeature is available only when **Effective medium** is selected as the material parameter (for example, **Relative permeability** or **Relative permittivity**) in the parent feature node when it is available with the physics interface (for example, the **Wave Equation, Electric** node). Then the subnodes are made available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu.

Use the **Effective Medium** subfeature to specify the material properties of a domain consisting of a porous medium using a mixture model. Depending on the specific physics interface being used, the subfeature can be used to provide a mixture model for the electric conductivity σ , the relative dielectric permittivity ε_r , or the relative magnetic permeability μ_r .

EFFECTIVE MEDIUM

This section is always available and is used to define the mixture model for the domain.

Select the **Number of materials** (up to 5) to be included in the mixture model.

For each material (**Material 1**, **Material 2**, and so on), select either **Domain material**, to use the material specified for the domain, or one of the other materials specified in the **Materials** node. For each material, enter a **Volume fraction** θ_1 , θ_2 , and so on.

The Volume fractions specified for the materials should be fractional (between 0 and 1) and should add to 1 in normal cases.



The availability of the **Effective Electrical Conductivity**, **Effective Relative Permittivity**, and **Effective Relative Permeability** sections depend on the material properties used in the physics interface. In addition, these sections are only active if **Effective medium** is selected in the corresponding material property for the parent feature node.

EFFECTIVE ELECTRICAL CONDUCTIVITY, EFFECTIVE RELATIVE PERMITTIVITY, OR EFFECTIVE RELATIVE PERMEABILITY

Select the averaging method to use in the mixture model between the **Volume average** of the material property (for example, **conductivity** or **permittivity**), the volume average of its inverse (for example, the **resistivity**), or the **Power law**. For each material, specify

either **From material**, to take the value from the corresponding material specified in the **Effective Medium** section, or **User defined** to manually input a value.



Effective Conductivity in Effective Media and Mixtures

The Electromagnetic Waves, Transient Interface

The **Electromagnetic Waves, Transient (temw)** interface () , found under the **Radio Frequency** branch () when adding a physics interface, is used to solve a time-domain wave equation for the magnetic vector potential. The sources can be in the form of point dipoles, line currents, or incident fields on boundaries or domains. It is primarily used to model electromagnetic wave propagation in different media and structures when a time-domain solution is required — for example, for non-sinusoidal waveforms or for nonlinear media. Typical applications involve the propagation of electromagnetic pulses.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Wave Equation, Electric, Perfect Electric Conductor**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and mass sources. You can also right-click **Electromagnetic Waves, Transient** to select physics features from the context menu.

Except where indicated, most of the settings are the same as for [The Electromagnetic Waves, Frequency Domain 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 `temw`.

COMPONENTS

This section is available for 2D and 2D axisymmetric components.

Select the **Electric field components solved for**. Select:

- **Three-component vector** (the default) to solve using a full three-component vector for the electric field \mathbf{E} .

- **Out-of-plane vector** to solve for the electric field vector component perpendicular to the modeling plane, assuming that there is no electric field in the plane.
- **In-plane vector** to solve for the electric field vector components in the modeling plane assuming that there is no electric field perpendicular to the plane.

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Magnetic vector potential A** . The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

Select the shape order for the **Magnetic vector potential** dependent variable — **Linear**, **Quadratic** (the default), or **Cubic**. For more information about the **Discretization** section, see [Settings for the Discretization Sections](#) in the *COMSOL Multiphysics Reference Manual*.



- [Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Transient Interface](#)
- [Theory for the Electromagnetic Waves Interfaces](#)



Transient Modeling of a Coaxial Cable: Application Library path
RF_Module/Verification_Examples/coaxial_cable_transient

Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Transient Interface

The [Electromagnetic Waves, Transient Interface](#) shares most of its nodes with [The Electromagnetic Waves, Frequency Domain Interface](#).

The domain, boundary, edge, point, and pair 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.

DOMAIN

These nodes are unique for this physics interface and described in this section:

- [Wave Equation, Electric](#)
- [Initial Values](#)
- [Drude-Lorentz Polarization](#)
- [Far-Field Domain](#)
- [Far-Field Calculation](#)

BOUNDARY CONDITIONS

With no surface currents present the boundary conditions

$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{0}$$

$$\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$

need to be fulfilled. Depending on the field being solved for, it is necessary to analyze these conditions differently. When solving for \mathbf{A} , the first condition can be formulated in the following way.

$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{n}_2 \times \left(\frac{\partial \mathbf{A}_2}{\partial t} - \frac{\partial \mathbf{A}_1}{\partial t} \right) = \frac{\partial}{\partial t} (\mathbf{n}_2 \times (\mathbf{A}_2 - \mathbf{A}_1))$$

The tangential component of the magnetic vector potential is always continuous and thus the first condition is fulfilled. The second condition is equivalent to the natural boundary condition.

$$-\mathbf{n} \times (\mu_r^{-1} \nabla \times \mathbf{A}_1 - \mu_r^{-1} \nabla \times \mathbf{A}_2) = -\mathbf{n} \times \mu_r^{-1} (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$

and is therefore also fulfilled.

These nodes and subnodes are available and described for the Electromagnetic Waves, Frequency Domain interface (listed in alphabetical order):

- [Archie's Law](#)
- [Lumped Port](#)
- [Magnetic Field](#)
- [Perfect Electric Conductor](#)
- [Perfect Magnetic Conductor](#)
- [Periodic Condition](#)
- [Effective Medium](#)
- [Scattering Boundary Condition](#)
- [Surface Current Density](#)

EDGE, POINT, AND PAIR

These edge, point, and pair nodes are available and described for the Electromagnetic Waves, Frequency Domain interface (listed in alphabetical order):

- [Edge Current](#)
- [Electric Point Dipole](#) (2D and 3D components)
- [Line Current \(Out-of-Plane\)](#) (2D and 2D axisymmetric components)
- [Magnetic Point Dipole](#) (2D and 3D components)
- [Perfect Electric Conductor](#)
- [Perfect Magnetic Conductor](#)
- [Surface Current Density](#)



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](#) 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.

Wave Equation, Electric

The **Wave Equation, Electric** node is the main node for the Electromagnetic Waves, Transient interface. The governing equation can be written in the form

$$\mu_0 \sigma \frac{\partial \mathbf{A}}{\partial t} + \mu_0 \epsilon_0 \frac{\partial}{\partial t} \left(\epsilon_r \frac{\partial \mathbf{A}}{\partial t} \right) + \nabla \times (\mu_r^{-1} \nabla \times \mathbf{A}) = 0$$

for transient problems with the constitutive relations $\mathbf{B} = \mu_0 \mu_r \mathbf{H}$ and $\mathbf{D} = \epsilon_0 \epsilon_r \mathbf{E}$. Other constitutive relations can also be handled for transient problems. The **Divergence Constraint** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

ELECTRIC DISPLACEMENT FIELD

Select an **Electric displacement field model** — **Relative permittivity** (the default), **Refractive index**, **Polarization**, **Remanent electric displacement**, or **Drude-Lorentz dispersion model**.

Relative Permittivity

When **Relative permittivity** is selected, the default **Relative permittivity** ϵ_r (dimensionless) takes values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. If **Effective medium** is selected, the **Effective medium** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Refractive Index

When **Refractive index** is selected, the default **Refractive index** n (dimensionless) takes the value **From material**. To specify the refractive index and assume a relative permeability of unity and zero conductivity, for one or both of the options, select **User defined** then choose **Isotropic**, **Diagonal**, **Symmetric**, or **Full**. Enter values or expressions in the field or matrix.



Notice that only the real part of the refractive index is used for the transient formulation.

Polarization

For **Polarization** enter coordinates for the **Polarization** \mathbf{P} (SI unit: C/m²).

Remanent Electric Displacement

For **Remanent electric displacement** enter coordinates for the **Remanent electric displacement** \mathbf{D}_r (SI unit: C/m²). Then select **User defined** or **From Material** as above for the **Relative permittivity** ϵ_r .

Drude-Lorentz Dispersion Model

For **Drude-Lorentz Dispersion Model** select **User defined** or **From Material** for the **Relative permittivity**, **high-frequency** ϵ_∞ and enter a value for the **Plasma frequency** ω_p (SI unit: rad/s).

When **Drude-Lorentz Dispersion Model** is selected, the **Drude-Lorentz Polarization** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. Each **Drude-Lorentz Polarization** subnode adds another polarization term \mathbf{P}_n to the electric displacement field \mathbf{D} , defined by

$$\mathbf{D} = \epsilon_0 \epsilon_\infty \mathbf{E} + \sum_{n=1}^N \mathbf{P}_n,$$

where the polarization is the solution to the ordinary differential equation

$$\left(\frac{\partial^2}{\partial t^2} + \Gamma_n \frac{\partial}{\partial t} + \omega_n^2 \right) \mathbf{P}_n = \epsilon_0 f_n \omega_p^2 \mathbf{E}.$$

For more information, see the [Drude-Lorentz Polarization](#) feature.

MAGNETIC FIELD

This section is available if **Relative permittivity**, **Polarization**, or **Remanent electric displacement** are chosen as the **Electric displacement field model**.

Select the **Constitutive relation** — **Relative permeability** (the default), **Remanent flux density**, or **Magnetization**.

Relative Permeability

For **Relative permeability** the relative permeability μ_r uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix. If **Effective medium** is selected, the **Effective medium** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Remanent Flux Density

For **Remanent flux density** the relative permeability μ_r uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix. Then enter coordinates for the **Remanent flux density** \mathbf{B}_r (SI unit: T).

Magnetization

For **Magnetization** enter coordinates for \mathbf{M} (SI unit: A/m).

CONDUCTION CURRENT

This section is available if **Relative permittivity**, **Polarization**, or **Remanent electric displacement** are chosen as the **Electric displacement field model**.

By default, the **Electrical conductivity** σ (SI unit: S/m) uses values **From material**.

- For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the current and enter values or expressions in the field or matrix.

- For **Linearized resistivity** the default values for the **Reference temperature** T_{ref} (SI unit: K), **Resistivity temperature coefficient** α (SI unit: 1/K), and **Reference resistivity** ρ_0 (SI unit: Ωm) use values **From material**. For **User defined** enter other values or expressions for any of these variables.
- If **Effective medium** is selected, the **Effective medium** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.
- If **Archie's Law** is selected, the **Archie's Law** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Initial Values

The **Initial Values** node adds an initial value for the magnetic vector potential and its time derivative that serves as initial conditions for the transient simulation.

INITIAL VALUES

Enter values or expressions for the initial values of the components of the magnetic vector potential \mathbf{A} (SI unit: Wb/m) and its time derivative $\partial\mathbf{A}/\partial t$ (SI unit: V/m). The default values are 0 Wb/m and 0 V/m, respectively.

Drude-Lorentz Polarization

This subfeature is available only when **Drude-Lorentz Dispersion Model** is selected as the **Electric displacement field model** in the [Wave Equation, Electric](#) feature node. Then the subnodes are made available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu.

Each **Drude-Lorentz Polarization** subnode adds another polarization term \mathbf{P}_n to the electric displacement field \mathbf{D} , defined by

$$\mathbf{D} = \epsilon_0\epsilon_\infty\mathbf{E} + \sum_{n=1}^N \mathbf{P}_n,$$

where the polarization is the solution to the ordinary differential equation

$$\left(\frac{\partial^2}{\partial t^2} + \Gamma_n \frac{\partial}{\partial t} + \omega_n^2 \right) \mathbf{P}_n = \epsilon_0 f_n \omega_p^2 \mathbf{E}.$$

Here Γ_n is a damping coefficient, ω_n is a resonance frequency, f_n is an oscillator strength, and ω_p is the plasma frequency.

Enter values or expressions for the **Oscillator strength** f_n (SI unit: 1), the **Resonance frequency** ω_n (SI unit: rad/s), and the **Damping in time** coefficient Γ_n (SI unit: rad/s).

INITIAL VALUES

Enter values or expressions for the initial values of the components of the Drude-Lorentz polarization \mathbf{P}_n (SI unit: C/m²) and its time derivative $\partial\mathbf{P}_n/\partial t$ (SI unit: A/m²). The default values are 0 C/m² and 0 A/m², respectively.

DISCRETIZATION

To display this section, click the **Show More Options** button () and select **Discretization** from the **Show More Options** dialog box. Select the element order from the list box for the Drude-Lorentz polarization \mathbf{P}_n .



Time-Domain Modeling of Dispersive Drude-Lorentz Media:
Application Library path **RF_Module/Tutorials/drude_lorentz_media**

The Transmission Line Interface

The **Transmission Line (tl)** interface () , found under the **Radio Frequency** branch () when adding a physics interface, is used to study propagation of waves along one-dimensional transmission lines. The physics interface solves the time-harmonic transmission line equation for the electric potential.

The physics interface is used when solving for electromagnetic wave propagation along one-dimensional transmission lines and is available in 1D, 2D, and 3D. The physics interface has Eigenfrequency and Frequency Domain study types available. The Frequency Domain study is used for source driven simulations for a single frequency or a sequence of frequencies.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Transmission Line Equation**, **Absorbing Boundary**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Transmission Line** 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 `tl`.

PORT SWEEP SETTINGS

Select the **Activate port sweep** check box to switch on the port sweep. When selected, this invokes a parametric sweep over the lumped ports in addition to the automatically generated frequency sweep. The generated lumped parameters are in the form of an S-parameter matrix. For **Activate port sweep** enter a **Sweep parameter name** (the default is `PortName`) to assign a specific name to the variable that controls the port number solved for during the sweep.

For this physics interface, the S-parameters are subject to **Touchstone file export**. Click **Browse** to locate the file, or enter a file name and path. Select an **Output format** — **Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

DEPENDENT VARIABLES

The dependent variable (field variable) is the **Electric potential V** (SI unit: V). The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

Select the shape order for the **Electric potential** dependent variable — **Linear**, **Quadratic** (the default), or **Cubic**. For more information about the **Discretization** section, see [Settings for the Discretization Sections](#) in the *COMSOL Multiphysics Reference Manual*.



- [Domain, Boundary, Edge, Point, and Pair Nodes for the Transmission Line Equation Interface](#)
- [Theory for the Transmission Line Interface](#)
- [Visualization and Selection Tools](#) in the *COMSOL Multiphysics Reference Manual*



Quarter-Wave Transformer: Application Library path **RF_Module/Transmission_Lines_and_Waveguides/quarter_wave_transformer**

Domain, Boundary, Edge, Point, and Pair Nodes for the Transmission Line Equation Interface

The [Transmission Line Interface](#) has these domain, boundary, edge, point, 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.



Select **Edges** for 3D models, **Boundaries** for 2D models, and **Domains** for 1D models. **Points** are available for all space dimensions (3D, 2D, and 1D).

For all space dimensions, select **Points** for the boundary condition.

- [Absorbing Boundary](#)
- [Incoming Wave](#)
- [Initial Values](#)
- [Open Circuit](#)
- [Terminating Impedance](#)
- [Transmission Line Equation](#)
- [Short Circuit](#)
- [Lumped Port](#)



Theory for the Transmission Line Boundary Conditions

Transmission Line Equation

The **Transmission Line Equation** node is the main feature of the Transmission Line interface. It defines the 1D wave equation for the electric potential. The wave equation is written in the form

$$\frac{\partial}{\partial x} \left(\frac{1}{R + i\omega L} \frac{\partial V}{\partial x} \right) - (G + i\omega C)V = 0$$

where R , L , G , and C are the distributed resistance, inductance, conductance, and capacitance, respectively.

TRANSMISSION LINE EQUATION

Enter the values for the following:

- **Distributed resistance** R (SI unit: $\text{m}\cdot\text{kg}/(\text{s}^3\cdot\text{A}^2)$). The default is $0 \text{ m}\cdot\text{kg}/(\text{s}^3\cdot\text{A}^2)$.
- **Distributed inductance** L (SI unit: H/m). The default is $2.5\text{e-}6 \text{ H}/\text{m}$.
- **Distributed conductance** G (SI unit: S/m). The default is $0 \text{ S}/\text{m}$.
- **Distributed capacitance** C (SI unit: F/m). The default is $1\text{e-}9 \text{ F}/\text{m}$.

The default values give a characteristic impedance for the transmission line of 50Ω .

Initial Values

The **Initial Values** node adds an initial value for the electric potential that can serve as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter values or expressions for the initial values of the **Electric potential** V (SI unit: V).

Absorbing Boundary

The **Absorbing Boundary** condition is stated as

$$\frac{\mathbf{n} \cdot \nabla V}{R + j\omega L} + \frac{V}{Z_0} = 0$$

where γ is the complex propagation constant defined by

$$\gamma = \sqrt{(R + i\omega L)(G + i\omega C)}$$

and \mathbf{n} is the normal pointing out of the domain. The *absorbing boundary* condition prescribes that propagating waves are absorbed at the boundary and, thus, that there is no reflection at the boundary. The Absorbing Boundary condition is only available on external boundaries.



Theory for the Transmission Line Boundary Conditions

Incoming Wave

The **Incoming Wave** boundary condition

$$\frac{\mathbf{n} \cdot \nabla V}{R + j\omega L} + \frac{V - 2V_0}{Z_0} = 0$$

lets a wave of complex amplitude V_{in} enter the domain. The complex propagation constant γ and the outward-pointing normal \mathbf{n} are defined in the section describing the [Absorbing Boundary](#) node. The Incoming Wave boundary condition is only available on external boundaries.

VOLTAGE

Enter the value or expression for the input **Electric potential** V_0 (SI unit: V). The default is 1 V.



Theory for the Transmission Line Boundary Conditions

Open Circuit

The **Open Circuit** boundary condition is a special case of the [Terminating Impedance](#) boundary condition, assuming an infinite impedance, and, thus, zero current at the boundary. The condition is thus

$$\mathbf{n} \cdot \nabla V = 0$$

The Open Circuit boundary condition is only available on external boundaries.



Theory for the Transmission Line Boundary Conditions

Terminating Impedance

The **Terminating Impedance** boundary condition

$$\frac{\mathbf{n} \cdot \nabla V}{R + j\omega L} + \frac{V}{Z_L} = 0$$

specifies the terminating impedance to be Z_L . Notice that the [Absorbing Boundary](#) condition is a special case of this boundary condition for the case when

$$Z_L = Z_0 = \sqrt{\frac{R + j\omega L}{G + j\omega C}}$$

The [Open Circuit](#) and [Short Circuit](#) boundary conditions are also special cases of this condition. The Terminating Impedance boundary condition is only available on external boundaries.

IMPEDANCE

Enter the value or expression for the **Impedance** Z_L (SI unit: Ω). The default is 50 Ω .



Theory for the Transmission Line Boundary Conditions

Short Circuit

The **Short Circuit** node is a special case of the **Terminating Impedance** boundary condition, assuming that impedance is zero and, thus, the electric potential is zero. The constraint at this boundary is, thus, $V = 0$.

CONSTRAINT SETTINGS

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



Theory for the Transmission Line Boundary Conditions

Lumped Port

Use the **Lumped Port** node to apply a voltage or current excitation of a model or to connect to a circuit. The **Lumped Port** node also defines S-parameters (reflection and transmission coefficients) that can be used in later postprocessing steps.

PORT PROPERTIES

Enter a unique **Port Name**. It is recommended to use a numeric name as it is used to define the elements of the S-parameter matrix and numeric port names are also required for port sweeps and Touchstone file export.

Select a **Type of Port** — **Cable** (the default), **Current**, or **Circuit**.

SETTINGS



If a **Circuit** port type is selected under **Port Properties**, this section does not require any selection.

- If a **Cable** port type is selected under **Port Properties**, enter the **Characteristic impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω .
- If a **Current** terminal type is selected under **Port Properties**, enter a **Terminal current** I_0 (SI unit: A). The default is 1 A.

If **Cable** is selected as the port type, select the **Wave excitation at this port** check box to enter values or expressions for the:

- **Electric potential** V_0 (SI unit: V). The default is 1 V.
- **Port phase** θ_{in} (SI unit: rad). The default is 0 radians.



- [S-Parameters and Ports](#)
 - [Lumped Ports with Voltage Input](#)
 - [Theory for the Transmission Line Boundary Conditions](#)
-

The Electromagnetic Waves, Time Explicit Interface

The **Electromagnetic Waves, Time Explicit (ewte)** interface () , found under the **Radio Frequency** branch () when adding a physics interface, is used to model time-dependent electromagnetic wave propagation in linear media. The sources can be in the form of volumetric electric or magnetic currents, or electric surface currents or fields on boundaries.

This physics interface solves two first-order partial differential equations (Faraday's law and Maxwell-Ampère's law) for the electric and magnetic fields using the time explicit discontinuous Galerkin method.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Wave Equations**, **Perfect Electric Conductor**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Electromagnetic Waves, Time Explicit** to select physics features from the context menu.

The interface includes absorbing layers that are used to set up effective nonreflecting like boundary conditions. These features are added from the **Definitions** toolbar, by clicking **Absorbing Layer**. If COMSOL Multiphysics is not running in full-screen mode nor in a large window, **Absorbing Layer** is accessible on the **Definitions** toolbar by first clicking **Coordinate Systems** and then **Absorbing Layer**. You can also right-click **Definitions** in the **Model Builder** and select **Absorbing Layer** 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 `ewte`.

COMPONENTS

This section is available for 2D and 2D axisymmetric components.

Select the **Field components solved for**:

- **Full wave** (the default) to solve using a full three-component vector for the electric field \mathbf{E} and the magnetic field \mathbf{H} .
- **E in plane (TM wave)** to solve for the electric field vector components in the modeling plane and one magnetic field vector component perpendicular to the plane, assuming that there is no electric field perpendicular to the plane and no magnetic field components in the plane.
- **H in plane (TE wave)** to solve for the magnetic field vector components in the modeling plane and one electric field vector component perpendicular to the plane.

DEPENDENT VARIABLES

The dependent variables (field variables) are for the **Electric field vector \mathbf{E}** and for the **Magnetic field vector \mathbf{H}** . The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

Select the shape order for the **Electric and magnetic fields** dependent variables (the same order for both fields) — **Linear**, **Quadratic**, or **Cubic** (the default). For more information about the **Discretization** section, see [Settings for the Discretization Sections](#) in the *COMSOL Multiphysics Reference Manual*.



- [Domain, Boundary, and Pair Nodes for the Electromagnetic Waves, Time Explicit Interface](#)
 - [Theory for the Electromagnetic Waves, Time Explicit Interface](#)
-

Domain, Boundary, and Pair Nodes for the Electromagnetic Waves, Time Explicit Interface

The [Electromagnetic Waves, Time Explicit Interface](#) has these domain and boundary 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.



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.



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.

- [Electric Field](#)
- [Electric Current Density](#)
- [Flux/Source](#)
- [Initial Values](#)
- [Scattering Boundary Condition](#)
- [Magnetic Field](#)
- [Magnetic Current Density](#)
- [Perfect Electric Conductor](#)
- [Perfect Magnetic Conductor](#)
- [Surface Current Density](#)
- [Wave Equations](#)
- [Lumped Port](#)

Wave Equations

The **Wave Equations** node is the main node for the Electromagnetic Waves, Time Explicit interface. The governing transient equations can be written in the form

$$\begin{aligned}\nabla \times \mathbf{H} &= \sigma \mathbf{E} + \frac{\partial \mathbf{D}}{\partial t} \\ \nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t}\end{aligned}$$

with the constitutive relations $\mathbf{B} = \mu_0 \mu_r \mathbf{H}$ and $\mathbf{D} = \epsilon_0 \epsilon_r \mathbf{E}$, which reads

$$\begin{aligned}\epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} + \sigma \mathbf{E} &= 0 \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \times \mathbf{E} &= 0\end{aligned}$$

MATERIAL PROPERTIES

The default **Relative permittivity** ϵ_r (dimensionless), **Relative permeability** μ_r (dimensionless), and **Electrical conductivity** σ (SI unit: S/m) take values **From material**.

For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix.

NUMERICAL PARAMETERS

The defaults for each parameter are as follows:

- **Lax-Friedrichs flux parameter for E field** τ_E (SI unit: S), the default is $0.5/Z$ for Ampere's law.
- **Lax-Friedrichs flux parameter for H field** τ_H (SI unit: Ω), the default is $0.5 Z$ for Faraday's law, where Z is the impedance of vacuum.
- **Estimate of maximum wave speed** c_{\max} (SI unit: m/s) the default is taken from the speed of light in a vacuum c_{const} .

FILTER PARAMETERS

The filter provides higher-order smoothing of nodal discontinuous Galerkin formulations and is intended to be used for absorbing layers, but you can also use it to stabilize linear wave problems with highly varying coefficients. The filter is constructed by transforming the solution (in each global time step) to an orthogonal polynomial representation, multiplying with a damping factor and then transforming back to the (Lagrange) nodal basis. Select the **Activate** check box to use this filter.

The exponential filter can be described by the matrix formula

$$V\Lambda V^{-1}$$

where V is a Vandermonde matrix induced by the node points, and Λ is a diagonal matrix with the exponential damping factors on the diagonal:

$$\Lambda_{mm} = \sigma(\eta) = \begin{cases} 1, & 0 \leq \eta \leq \eta_c \\ e^{-\alpha \left(\frac{\eta - \eta_c}{1 - \eta_c}\right)^{2s}}, & \eta_c \leq \eta \leq 1 \end{cases}$$

where

$$\eta = \eta(m) = \frac{i_m}{N_p}$$

and N_p is the basis function and i_m the polynomial order for coefficient m . α (default value: 36), η_c (default value: 1), and s (default value: 3) are the filter parameters that

you specify in the corresponding text fields. The damping is derived from a spatial dissipation operator of order $2s$. For $s = 1$, you obtain a damping that is related to the classical 2nd-order Laplacian. Higher order (larger s) gives less damping for the lower-order polynomial coefficients (a more pronounced low-pass filter), while keeping the damping property for the highest values of η , which is controlled by α . The default values 36 for α correspond to maximal damping for $\eta = 1$. It is important to realize that the effect of the filter is influenced by how much of the solution (energy) is represented by the higher-order polynomial coefficients. For a well resolved solution this is a smaller part than for a poorly resolved solution. The effect is stronger for poorly resolved solutions than for well resolved ones. This is one of the reasons why this filter is useful in an absorbing layer where the energy is transferred to the higher-order coefficients through a coordinate transformation. See [Ref. 1](#) (Chapter 5) for more information.

α must be positive; $\alpha = 0$ means no dissipation, and the maximum value is related to the machine precision, $-\log(\epsilon)$, which is approximately 36. η_c should be between 0 and 1, where $\eta_c = 0$ means maximum filtering, and $\eta_c = 1$ means no filtering, even if filtering is active.

When using **Absorbing Layer** features, add an additional **Wave Equations** feature for the corresponding domain selection. Select the **Activate** check box and add filter parameters. An example of a filter parameter combination that can be used for a **Wave Equations** feature active on an **Absorbing Layer** domain selection is $\alpha = 0.1$, $\eta_c = 0.01$, and $s = 2$. However, other combinations could work better, depending on the particular application.



Absorbing Layers

Reference

1. J.S. Hesthaven and T. Warburton, *Nodal Discontinuous Galerkin Methods — Algorithms, Analysis, and Applications*, Springer, 2008.

Initial Values

The **Initial Values** node adds the initial values for the **Electric field** and **Magnetic field** variables that serve as an initial condition for the transient simulation.

DOMAIN SELECTION

If there is more than one type of domain, each with different initial values defined, it might be necessary to remove these domains from the selection. These are then defined in an additional **Initial Values** node.

INITIAL VALUES

Enter values or expressions for the initial values of the components of the **Electric field \mathbf{E}** (SI unit: V/m) and **Magnetic field \mathbf{H}** (SI unit: A/m). The default values are 0 for all vector components.

Electric Current Density

The **Electric Current Density** node adds an external current density to the specified domains, which appears on the right-hand side of Ampere's law

$$\epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} + \sigma \mathbf{E} = -\mathbf{J}_e$$

ELECTRIC CURRENT DENSITY

Based on space dimension, enter the coordinates (**x**, **y**, and **z** for 3D components for example) of the **Electric current density \mathbf{J}_e** (SI unit: A/m²).

Magnetic Current Density

The **Magnetic Current Density** node adds an external current density to the specified domains, which appears on the right-hand side of Faraday's law

$$\mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \times \mathbf{E} = -\mathbf{J}_m$$

MAGNETIC CURRENT DENSITY

Based on space dimension, enter the coordinates (**x**, **y**, and **z** for 3D components for example) of the **Magnetic current density \mathbf{J}_m** (SI unit: V/m²).

Electric Field

The **Electric Field** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{n} \times \mathbf{E}_0$$

specifies the tangential component of the electric field. The commonly used special case of zero tangential electric field (perfect electric conductor) is described in the next section.

ELECTRIC FIELD

Enter values or expressions for the components of the **Electric field** \mathbf{E}_0 (SI unit: V/m).

Perfect Electric Conductor

The **Perfect Electric Conductor** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{0}$$

is a special case of the electric field boundary condition that sets the tangential component of the electric field to zero. It is used for the modeling of a lossless metallic surface, for example, a ground plane or as a symmetry type boundary condition.

It imposes symmetry for magnetic fields and antisymmetry for electric fields and electric currents. It supports induced electric surface currents and thus any prescribed or induced electric currents (volume, surface, or edge currents) flowing into a perfect electric conductor boundary is automatically balanced by induced surface currents.

Magnetic Field

The **Magnetic Field** node adds a boundary condition for specifying the tangential component of the magnetic field at the boundary:

$$\mathbf{n} \times \mathbf{H} = \mathbf{n} \times \mathbf{H}_0$$

MAGNETIC FIELD

Enter values or expressions for the components of the **Magnetic field** \mathbf{H}_0 (SI unit: A/m).

Perfect Magnetic Conductor

The **Perfect Magnetic Conductor** boundary condition

$$\mathbf{n} \times \mathbf{H} = \mathbf{0}$$

is a special case of the surface current density boundary condition that sets the tangential component of the magnetic field and thus also the surface current density

to zero. On external boundaries, this can be interpreted as a “high surface impedance” boundary condition or used as a symmetry type boundary condition. It imposes symmetry for electric fields and electric currents. Electric currents (volume, surface, or edge currents) are not allowed to flow into a perfect magnetic conductor boundary as that would violate current conservation. On interior boundaries, the perfect magnetic conductor boundary condition literally sets the tangential magnetic field to zero which in addition to setting the surface current density to zero also makes the tangential electric field discontinuous.

Surface Current Density

The **Surface Current Density** boundary condition

$$\begin{aligned} -\mathbf{n} \times \mathbf{H} &= \mathbf{J}_s \\ \mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) &= \mathbf{J}_s \end{aligned}$$

specifies a surface current density at both exterior and interior boundaries. The current density is specified as a three-dimensional vector, but because it needs to flow along the boundary surface, COMSOL Multiphysics projects it onto the boundary surface and neglects its normal component. This makes it easier to specify the current density and avoids unexpected results when a current density with a component normal to the surface is given.

SURFACE CURRENT DENSITY

Enter values or expressions for the components of the **Surface current density** \mathbf{J}_{s0} (SI unit: A/m). The defaults are 0 A/m for all vector components.

Scattering Boundary Condition

The **Scattering Boundary Condition**

$$\mathbf{n} \times \mathbf{E} = Z_0 \mathbf{H}$$

specifies the tangential component of both electric and magnetic fields.

IMPEDANCE

Enter the value or expression for the medium **Impedance** Z_0 (SI unit: Ω). By default, the Z_0 uses the value of the vacuum’s impedance. Then select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the material characteristics and enter values or expressions in the field or matrix.

Flux/Source

The **Flux/Source** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{E}_0$$

$$\mathbf{n} \times \mathbf{H} = \mathbf{H}_0$$

specifies the tangential component of both electric and magnetic fields. This boundary condition is available when **Advanced Physics Options** is selected in the **Show More Options** dialog box on the **Model Builder** toolbar.

BOUNDARY FLUX/SOURCE

Enter values or expressions for the components of the tangential **Electric field** \mathbf{E}_0 (SI unit: V/m) and the tangential **Magnetic field** \mathbf{H}_0 (SI unit: A/m).

Background Field

The **Background Field** feature triggers the scattered field formulation, where the dependent variable is the relative field. The same wave equations are used as in the full field formulation, but the total field that enters the equations are written as the sum of the relative field and the background field, $\mathbf{E} = \mathbf{E}_{\text{relative}} + \mathbf{E}_{\text{background}}$, and it is the dependent variable $\mathbf{E}_{\text{relative}}$ that is solved for. When the background field is a solution of the wave equation, the relative field is the scattered field.

SETTINGS

Select a **Background wave type** — **User defined** (the default), or **Modulated Gaussian pulse**.

User Defined

Enter the component expressions for the **Background electric field** \mathbf{E}_b (SI unit: V/m) and **Background magnetic field** \mathbf{H}_b (SI unit: A/m). The entered expressions must be differentiable in time domain since the derivative of the background field is used in the governing equations.

Modulated Gaussian pulse

Select a **Direction of propagation** — **+x** (the default), **-x**, **+y**, **-y**, or for 3D components, **Along the +z**, or **-z**.

Select a **Polarization** — **y** (the default), **z**, or **x**. The list of available polarization varies based on the selection of **Direction of propagation**.

- Enter a **Center frequency** f_0 (SI unit: Hz). The default is 1 GHz.
- Enter a **Phase velocity** v_p (SI unit: m/s). The default is `c_const`.
- Enter a **Wave impedance** Z (SI unit: Ω). The default is `Z0_const`.
- Enter a **Distance from origin to wave launching plane** d_{offset} (SI unit: m). The default is 0 m.

For a modulated Gaussian pulse propagating in the positive x direction, the electric field is expressed as

$$E(x, t) = \frac{1}{\tau\sqrt{2\pi}} \exp\left(-\frac{\left(t - \mu - \frac{x + d_{\text{offset}}}{v_p}\right)^2}{2\tau^2}\right) \sin\left(2\pi f_0\left(t - \frac{x}{v_p}\right)\right)$$

where τ is the pulse duration, defined as $1/2f_0$, μ is a time delay set to $2/f_0$, and v_p is the phase velocity. The time delay μ is used to excite a modulated Gaussian pulse whose initial magnitude is very small when it is launched and gradually increases as it propagates.



Wideband RCS Calculation Using Time-Domain Simulation and FFT: Application Library path **RF_Module/Scattering_and_RCS/rcs_time_explicit** demonstrates how to set up a background field.

Far-Field Domain

To set up a far-field calculation, add a **Far-Field Domain** node and specify the far-field domains in its Settings window. Use Far-Field Calculation subnodes (one is added by default) to specify all other settings needed to define the far-field calculation. By default, all of the domains are selected. The selection can be modified. In that case, select only a homogeneous domain or domain group that is outside of all radiating and scattering objects and which has the material settings of the far-field medium.

Far-Field Calculation

A **Far-Field Calculation** subnode is added by default to the **Far-Field Domain** node and is used to select boundaries corresponding to a single closed surface surrounding all

radiating and scattering objects. By default, all exterior boundaries of the **Far-Field Domain** are selected. Symmetry reduction of the geometry makes it relevant to select boundaries defining a nonclosed surface. Also use this feature to indicate symmetry planes and symmetry cuts applied to the geometry, and whether the selected boundaries are defining the inside or outside of the far field domain; that is, to say whether they are facing away from infinity or toward infinity.

FAR-FIELD CALCULATION

Enter a **Far-field variable name**. The default is E_{far} .

Select as needed the **Symmetry in the $x=0$ plane**, **Symmetry in the $y=0$ plane**, or **Symmetry in the $z=0$ plane** check boxes to use it your model when calculating the far-field variable. The symmetry planes have to coincide with one of the Cartesian coordinate planes.

When a check box is selected, also choose the type of symmetry to use from the **Symmetry type** list that appears — **Symmetry in E (PMC)** or **Symmetry in H (PEC)**. The selection should match the boundary condition used for the symmetry boundary. Using these settings, include the parts of the geometry that are not in the model for symmetry reasons in the far-field analysis.

From the **Boundary relative to domain** list, select **Inside** or **Outside** (the default) to define if the selected boundaries are defining the inside or outside of the far-field domain (that is, whether facing away from infinity or toward infinity).



A **Time to Frequency FFT** study step must be added after the **Time Dependent** study step to generate the necessary frequency-domain data, used in the far-field analysis.

Theory for the Electromagnetic Waves Interfaces

The Electromagnetic Waves, Frequency Domain Interface and The Electromagnetic Waves, Transient Interface theory is described in this section:

- [Introduction to the Physics Interface Equations](#)
- [Frequency Domain Equation](#)
- [Time Domain Equation](#)
- [Vector Elements](#)
- [Eigenfrequency Calculations](#)
- [Gaussian Beams as Background Fields](#)
- [Effective Material Properties in Effective Media and Mixtures](#)
- [Effective Conductivity in Effective Media and Mixtures](#)
- [Effective Relative Permittivity in Effective Media and Mixtures](#)
- [Effective Relative Permeability in Effective Media and Mixtures](#)
- [Archie's Law Theory](#)

Introduction to the Physics Interface Equations

Formulations for high-frequency waves can be derived from Maxwell-Ampère's and Faraday's laws,

$$\begin{aligned}\nabla \times \mathbf{H} &= \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \\ \nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t}\end{aligned}$$

Using the constitutive relations for linear materials $\mathbf{D} = \epsilon \mathbf{E}$ and $\mathbf{B} = \mu \mathbf{H}$ as well as a current $\mathbf{J} = \sigma \mathbf{E}$, these two equations become

$$\begin{aligned}\nabla \times \mathbf{H} &= \sigma \mathbf{E} + \frac{\partial \epsilon \mathbf{E}}{\partial t} \\ \nabla \times \mathbf{E} &= -\mu \frac{\partial \mathbf{H}}{\partial t}\end{aligned}$$

Frequency Domain Equation

Writing the fields on a time-harmonic form, assuming a sinusoidal excitation and linear media,

$$\begin{aligned}\mathbf{E}(x, y, z, t) &= \mathbf{E}(x, y, z)e^{j\omega t} \\ \mathbf{H}(x, y, z, t) &= \mathbf{H}(x, y, z)e^{j\omega t}\end{aligned}$$

the two laws can be combined into a time harmonic equation for the electric field, or a similar equation for the magnetic field

$$\begin{aligned}\nabla \times (\mu^{-1} \nabla \times \mathbf{E}) - \omega^2 \epsilon \mathbf{E} &= \mathbf{0} \\ \nabla \times (\epsilon^{-1} \nabla \times \mathbf{H}) - \omega^2 \mu \mathbf{H} &= \mathbf{0}\end{aligned}$$

The first of these, based on the electric field is used in [The Electromagnetic Waves, Frequency Domain Interface](#).

Using the relation $\epsilon_r = n^2$, where n is the refractive index, the equation can alternatively be written

$$\nabla \times (\nabla \times \mathbf{E}) - k_0^2 n^2 \mathbf{E} = \mathbf{0}$$

The wave number in vacuum k_0 is defined by

$$k_0 = \omega \sqrt{\epsilon_0 \mu_0} = \frac{\omega}{c_0}$$

where c_0 is the speed of light in vacuum.

When the equation is written using the refractive index, the assumption is that $\mu_r = 1$ and $\sigma = 0$ and only the constitutive relations for linear materials are available. When solving for the scattered field the same equations are used but $\mathbf{E} = \mathbf{E}_{sc} + \mathbf{E}_i$ and \mathbf{E}_{sc} is the dependent variable.

EIGENFREQUENCY ANALYSIS

When solving the frequency domain equation as an eigenfrequency problem the eigenvalue is the complex eigenfrequency $\lambda = j\omega + \delta$, where δ is the damping of the solution. The *Q-factor* is given from the eigenvalue by the formula

$$Q_{\text{fact}} = \frac{\omega}{2|\delta|}$$

MODE ANALYSIS AND BOUNDARY MODE ANALYSIS

In mode analysis and boundary mode analysis COMSOL Multiphysics solves for the propagation constant. The time-harmonic representation is almost the same as for the eigenfrequency analysis, but with a known propagation in the out-of-plane direction

$$\mathbf{E}(\mathbf{r}, t) = \text{Re}(\tilde{\mathbf{E}}(\mathbf{r}_T)e^{j\omega t - j\beta z}) = \text{Re}(\tilde{\mathbf{E}}(\mathbf{r})e^{j\omega t - \alpha z})$$

The spatial parameter, $\alpha = \delta_z + j\beta = -\lambda$, can have a real part and an imaginary part. The propagation constant is equal to the imaginary part, and the real part, δ_z , represents the damping along the propagation direction. When solving for all three electric field components the allowed anisotropy of the optionally complex relative permittivity and relative permeability is limited to:

$$\epsilon_{rc} = \begin{bmatrix} \epsilon_{rxx} & \epsilon_{rxy} & 0 \\ \epsilon_{ryx} & \epsilon_{ryy} & 0 \\ 0 & 0 & \epsilon_{rzz} \end{bmatrix} \quad \mu_r = \begin{bmatrix} \mu_{rxx} & \mu_{rxy} & 0 \\ \mu_{ryx} & \mu_{ryy} & 0 \\ 0 & 0 & \mu_{rzz} \end{bmatrix}$$



Limiting the electric field component solved for to the out-of-plane component for TE modes requires that the medium is homogeneous; that is, μ and ϵ are constant. When solving for the in-plane electric field components for TM modes, μ can vary but ϵ must be constant. It is strongly recommended to use the most general approach, that is solving for all three components which is sometimes referred to as “perpendicular hybrid-mode waves”.

Variables Influenced by Mode Analysis

The following table lists the variables that are influenced by the mode analysis:

NAME	EXPRESSION	CAN BE COMPLEX	DESCRIPTION
beta	imag(-lambda)	No	Propagation constant
dampz	real(-lambda)	No	Attenuation constant
dampzdB	20*log10(exp(1))*dampz	No	Attenuation per meter in dB
neff	j*lambda/k0	Yes	Effective mode index

PROPAGATING WAVES IN 2D

In 2D, different polarizations can be chosen by selecting to solve for a subset of the 3D vector components. When selecting all three components, the 3D equation applies

with the addition that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field.

In 2D, the electric field varies with the out-of-plane wave number k_z as

$$\mathbf{E}(x, y, z) = \tilde{\mathbf{E}}(x, y) \exp(-ik_z z).$$

The wave equation is thereby rewritten as

$$(\nabla - ik_z \mathbf{z}) \times [\mu_r^{-1} (\nabla - ik_z \mathbf{z}) \times \tilde{\mathbf{E}}] - k_0^2 \epsilon_{rc} \tilde{\mathbf{E}} = \mathbf{0},$$

where \mathbf{z} is the unit vector in the out-of-plane z direction.

Similarly, in 2D axisymmetry, the electric field varies with the azimuthal mode number m as

$$\mathbf{E}(r, \varphi, z) = \tilde{\mathbf{E}}(r, z) \exp(-im\varphi)$$

and the wave equation is expressed as

$$\left(\nabla - i \frac{m}{r} \boldsymbol{\varphi} \right) \times \left[\mu_r^{-1} \left(\nabla - i \frac{m}{r} \boldsymbol{\varphi} \right) \times \tilde{\mathbf{E}} \right] - k_0^2 \epsilon_{rc} \tilde{\mathbf{E}} = \mathbf{0},$$

where $\boldsymbol{\varphi}$ is the unit vector in the out-of-plane φ direction.

In-plane Hybrid-Mode Waves

Solving for all three components in 2D is referred to as “hybrid-mode waves”. The equation is formally the same as in 3D with the addition that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field

In-plane TM Waves

The TM waves polarization has only one magnetic field component in the z direction, and the electric field lies in the modeling plane. Thus the time-harmonic fields can be obtained by solving for the in-plane electric field components only. The equation is formally the same as in 3D, the only difference being that the out-of-plane electric field component is zero everywhere and that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field.

In-plane TE Waves

As the field propagates in the modeling xy -plane a TE wave has only one nonzero electric field component, namely in the z direction. The magnetic field lies in the

modeling plane. Thus the time-harmonic fields can be simplified to a scalar equation for E_z ,

$$-\nabla \cdot (\tilde{\mu}_r \nabla E_z) - \epsilon_{rzz} k_0^2 E_z = 0$$

where

$$\tilde{\mu}_r = \frac{\mu_r^T}{\det(\mu_r)}$$

To be able to write the fields in this form, it is also required that ϵ_r , σ , and μ_r are nondiagonal only in the xy -plane. μ_r denotes a 2-by-2 tensor, and ϵ_{rzz} and σ_{zz} are the relative permittivity and conductivity in the z direction.

Axisymmetric Hybrid-Mode Waves

Solving for all three components in 2D is referred to as “hybrid-mode waves”. The equation is formally the same as in 3D with the addition that spatial derivatives with respect to ϕ are evaluated for the prescribed azimuthal mode number dependence of the electric field.

Axisymmetric TM Waves

A TM wave has a magnetic field with only a ϕ component and thus an electric field with components in the rz -plane only. The equation is formally the same as in 3D, the only difference being that the ϕ component is zero everywhere and that spatial derivatives with respect to ϕ are evaluated for the prescribed azimuthal mode number dependence of the electric field.

Axisymmetric TE Waves

A TE wave has only an electric field component in the ϕ direction, and the magnetic field lies in the modeling plane. Given these constraints, the 3D equation can be simplified to a scalar equation for E_ϕ . To write the fields in this form, it is also required that ϵ_r and μ_r are nondiagonal only in the rz -plane. μ_r denotes a 2-by-2 tensor, while $\epsilon_{r\phi\phi}$ and $\sigma_{\phi\phi}$ are the relative permittivity and conductivity in the ϕ direction.

INTRODUCING LOSSES IN THE FREQUENCY DOMAIN

Electric Losses

The frequency domain equations allow for several ways of introducing electric losses. Finite conductivity results in a *complex permittivity*,

$$\epsilon_c = \epsilon - j\frac{\sigma}{\omega}$$

The conductivity gives rise to ohmic losses in the medium.

A more general approach is to use a complex permittivity,

$$\epsilon_c = \epsilon_0(\epsilon' - j\epsilon'')$$

where ϵ' is the real part of ϵ_r , and all losses (dielectric and conduction losses) are given by ϵ'' . The dielectric loss model can also single out the losses from finite conductivity (so that ϵ'' only represents dielectric losses) resulting in:

$$\epsilon_c = \epsilon_0\left(\epsilon' - j\left(\frac{\sigma}{\omega\epsilon_0} + \epsilon''\right)\right)$$

The complex permittivity can also be introduced as a loss tangent:

$$\epsilon_c = \epsilon_0\epsilon'(1 - j\tan\delta)$$



When specifying losses through a loss tangent, conductivity is not allowed as an input.

In optics and photonics applications, the refractive index is often used instead of the permittivity. In materials where μ_r is 1, the relation between the complex refractive index

$$\bar{n} = n - j\kappa$$

and the complex relative permittivity is

$$\epsilon_{rc} = \bar{n}^2$$

that is

$$\epsilon'_r = n^2 - \kappa^2$$

$$\epsilon''_r = 2n\kappa$$

The inverse relations are

$$n^2 = \frac{1}{2}(\epsilon'_r + \sqrt{\epsilon'^2_r + \epsilon''^2_r})$$

$$\kappa^2 = \frac{1}{2}(-\epsilon'_r + \sqrt{\epsilon'^2_r + \epsilon''^2_r})$$

The parameter κ represents a damping of the electromagnetic wave. When specifying the refractive index, conductivity is not allowed as an input.

In the physics and optics literature, the time harmonic form is often written with a minus sign (and “ i ” instead of “ j ”):

$$\mathbf{E}(x, y, z, t) = \mathbf{E}(x, y, z)e^{-i\omega t}$$

This makes an important difference in how loss is represented by complex material coefficients like permittivity and refractive index, that is, by having a positive imaginary part rather than a negative one. Therefore, material data taken from the literature might have to be conjugated before using it in a model.

Magnetic Losses

The frequency domain equations allow for magnetic losses to be introduced as a *complex relative permeability*.

$$\mu_r = (\mu' - j\mu'')$$

The complex relative permeability can be combined with any electric loss model except refractive index.

Time Domain Equation

The relations $\mu\mathbf{H} = \nabla \times \mathbf{A}$ and $\mathbf{E} = -\partial\mathbf{A}/\partial t$ (using the gauge for which the scalar electric potential vanishes) make it possible to rewrite Maxwell-Ampère’s law using the magnetic potential.

$$\mu_0\sigma\frac{\partial\mathbf{A}}{\partial t} + \mu_0\frac{\partial}{\partial t}\epsilon\frac{\partial\mathbf{A}}{\partial t} + \nabla \times (\mu_r^{-1}\nabla \times \mathbf{A}) = 0$$

This is the equation used by [The Electromagnetic Waves, Transient Interface](#). It is suitable for the simulation of nonsinusoidal waveforms or nonlinear media.

Using the relation $\epsilon_r = n^2$, where n is the refractive index, the equations can alternatively be written

$$\mu_0 \epsilon_0 \frac{\partial}{\partial t} \left(n^2 \frac{\partial \mathbf{A}}{\partial t} \right) + \nabla \times (\nabla \times \mathbf{A}) = 0$$

WAVES IN 2D

In 2D, different polarizations can be chosen by selecting to solve for a subset of the 3D vector components. When selecting all three components, the 3D equation applies with the addition that out-of-plane spatial derivatives are set to zero.

In-plane Hybrid-Mode Waves

Solving for all three components in 2D is referred to as “hybrid-mode waves”. The equation form is formally the same as in 3D with the addition that out-of-plane spatial derivatives are set to zero.

In-plane TM Waves

The TM waves polarization has only one magnetic field component in the z direction, and thus the electric field and vector potential lie in the modeling plane. Hence it is obtained by solving only for the in-plane vector potential components. The equation is formally the same as in 3D, the only difference being that the out-of-plane vector potential component is zero everywhere and that out-of-plane spatial derivatives are set to zero.

In-plane TE Waves

As the field propagates in the modeling xy -plane a TE wave has only one nonzero vector potential component, namely in the z direction. The magnetic field lies in the modeling plane. Thus the equation in the time domain can be simplified to a scalar equation for A_z :

$$\mu_0 \sigma \frac{\partial A_z}{\partial t} + \mu_0 \epsilon_0 \frac{\partial}{\partial t} \left(\epsilon_r \frac{\partial A_z}{\partial t} \right) + \nabla \cdot (\mu_r^{-1} (\nabla A_z)) = 0$$

Using the relation $\epsilon_r = n^2$, where n is the refractive index, the equation can alternatively be written

$$\mu_0 \epsilon_0 \frac{\partial}{\partial t} \left(n^2 \frac{\partial A_z}{\partial t} \right) + \nabla \cdot (\nabla A_z) = 0$$

When using the refractive index, the assumption is that $\mu_r = 1$ and $\sigma = 0$ and only the constitutive relations for linear materials can be used.

Axisymmetric Hybrid-Mode Waves

Solving for all three components in 2D is referred to as “hybrid-mode waves”. The equation form is formally the same as in 3D with the addition that spatial derivatives with respect to φ are set to zero.

Axisymmetric TM Waves

TM waves have a magnetic field with only a φ component and thus an electric field and a magnetic vector potential with components in the rz -plane only. The equation is formally the same as in 3D, the only difference being that the φ component is zero everywhere and that spatial derivatives with respect to φ are set to zero.

Axisymmetric TE Waves

A TE wave has only a vector potential component in the φ direction, and the magnetic field lies in the modeling plane. Given these constraints, the 3D equation can be simplified to a scalar equation for A_φ . To write the fields in this form, it is also required that ϵ_r and μ_r are nondiagonal only in the rz -plane. μ_r denotes a 2-by-2 tensor, while $\epsilon_{r\varphi\varphi}$ and $\sigma_{\varphi\varphi}$ are the relative permittivity and conductivity in the φ direction.

Vector Elements

Whenever solving for more than a single vector component, it is not possible to use Lagrange elements for electromagnetic wave modeling. The reason is that they force the fields to be continuous everywhere. This implies that the physics interface conditions, which specify that the normal components of the electric and magnetic fields are discontinuous across interior boundaries between media with different permittivity and permeability, cannot be fulfilled. To overcome this problem, the Electromagnetic Waves, Frequency Domain interface uses *vector elements*, which do not have this limitation.

The solution obtained when using vector elements also better fulfills the divergence conditions $\nabla \cdot \mathbf{D} = 0$ and $\nabla \cdot \mathbf{B} = 0$ than when using Lagrange elements.

Eigenfrequency Calculations

When making eigenfrequency calculations, there are a few important things to note:

- Nonlinear eigenvalue problems appear for impedance boundary conditions with nonzero conductivity and for *scattering boundary conditions* adjacent to domains with nonzero conductivity. Such problems have to be treated specially.
- Some of the boundary conditions, such as the surface current density condition and the electric field condition, can specify a source in the eigenvalue problem. These conditions are available as a general tool to specify arbitrary expressions between the **H** field and the **E** field. Avoid specifying solution-independent sources for these conditions because the eigenvalue solver ignores them anyway.

Using the default parameters for the eigenfrequency study, it might find a large number of false eigenfrequencies, which are almost zero. This is a known consequence of using vector elements. To avoid these eigenfrequencies, change the parameters for the eigenvalue solver in the **Study Settings**. Adjust the settings so that the solver searches for eigenfrequencies closer to the lowest eigenfrequency than to zero.

Gaussian Beams as Background Fields

When solving for the scattered field, the background wave type can be set to a predefined Gaussian beam from within the **Settings** of [The Electromagnetic Waves, Frequency Domain Interface](#). In the paraxial approximation the background field for a Gaussian beam propagating along the z -axis is defined below,

$$\mathbf{E}_b(x, y, z) = \mathbf{E}_{bg0} \frac{w_0}{w(z)} \exp \left[-\frac{\rho^2}{w^2(z)} - jkz - jk \frac{\rho^2}{2R(z)} + j\eta(z) \right],$$

where w_0 is the beam radius, p_0 is the focal plane on the z -axis, \mathbf{E}_{bg0} is the background electric field amplitude and the spot radius for different positions along the propagation axis is given by

$$w(z) = w_0 \sqrt{1 + \left(\frac{z - p_0}{z_0} \right)^2}.$$

$$R(z) = (z - p_0) \left[1 + \left(\frac{z_0}{z - p_0} \right)^2 \right]$$

defines the radius of curvature for the phase of the field and the so called Gouy phase shift is given by

$$\eta(z) = \text{atan}\left(\frac{z - p_0}{z_0}\right).$$

The equations above are expressed using the Rayleigh range z_0 and the transverse coordinate ρ , defined by

$$z_0 = \frac{k_0 w_0^2}{2}, \rho^2 = x^2 + y^2.$$

Note that the time-harmonic *ansatz* in COMSOL is $e^{j\omega t}$ and with this convention, the beam above propagates in the $+z$ direction. The equations are modified accordingly for beams propagating along the other coordinate axes.

The background field for a Gaussian beam is defined in a similar way for 2D components. In the particular case where the beam propagates along the x -axis, the background field is defined as

$$\mathbf{E}_b(x, y, z) = \mathbf{E}_{bg0} \sqrt{\frac{w_0}{w(x)}} \exp\left[-\frac{y^2}{w^2(x)} - jkx - jk\frac{y^2}{2R(x)} + j\frac{\eta(x)}{2}\right].$$

For a beam propagating along the y -axis, the coordinates x and y are interchanged.

Notice that the expressions above for Gaussian beams are not solutions to the Helmholtz equation, but to the so called paraxial approximation of the Helmholtz equation. This means that these equations become less accurate the smaller the spot radius is and should not be used when the spot radius is of the same size as or smaller than the wavelength.

To circumvent the problem that the paraxial approximation formula is not a solution to the Helmholtz equation, a plane wave expansion can be used to approximate a Gaussian beam. Since each plane wave is a solution to Helmholtz equation, also the expansion is a solution to Helmholtz equation.

The plane wave expansion approximates the Gaussian distribution in the focal plane

$$\mathbf{E}_{b, \text{Gauss}}(\mathbf{r}) = E_0 \exp\left(-\frac{x^2 + y^2}{w_0^2}\right) \mathbf{e} = \sum_{l=-L}^L \sum_{m=-M}^M \sum_{n=0}^1 a_{lmn} \mathbf{u}_n(\mathbf{k}_{lm}) \exp(-i\mathbf{k}_{lm} \cdot \mathbf{r}),$$

where the beam is assumed to be propagating in the z direction, the focal plane is spanned by the x and y coordinates, \mathbf{e} is the unit magnitude transverse polarization in the focal plane, l and m denote the indices for the wave vectors, the index n accounts for the two polarizations per wave vector \mathbf{k}_{lm} , a_{lmn} is the amplitude, $\mathbf{u}_n(\mathbf{k}_{lm})$ is the unit magnitude polarization, and \mathbf{r} is the position vector.

Multiplying with the conjugate of the exponential factor above and the polarization factor $\mathbf{u}_n(\mathbf{k}_{lm})$ and applying a surface integral over the entire focal plane allows us to extract the amplitudes as

$$a_{lmn} = \frac{E_0 w_0^2 (\mathbf{e} \cdot \mathbf{u}_n(\mathbf{k}_{lm}))}{4\pi} \exp\left(-\frac{k_{t,lm}^2 w_0^2}{4}\right),$$

where $k_{t,lm}$ is the magnitude of the transverse wave vector component.

Effective Material Properties in Effective Media and Mixtures

One way of dealing with effective media or mixtures of solids in electromagnetic models is to replace them with an homogenized medium. The electric and magnetic properties of this medium are computed from the properties of each phase by means of an averaging formula.

There are several possible approaches to compute an average material property starting from the material properties and the volume fraction of each material.

The following sections illustrate the different formulas available to compute the *effective electrical conductivity*, the *effective relative permittivity* and the *effective relative permeability* of a homogenized medium. In the following, volume fractions of the materials are indicated with θ_i , where i is the material index, and they are assumed to be fractional (between 0 and 1). Up to five different materials can be specified as phases of the mixture. Typically, their volume fractions should add up to 1.

Three methods are available to compute the averaged electrical conductivity of the mixture.

VOLUME AVERAGE, CONDUCTIVITY

If the electric conductivities of the two materials are not so different from each other, a simple form of averaging can be used, such as a volume average:

$$\sigma = \sum_{i=1}^n \theta_i \sigma_i = \theta_1 \sigma_1 + \theta_2 \sigma_2 + \dots$$

where σ_i is the conductivity of the material i . This is equivalent to a “parallel” system of resistivities.

If the conductivities are defined by second order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RESISTIVITY

A similar expression for the effective conductivity can be used, which mimics a “series” connection of resistivities. Equivalently, the effective conductivity is obtained from

$$\frac{1}{\sigma} = \sum_{i=0}^n \frac{\theta_i}{\sigma_i} = \frac{\theta_1}{\sigma_1} + \frac{\theta_2}{\sigma_2} + \dots$$

If the conductivities are defined by second order tensors, the inverse of the tensors are used.

POWER LAW

A power law gives the following expression for the equivalent conductivity:

$$\sigma = \prod_{i=0}^n \sigma_i^{\theta_i} = \sigma_1^{\theta_1} \sigma_2^{\theta_2} \dots$$

The effective conductivity calculated by [Volume Average, Conductivity](#) is the upper bound, the effective conductivity calculated by [Volume Average, Resistivity](#) is the lower bound, and the [Power Law](#) average is somewhere in between these two.

Three methods are available to compute the averaged electrical conductivity of the mixture.

VOLUME AVERAGE, PERMITTIVITY

If the relative permittivity of the two materials is not so different from each other, the effective relative permittivity ϵ_r is calculated by simple volume average:

$$\epsilon = \sum_{i=1}^n \theta_i \epsilon_i = \theta_1 \epsilon_1 + \theta_2 \epsilon_2 + \dots$$

where ϵ_i is the relative permeability of the material i .

If the permittivity is defined by second-order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RECIPROCAL PERMITTIVITY

The second method is the volume average of the inverse of the permittivities:

$$\frac{1}{\epsilon} = \sum_{i=0}^n \frac{\theta_i}{\epsilon_i} = \frac{\theta_1}{\epsilon_1} + \frac{\theta_2}{\epsilon_2} + \dots$$

If the permittivity is defined by a second-order tensor, the inverse of the tensor is used.

POWER LAW

A power law gives the following expression for the equivalent permittivity:

$$\epsilon = \prod_{i=0}^n \epsilon_i^{\theta_i} = \epsilon_1^{\theta_1} \epsilon_2^{\theta_2} \dots$$

The effective permeability calculated by [Volume Average, Permittivity](#) is the upper bound, the effective permeability calculated by [Volume Average, Reciprocal Permittivity](#) is the lower bound, and the [Power Law](#) average gives a value somewhere in between these two.

Three methods are available to compute the averaged electrical conductivity of the mixture.

VOLUME AVERAGE, PERMEABILITY

If the relative permeability of the two materials is not so different from each other, the effective relative permeability μ_r is calculated by simple volume average:

$$\mu = \sum_{i=1}^n \theta_i \mu_i = \theta_1 \mu_1 + \theta_2 \mu_2 + \dots$$

where μ_i is the relative permeability of the material i .

If the permeability is defined by second-order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RECIPROCAL PERMEABILITY

The second method is the volume average of the inverse of the permeabilities:

$$\frac{1}{\mu} = \sum_{i=0}^n \frac{\theta_i}{\mu_i} = \frac{\theta_1}{\mu_1} + \frac{\theta_2}{\mu_2} + \dots$$

If the permeability is defined by a second-order tensor, the inverse of the tensor is used.

POWER LAW

A power law gives the following expression for the equivalent permeability:

$$\mu = \prod_{i=0}^n \mu_i^{\theta_i} = \mu_1^{\theta_1} \mu_2^{\theta_2} \dots$$

The effective permeability calculated by [Volume Average, Permeability](#) is the upper bound, the effective permeability calculated by [Volume Average, Reciprocal Permeability](#) is the lower bound, and the [Power Law](#) average gives a value somewhere in between these two.

Archie's Law Theory

The electrical conductivity of the materials composing saturated rocks and soils can vary over many orders of magnitude. For instance, in the petroleum reservoirs, normal sea water (or brine) has a typical conductivity of around 3 S/m, whereas hydrocarbons are typically much more resistive and have conductivities in the range 0.1–0.01 S/m.

The porous rocks and sediments can have even lower conductivities. In variably saturated soils, the conductivity of air is roughly ten orders of magnitude lower than the ground water. A simple volume average (of either conductivity or resistivity) in rocks or soils might give different results compared to experimental data.

Since most crustal rocks, sedimentary rocks, and soils are formed by nonconducting materials, Archie (Ref. 1) assumed that electric current are mainly caused by ion fluxes through the pore network. Originally, Archie's law is an empirical law for the effective conductivity of a fully saturated rock or soil, but it can be extended to variably saturated porous media.

Archie's law relates the effective conductivity to the fluid conductivity σ_L , fluid saturation s_L , and porosity ϵ_p :

$$\sigma = s_L^n \epsilon_p^m \sigma_L$$

here, m is the cementation exponent, a parameter that describes the connectivity of the pores. The cementation exponent normally varies between 1.3 and 2.5 for most sedimentary rocks and is close to 2 for sandstones. The lower limit $m = 1$ represents a volume average of the conductivities of a fully saturated, insulating (zero conductivity) porous matrix, and a conducting fluid. The saturation coefficient n is normally close to 2. The ratio $F = \sigma_L/\sigma$ is called the *formation factor*.

Archie's law does not take care of the relative permittivity of either fluids or solids, so the effective relative permittivity of the porous medium is normally consider as $\epsilon_r = 1$.

Reference for Archie's Law

1. G.E. Archie, "The Electric Resistivity as an Aid in Determining Some Reservoir Characteristics," *Trans. Am. Inst. Metal. Eng.*, vol. 146, pp. 54–62, 1942.

Theory for the Transmission Line Interface

The [Transmission Line Interface](#) theory is described in this section.

- [Introduction to Transmission Line Theory](#)
- [Theory for the Transmission Line Boundary Conditions](#)

Introduction to Transmission Line Theory

[Figure 4-5](#) is an illustration of a transmission line of length L . The distributed resistance R , inductance L , conductance G , and capacitance C , characterize the properties of the transmission line.

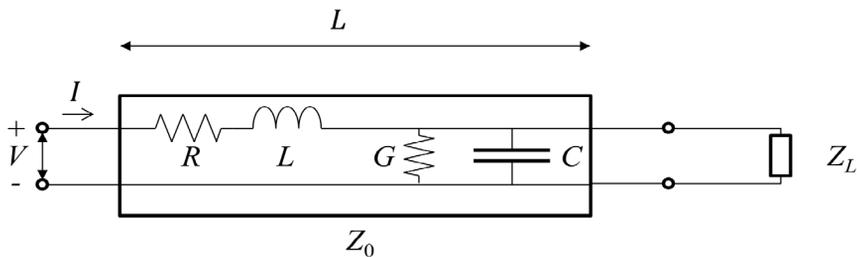


Figure 4-5: Schematic of a transmission line with a load impedance.

The distribution of the electric potential V and the current I describes the propagation of the signal wave along the line. The following equations relate the current and the electric potential

$$\frac{\partial V}{\partial x} = -(R + j\omega L)I \quad (4-1)$$

$$\frac{\partial I}{\partial x} = -(G + j\omega C)V \quad (4-2)$$

[Equation 4-1](#) and [Equation 4-2](#) can be combined to the second-order partial differential equation

$$\frac{\partial^2 V}{\partial x^2} = \gamma^2 V \quad (4-3)$$

where

$$\gamma = \sqrt{(R + j\omega L)(G + j\omega C)} = \alpha + j\beta$$

Here γ , α , and β are called the complex propagation constant, the attenuation constant, and the (real) propagation constant, respectively.



The attenuation constant, α , is zero if R and G are zero.

The solution to [Equation 4-3](#) represents a forward- and a backward-propagating wave

$$V(x) = V_+ e^{-\gamma x} + V_- e^{\gamma x} \quad (4-4)$$

By inserting [Equation 4-4](#) in [Equation 4-1](#) the current distribution is obtained.

$$I(x) = \frac{\gamma}{R + j\omega L} (V_+ e^{-\gamma x} - V_- e^{\gamma x})$$

If only a forward-propagating wave is present in the transmission line (no reflections), dividing the voltage by the current gives the characteristic impedance of the transmission line

$$Z_0 = \frac{V}{I} = \frac{R + j\omega L}{\gamma} = \sqrt{\frac{R + j\omega L}{G + j\omega C}}$$

To make sure that the current is conserved across interior boundaries, COMSOL Multiphysics solves the following wave equation (instead of [Equation 4-3](#))

$$\frac{\partial}{\partial x} \left(\frac{1}{R + j\omega L} \frac{\partial V}{\partial x} \right) - (G + j\omega C)V = 0 \quad (4-5)$$

Theory for the Transmission Line Boundary Conditions

The [Transmission Line Interface](#) has these boundary conditions:

$$V_1 = V_2 \quad (4-6)$$

and

$$I_1 = I_2 \quad (4-7)$$

In [Equation 4-6](#) and [Equation 4-7](#), the indices 1 and 2 denote the domains on the two sides of the boundary. The currents flowing out of a boundary are given by

$$I_i = -\frac{\mathbf{n}_i \cdot \nabla V_i}{R_i + j\omega L_i}, \quad i = 1, 2$$

where \mathbf{n}_i are the normals pointing out of the domain.

Because V is solved for, the electric potential is always continuous, and thus [Equation 4-6](#) is automatically fulfilled. [Equation 4-7](#) is equivalent to the natural boundary condition

$$\frac{1}{R_2 + j\omega L_2} \frac{\partial V}{\partial x} \Big|_2 - \frac{1}{R_1 + j\omega L_1} \frac{\partial V}{\partial x} \Big|_1 = 0$$

which is fulfilled with the wave-equation formulation in [Equation 4-5](#).

When the transmission line is terminated by a load impedance, as [Figure 4-5](#) shows, the current through the load impedance is given by

$$I(L) = \frac{V(L)}{Z_L} \quad (4-8)$$

Inserting [Equation 4-1](#) into [Equation 4-8](#), results in the [Terminating Impedance](#) boundary condition

$$\frac{1}{R + j\omega L} \frac{\partial V}{\partial x} + \frac{V}{Z_L} = 0 \quad (4-9)$$

If the arbitrary load impedance Z_L is replaced by the characteristic impedance of the transmission line Z_0 you get the [Absorbing Boundary](#) condition. By inserting the voltage, defined in [Equation 4-4](#), in [Equation 4-9](#) you can verify that the boundary condition does not allow any reflected wave (that is, V is zero).

The [Open Circuit](#) boundary condition is obtained by letting the load impedance become infinitely large, that is, no current flows through the load impedance.

On the other hand, the [Short Circuit](#) boundary condition specifies that the voltage at the load is zero. In COMSOL Multiphysics this is implemented as a constraint on the electric potential.

To excite the transmission line, use the [Incoming Wave](#) boundary condition. Referring to the left (input) end of the transmission line in [Figure 4-5](#), the forward propagating wave has a voltage amplitude of V_0 . Thus, the total voltage at this boundary is given by

$$V(0) = V = V_0 + V_.$$

Thereby, the current can be written as

$$I(0) = -\frac{1}{R + j\omega L} \frac{\partial V}{\partial x} \Big|_{x=0} = \frac{1}{Z_0} (V_0 - V_.) = \frac{2V_0 - V}{Z_0}$$

resulting in the boundary condition

$$-\frac{1}{R + j\omega L} \frac{\partial V}{\partial x} + \frac{V - 2V_0}{Z_0} = 0$$

For the [Lumped Port](#) boundary condition, the port current (positive when entering the transmission line) defines the boundary condition as

$$-\frac{1}{R + j\omega L} \frac{\partial V}{\partial x} - I_{\text{port}} = 0$$

where the port current I_{port} is given by

$$I_{\text{port}} = \frac{2V_0 - V}{Z_0}$$

for a Cable lumped port (see the [Lumped Port](#) section for a description of the lumped port settings).

For a Current-controlled lumped port, you provide I_{port} as an input parameter, whereas it is part of an electrical circuit equation for a Circuit-based lumped port.

Theory for the Electromagnetic Waves, Time Explicit Interface

The Electromagnetic Waves, Time Explicit Interface theory is described in this section:

- [The Equations](#)
- [In-Plane E Field or In-Plane H Field](#)
- [Fluxes as Dirichlet Boundary Conditions](#)
- [Absorbing Layers](#)

The Equations

Maxwell's equations are a set of equations, written in differential or integral form, stating the relationships between the fundamental electromagnetic quantities. These quantities are the:

- Electric field intensity \mathbf{E}
- Electric displacement or electric flux density \mathbf{D}
- Magnetic field intensity \mathbf{H}
- Magnetic flux density \mathbf{B}
- Current density \mathbf{J}
- Electric charge density ρ

For general time-varying fields, the differential form of Maxwell's equations can be written as

$$\begin{aligned}\nabla \times \mathbf{H} &= \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \\ \nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t} \\ \nabla \cdot \mathbf{D} &= \rho \\ \nabla \cdot \mathbf{B} &= 0\end{aligned}\tag{4-10}$$

The first two equations are also called Maxwell-Ampère's law and Faraday's law, respectively. Equation three and four are two forms of Gauss' law, the electric and magnetic form, respectively.

CONSTITUTIVE RELATIONS

To obtain a closed system of equations, the constitutive relations describing the macroscopic properties of the medium are included. These are given as

$$\begin{aligned}\mathbf{D} &= \epsilon_0 \mathbf{E} + \mathbf{P} \\ \mathbf{B} &= \mu_0 (\mathbf{H} + \mathbf{M}) \\ \mathbf{J} &= \sigma \mathbf{E}\end{aligned}\tag{4-11}$$

Here ϵ_0 is the permittivity of a vacuum, μ_0 is the permeability of a vacuum, and σ the electric conductivity of the medium. In the SI system, the permeability of a vacuum is chosen to be $4\pi \cdot 10^{-7}$ H/m. The velocity of an electromagnetic wave in a vacuum is given as c_0 and the permittivity of a vacuum is derived from the relation

$$\epsilon_0 = \frac{1}{c_0^2 \mu_0} = 8.854 \cdot 10^{-12} \text{ F/m} \approx \frac{1}{36\pi} \cdot 10^{-9} \text{ F/m}$$

The electric polarization vector \mathbf{P} describes how the material is polarized when an electric field \mathbf{E} is present. It can be interpreted as the volume density of electric dipole moments. \mathbf{P} is generally a function of \mathbf{E} . Some materials might have a nonzero \mathbf{P} also when there is no electric field present.

The magnetization vector \mathbf{M} similarly describes how the material is magnetized when a magnetic field \mathbf{H} is present. It can be interpreted as the volume density of magnetic dipole moments. \mathbf{M} is generally a function of \mathbf{H} . Permanent magnets, for example, have a nonzero \mathbf{M} also when there is no magnetic field present.

To get a wave equation for the \mathbf{E} field, for example, take the curl of the second equation in [Equation 4-10](#) (previously divided by μ_0), and insert it into the time derivative of the first row in [Equation 4-10](#)

$$-\nabla \times \left(\frac{1}{\mu_0} \nabla \times \mathbf{E} + \frac{\partial \mathbf{M}}{\partial t} \right) = \sigma \frac{\partial \mathbf{E}}{\partial t} + \epsilon_0 \frac{\partial^2 \mathbf{E}}{\partial t^2} + \frac{\partial^2 \mathbf{P}}{\partial t^2}$$

this is referred as curl-curl formulation in the literature (second order time derivatives and second order space derivatives).

LINEAR MATERIALS

In the simplest case linear materials, the polarization is directly proportional to the electric field, that is

$$\partial \mathbf{P} / \partial \mathbf{E} = \epsilon_0 \chi_e \quad \text{and} \quad \mathbf{P} = \epsilon_0 \chi_e \mathbf{E}$$

where χ_e is the electric susceptibility (which can be a scalar or a second-rank tensor). Similarly, the magnetization is directly proportional to the magnetic field, or

$$\partial \mathbf{M} / \partial \mathbf{H} = \chi_m \text{ and } \mathbf{M} = \chi_m \mathbf{H}$$

where χ_m is the magnetic susceptibility.

As a consequence, for linear materials, the constitutive relations in Equation 4-11 can be written as

$$\begin{aligned} \mathbf{D} &= \epsilon_0 \mathbf{E} + \mathbf{P} = \epsilon_0(1 + \chi_e) \mathbf{E} = \epsilon_0 \epsilon_r \mathbf{E} \\ \mathbf{B} &= \mu_0(\mathbf{H} + \mathbf{M}) = \mu_0(1 + \chi_m) \mathbf{H} = \mu_0 \mu_r \mathbf{H} \end{aligned}$$

Here, $\epsilon = \epsilon_0 \epsilon_r$ and $\mu = \mu_0 \mu_r$ are the permittivity and permeability of the material. The relative permittivity ϵ_r and the relative permeability μ_r are usually scalar properties but these can be second-rank symmetric (Hermitian) tensors for a general anisotropic material.

For general time-varying fields, Maxwell's equations in linear materials described in Equation 4-10 can be simplified to Maxwell-Ampère's law and Faraday's law:

$$\begin{aligned} \nabla \times \mathbf{H} &= \sigma \mathbf{E} + \epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} \\ \nabla \times \mathbf{E} &= -\mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} \end{aligned} \tag{4-12}$$

The electric conductivity σ can also be a scalar or a second-rank tensor. Another important assumption is that the relative permittivity ϵ_r , the relative permeability μ_r and the electric conductivity σ might change with position and orientation (inhomogeneous or anisotropic materials) but not with time.

FIRST-ORDER IMPLEMENTATION OF MAXWELL'S EQUATIONS

In order to accommodate Maxwell's equations in the coefficients for the Wave Form PDE interface in the form

$$d_a \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \Gamma(\mathbf{u}) = \mathbf{f}$$

the curl of a vector is written in divergence form as

$$\nabla \times \mathbf{u} = \nabla \cdot \begin{bmatrix} 0 & u_3 & -u_2 \\ -u_3 & 0 & u_1 \\ u_2 & -u_1 & 0 \end{bmatrix} \quad (4-13)$$

where the divergence is applied on each row of the flux $\Gamma(\mathbf{u})$.

Maxwell's equations in 3D

$$\begin{aligned} \epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} &= -\sigma \mathbf{E} \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \times \mathbf{E} &= \mathbf{0} \end{aligned}$$

are then accommodated to the Wave Form PDE as

$$\begin{aligned} d_E \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= \mathbf{f} \\ d_H \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= \mathbf{0} \end{aligned}$$

with the “mass” coefficients

$$d_E = \epsilon_0 \epsilon_r \quad \text{and} \quad d_H = \mu_0 \mu_r$$

the “flux” terms

$$\Gamma_E(\mathbf{H}) = - \begin{bmatrix} 0 & h_3 & -h_2 \\ -h_3 & 0 & h_1 \\ h_2 & -h_1 & 0 \end{bmatrix} \quad \text{and} \quad \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 & -e_2 \\ -e_3 & 0 & e_1 \\ e_2 & -e_1 & 0 \end{bmatrix}$$

and the “source” term $\mathbf{f} = -\sigma \mathbf{E}$.

THE LAX-FRIEDRICHS FLUX PARAMETERS



When using SI units (or other) for the electromagnetic fields and material properties, the Lax-Friedrichs flux parameter is not dimensionless and must have units of $\tau_E = 1/(2Z)$ for Ampère's law and $\tau_H = Z/2$ for Faraday's law, where Z is the impedance of the medium.

In-Plane E Field or In-Plane H Field

In the general case, in 2D and 2D axisymmetric, solving for three variables for each field is still required. The “in-plane H” or “in-plane E” assumption simplifies the problem to only three dependent variables.

TM WAVES IN 2D

For TM waves in 2D, solve for an in-plane electric field vector and one out-of-plane variable for the magnetic field. Maxwell’s equations then read

$$\begin{aligned}\epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma \cdot \mathbf{E} \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= 0\end{aligned}\tag{4-14}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} 0 & -h_3 \\ h_3 & 0 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} e_2 & -e_1 \end{bmatrix}\tag{4-15}$$

The divergence on $\Gamma_E(\mathbf{H})$ is applied row-wise. The conductivity and permittivity tensors σ and ϵ_r represent in-plane material properties, while the relative permeability μ_r is an out-of-plane scalar property.

The default Lax-Friedrichs flux parameters are $\tau_E = 1/(2Z)$ for Ampère’s law, and the scalar $\tau_H = Z/2$ for Faraday’s law, where Z is the impedance of a vacuum.

TE WAVES IN 2D

For TE waves in 2D, solve for an in-plane magnetic field vector and one out-of-plane variable for the electric field. Maxwell’s equations then read

$$\begin{aligned}\epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma \mathbf{E} \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= 0\end{aligned}\tag{4-16}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} -h_2 & h_1 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 \\ -e_3 & 0 \end{bmatrix}\tag{4-17}$$

The divergence of $\Gamma_H(\mathbf{E})$ is applied row-wise. The tensor of relative permeability μ_r represents in-plane material properties, while the relative permittivity ϵ_r and conductivity σ are out-of-plane scalar properties.

The default Lax-Friedrichs flux parameters are $\tau_E = 1/(2Z)$ for Ampère's law, and two scalar $\tau_H = Z/2$ for Faraday's law, where Z is the impedance of a vacuum.

Fluxes as Dirichlet Boundary Conditions

Consider Maxwell's equations in 3D

$$\begin{aligned}\epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma \mathbf{E} \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= \mathbf{0}\end{aligned}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} 0 & -h_3 & h_2 \\ h_3 & 0 & -h_1 \\ -h_2 & h_1 & 0 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 & -e_2 \\ -e_3 & 0 & e_1 \\ e_2 & -e_1 & 0 \end{bmatrix}$$

and the divergence on $\Gamma_E(\mathbf{H})$ and $\Gamma_H(\mathbf{E})$ applied row-wise.

For Ampère's law, the normal to the flux term on exterior boundaries reads

$$\mathbf{n} \cdot \Gamma_E(\mathbf{H}) = -\mathbf{n} \times \mathbf{H}$$

and for Faraday's law

$$\mathbf{n} \cdot \Gamma_H(\mathbf{E}) = \mathbf{n} \times \mathbf{E}$$

which means that normal fluxes on external boundaries can only prescribe tangential components for the fields.

BOUNDARY CONDITIONS

The boundary conditions for outer boundaries are computed from the normal fluxes $\mathbf{n} \cdot \Gamma_H(\mathbf{E})$ and $\mathbf{n} \cdot \Gamma_E(\mathbf{H})$.

- Perfect electric conductor $\mathbf{n} \times \mathbf{E} = \mathbf{0}$, or zero tangential components for \mathbf{E} , is obtained by setting $\mathbf{n} \cdot \Gamma_H(\mathbf{E}) = \mathbf{0}$.

- Perfect magnetic conductor $\mathbf{n} \times \mathbf{H} = \mathbf{0}$, or zero tangential components for \mathbf{H} , is obtained by prescribing $\mathbf{n} \cdot \Gamma_E(\mathbf{H}) = \mathbf{0}$.
- Electric field $\mathbf{n} \times \mathbf{E} = \mathbf{n} \times \mathbf{E}_0$, or $\mathbf{n} \cdot \Gamma_H(\mathbf{E}) = \mathbf{n} \times \mathbf{E}_0$.
- Magnetic field $\mathbf{n} \times \mathbf{H} = \mathbf{n} \times \mathbf{H}_0$, or $-\mathbf{n} \cdot \Gamma_E(\mathbf{H}) = \mathbf{n} \times \mathbf{H}_0$.
- For external boundaries, the surface currents BC means $\mathbf{n} \times \mathbf{H} = \mathbf{J}_s$, or $-\mathbf{n} \cdot \Gamma_E(\mathbf{H}) = \mathbf{J}_s$.

ABSORBING BOUNDARY CONDITION

A simple absorbing boundary can be implemented by setting $\mathbf{n} \times \mathbf{E} = Z\mathbf{H}$.

Absorbing Layers

The [Electromagnetic Waves, 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. 1](#).

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 hit the outer boundary in the normal direction. For the Absorbing Layer domain selection, add an additional [Wave Equations](#) feature, mark the **Activate** check box under the [Filter Parameters](#) section, and enter filter parameters. Filtering attenuates and filters out high-frequency components of the wave. Finally, at the outer boundary of the layer add a simple [Scattering Boundary Condition](#) 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*.

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 detailed on the **Geometry** and **Scaling** see the [Infinite Elements, Perfectly Matched Layers, and Absorbing Layers](#) 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 plot the coordinate variables `x_absorb_ab1`, `y_absorb_ab1`, and `z_absorb_ab1`. 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.



For an example of a filter parameter combination that can be used for a [Wave Equations](#) feature on an **Absorbing Layer** domain selection see the [Filter Parameters](#) section for the [Wave Equations](#) feature.

Reference

I. 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.

AC/DC Interfaces

This chapter summarizes the functionality of the electrical circuit interface found under the **AC/DC** branch () when adding a physics interface.

In this chapter:

- [The Electrical Circuit Interface](#)
- [Theory for the Electrical Circuit Interface](#)

See [The Electromagnetics Interfaces](#) in the *COMSOL Multiphysics Reference Manual* for other AC/DC interface and feature node settings.

The Electrical Circuit Interface

The **Electrical Circuit (cir)** interface () , found under the **AC/DC** branch () when adding a physics interface, is used to model currents and voltages in circuits including voltage and current sources, resistors, capacitors, inductors, and semiconductor devices. Models created with the Electrical Circuit interface can include connections to distributed field models. The physics interface supports stationary, frequency-domain and time-domain modeling and solves Kirchhoff's conservation laws for the voltages, currents and charges associated with the circuit elements.

When this physics interface is added, it adds a default **Ground Node** feature and associates that with node zero in the electrical circuit.



Circuit nodes are nodes in the electrical circuit (electrical nodes) and should not be confused with nodes in the Model Builder tree of the COMSOL Multiphysics software. Circuit node names are not restricted to numerical values but can contain alphanumeric characters.

DEVICE NAMES

Each circuit component has an associated **Device name**, which is constructed from a prefix identifying the type of the device and a string. The string can be specified in the feature's **Settings** window. The **Device name** is used to identify variables defined by the component, and for the SPICE Import and Export functionality.

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 `cir`.

RESISTANCE IN PARALLEL TO PN JUNCTIONS

For numerical stability, a large resistance is added automatically in parallel to the pn junctions in diodes and BJT devices. Enter a default value for the **Resistance in parallel to pn junctions** R_j (SI unit: Ω). The default value is $1 \cdot 10^{12} \Omega$.

CREATE UNIQUE NODES FOR NEW DEVICES

When this setting is selected (the default), newly added devices will be assigned unused node names. The devices will be disconnected from the rest of the circuit and the nodes should be updated to reflect the actual circuit connections. When this setting is deselected, new devices will be connected to the lowest-numbered nodes starting from 0.



- [Theory for the Electrical Circuit Interface](#)
 - [Connecting to Electrical Circuits](#)
-

ELECTRICAL CIRCUIT TOOLBAR

The following nodes are available from the **Electrical Circuit** ribbon toolbar (Windows users), **Electrical Circuit** context menu (Mac or Linux users), or right-click to access the context menu (all users):



For step-by-step instructions and general documentation descriptions, this is the **Electrical Circuit** toolbar.

- Ground Node
- Voltmeter
- Ampère Meter
- Resistor
- Capacitor
- Inductor
- Voltage Source
- Current Source
- Diode
- Voltage-Controlled Voltage Source¹
- Voltage-Controlled Current Source¹
- Current-Controlled Voltage Source¹
- Current-Controlled Current Source¹
- Subcircuit Definition
- Subcircuit Instance
- Mutual Inductance
- Transformer
- NPN BJT and PNP BJT²
- n-Channel MOSFET and p-Channel MOSFET²
- External I vs. U³
- External U vs. I³
- External I-Terminal³
- SPICE Circuit Import
- SPICE Circuit Export

¹ Selected from the **Dependent Sources** submenu when you right-click main node.

² Selected from the **Transistors** submenu when you right-click main node.

³ Selected from the **External Couplings** submenu when you right-click main node.

Ground Node

The **Ground Node** () feature adds a ground node with the default node number zero to the electrical circuit. This is the default node in the Electrical Circuit interface. More ground nodes can be added but those must have unique node numbers and are by default given higher node numbers.

GROUND CONNECTION

Set the **Node name** for the ground node in the circuit. The convention is to use 0 (zero) for the ground node. If adding more ground nodes, each must have a unique node name (number).

Voltmeter

The **Voltmeter** () feature connects a voltmeter (voltage measurement device) between two nodes in the electrical circuit. A voltmeter behaves electrically as an open circuit. The voltmeter node adds a Probe sampling the voltage across it.

DEVICE NAME

Enter a **Device name** for the voltmeter. The name is also used in the probe.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

Ampère Meter

The **Ammeter** () feature connects an ammeter (current measurement device) between two nodes in the electrical circuit. An ammeter behaves electrically as a short circuit. The ammeter node adds a Probe sampling the current through it.

DEVICE NAME

Enter a Device name for the ammeter. The name is also used in the probe.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Resistor

The **Resistor** () feature connects a resistor between two nodes in the electrical circuit.

DEVICE NAME

Enter a **Device name** for the resistor. The prefix is R.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Capacitor

The **Capacitor** () feature connects a capacitor between two nodes in the electrical circuit.

DEVICE NAME

Enter a **Device name** for the capacitor. The prefix is C.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the capacitor.

DEVICE PARAMETERS

Enter the **Capacitance** of the capacitor.

Inductor

The **Inductor** () feature connects an inductor between two nodes in the electrical circuit.

DEVICE NAME

Enter a Device name for the inductor. The prefix is L.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the inductor.

DEVICE PARAMETERS

Enter the **Inductance** of the inductor.

Voltage Source

The **Voltage Source** () feature connects a voltage source between two nodes in the electrical circuit.

DEVICE NAME

Enter a **Device name** for the voltage source. The prefix is V.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal.

DEVICE PARAMETERS

Enter the **Source type** that should be adapted to the selected study type. It can be **DC-source**, **AC-source**, or a time-dependent **Sine source** or **Pulse source**. Depending on the choice of source, also specify the following parameters:

- For a DC-source, the **Voltage V_{src}** (default value: 1 V). DC-sources are active in Stationary and Time-Dependent studies.
- For an AC-source: the **Voltage V_{src}** (default value: 1 V) and the **Phase Θ** (default value: 0 rad). AC-sources are active in Frequency Domain studies only.
- For a sine source: the **Voltage V_{src}** (default value: 1 V), the **Offset V_{off}** (default value: 0 V), the **Frequency** (default value: 1 kHz), and the **Phase Θ** (default value: 0 rad). The sine sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.
- For a pulse source: the **Voltage V_{src}** (default value: 1 V), the **Offset V_{off}** (default value: 0 V), the **Delay t_d** (default value: 0s), the **Rise time t_r** and **Fall time t_f** (default values: 0 s), the **Pulse width p_w** (default value: 1 μs), and the **Period T_{per}** (default value: 2 μs). The pulse sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.

All values are peak values rather than RMS.



For the AC source, the frequency is a global input set by the solver. AC sources should be used in Frequency-domain studies only. Do not use the **Sine source** unless the model is time-dependent.

Current Source

The **Current Source** () feature connects a current source between two nodes in the electrical circuit.

DEVICE NAME

Enter a **Device name** for the current source. The prefix is I.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the current source. The first node represents the positive reference terminal from where the current flows through the source to the second node.

DEVICE PARAMETERS

Enter the **Source type** that should be adapted to the selected study type. It can be **DC-source**, **AC-source**, or a time-dependent **Sine source** or **Pulse source**. Depending on the choice of source, also specify the following parameters:

- For a DC-source, the **Current i_{src}** (default value: 1 A). DC-sources are active in Stationary and Time-Dependent studies.
- For an AC-source: the **Current i_{src}** (default value: 1 A) and the **Phase Θ** (default value: 0 rad). AC-sources are active in Frequency Domain studies only.
- For a sine source: the **Current i_{src}** (default value: 1 A), the **Offset i_{off}** (default value: 0 A), the **Frequency** (default value: 1 kHz), and the **Phase Θ** (default value: 0 rad). The sine sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.
- For a pulse source: the **Current i_{src}** (default value: 1 A), the **Offset i_{off}** (default value: 0 A), the **Delay t_d** (default value: 0 s), the **Rise time t_r** and **Fall time t_f** (default values: 0 s), the **Pulse width p_w** (default value: 1 μ s), and the **Period T_{per}** (default value: 2 μ s). The pulse sources are active in Time-Dependent studies and also in Stationary studies, providing that a value for t has been provided as a model parameter or global variable.

All values are peak values rather than RMS.



For the AC source, the frequency is a global input set by the solver. AC sources should be used in frequency-domain studies only. Do not use the **Sine source** unless the model is time-dependent.

Voltage-Controlled Voltage Source

The **Voltage-Controlled Voltage Source** () feature connects a voltage-controlled voltage source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

DEVICE NAME

Enter a **Device name** for the voltage-controlled voltage source. The prefix is E.

NODE CONNECTIONS

Specify four **Node names**: the first pair for the connection nodes for the voltage source and the second pair defining the input control voltage. The first node in a pair represents the positive reference terminal.

DEVICE PARAMETERS

Enter the voltage **Gain**. The resulting voltage is this number multiplied by the control voltage.

Voltage-Controlled Current Source

The **Voltage-Controlled Current Source** () feature connects a voltage-controlled current source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

DEVICE NAME

Enter a **Device name** for the voltage-controlled current source. The prefix is G.

NODE CONNECTIONS

Specify four **Node names**: the first pair for the connection nodes for the current source and the second pair defining the input control voltage. The first node in a pair represents the positive voltage reference terminal or the one from where the current flows through the source to the second node.

DEVICE PARAMETERS

Enter the source **Gain** (SI units: S). The resulting current is this number multiplied by the control voltage. It represents the transconductance of the source.

Current-Controlled Voltage Source

The **Current-Controlled Voltage Source** () feature connects a current-controlled voltage source between two nodes in the electrical circuit. The input control current is the one flowing through a two-pin device.

DEVICE NAME

Enter a **Device name** for the current-controlled voltage source. The prefix is H.

NODE CONNECTIONS

Set two **Node names** for the connection nodes for the voltage source. The first node in a pair represents the positive reference terminal.

DEVICE PARAMETERS

Enter the voltage **Gain** and select the **Device** whose current is taken as the control current. The resulting voltage is this number multiplied by the control current through the named **Device** (any two-pin device). Thus it formally has the unit of resistance.

Current-Controlled Current Source

The **Current-Controlled Current Source** () feature connects a current-controlled current source between two nodes in the electrical circuit. The input control current is the one flowing through a named device that must be a two-pin device.

DEVICE NAME

Enter a **Device name** for the current-controlled current source. The prefix is F.

NODE CONNECTIONS

Specify two **Node names** for the connection nodes for the current source. The first node in a pair represents the positive reference terminal from where the current flows through the source to the second node.

DEVICE PARAMETERS

Enter the current **Gain** and select the **Device** whose current is taken as the control current. The resulting current is this number multiplied by the control current through the **Device**.

Subcircuit Definition

The **Subcircuit Definition** () feature is used to define subcircuits, which can be inserted as devices into the main circuit using **Subcircuit Instance** nodes. Create the subcircuit by adding subnodes to the **Subcircuit Definition** node, either by using the **Physics** toolbar, or by right-clicking the **Subcircuit Definition**.

SUBCIRCUIT PINS

Define the **Pin names** at which the subcircuit connects to the main circuit or to other subcircuits when referenced by a **Subcircuit Instance** node. The **Pin names** refer to circuit nodes in the subcircuit. The order in which the **Pin names** are defined is the

order in which they are referenced by a Subcircuit Instance node. The devices constituting the subcircuit should be connected only to the subcircuit's pins and to themselves.

Subcircuit Instance

The **Subcircuit Instance** () feature represents an instance of a subcircuits defined by a [Subcircuit Definition](#) feature.

DEVICE NAME

Enter a **Device name** for the subcircuit instance. The prefix is X.

NODE CONNECTIONS

Select the **Name of subcircuit link** from the list of defined subcircuits in the circuit model and the circuit **Node names** at which the subcircuit instance connects to the main circuit or to another subcircuit if used therein.

NPN BJT and PNP BJT

The **NPN BJT** () and the **PNP BJT** () device models are large-signal models for bipolar junction transistors (BJT). It is an advanced device model and no thorough description and motivation of the many input parameters are attempted here. Many device manufacturers provide model input parameters for this BJT model. For any particular make of BJT, the device manufacturer should be the primary source of information.

DEVICE NAME

Enter a **Device name** for the BJT. The prefix is Q.

NODE CONNECTIONS

Specify three **Node names** for the connection nodes for the **BJT** device. These represent the *collector*, *base*, and *emitter* nodes for the **NPN** transistor, and the *emitter*, *base*, and *collector* nodes for the **PNP** transistor.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular BJT, the device manufacturer should be the primary source of information.



The interested reader is referred to [Ref. 1](#) for more details on semiconductor modeling within circuits.

For an explanation of the **Model Parameters** see [Bipolar Transistors](#).

n-Channel MOSFET and p-Channel MOSFET

The **n-Channel MOSFET** () and the **p-Channel MOSFET** () device models are large-signal models for, respectively, an n-Channel MOS field-effect transistor (MOSFET) and p-Channel MOSFET. These are advanced device models and no thorough description and motivation of the many input parameters are attempted here. Many device manufacturers provide model parameters for the MOSFET models. For any particular make of MOSFET, the device manufacturer should be the primary source of information.

DEVICE NAME

Enter a **Device name** for the MOSFET. The prefix is M.

NODE CONNECTIONS

Specify four **Node names** for the connection nodes for the **n-Channel MOSFET** or **p-Channel MOSFET** device. These represent the *drain*, *gate*, *source*, and *bulk* nodes, respectively.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular MOSFET, the device manufacturer should be the primary source of information.



The interested reader is referred to [Ref. 1](#) for more details on semiconductor modeling within circuits.

For an explanation of the **Model Parameters** see [MOSFET Transistors](#).

Mutual Inductance

The **Mutual Inductance** allows specifying a coupling between two existing **Inductor** features in the circuit. The mutual inductance of the coupling is

$$M = k\sqrt{L_1L_2}$$

where k is the coupling factor and L_1 and L_2 are the inductances of the inductors.

DEVICE PARAMETERS

Enter values or expressions for the:

- **Coupling factor** k (dimensionless). The value must be between 0 and 1, and the default is 0.98.
- **First inductance** L_1 (SI unit: H) and **Second inductance** L_2 (SI unit: H). These must be set to two different **Inductor** features in the circuit.

Transformer

The **Transformer** feature represents either a combination of two **Inductor** and a **Mutual Inductance** features, or an ideal transformer.

NODE CONNECTIONS

Enter or edit the table in the **Node names** column for the **primary** and **secondary** node connections.

DEVICE PARAMETERS

Choose a **Transformer model** — **Specify inductors** (the default) or **Ideal transformer**.

For **Specify inductors** enter values or expressions for the:

- **Coupling factor** k (dimensionless). The default is 0.98.
- **First inductance** L_1 (SI unit: H). The default is 1 mH.
- **Second inductance** L_2 (SI unit: H). The default is 1 mH.

For **Ideal transformer** enter values or expressions for the **Winding ratio** N_1/N_2 (dimensionless). The default is 10.

Diode

The **Diode** device model () is a large-signal model for a diode. It is an advanced device model and no thorough description and motivation of the many input parameters are attempted here. The interested reader is referred to [Ref. 1](#) for more details on semiconductor modeling within circuits. Many device manufacturers provide model parameters for this diode model. For any particular make of diode, the device manufacturer should be the primary source of information.

DEVICE NAME

Enter a **Device name** for the diode. The prefix is D.

NODE CONNECTIONS

Specify two **Node names** for the positive and negative nodes for the **Diode** device.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular diode, the device manufacturer should be the primary source of information.



For an explanation of the **Model Parameters** see [Diode](#).

External I vs. U

The **External I vs. U** () feature connects an arbitrary voltage measurement (for example, a circuit terminal or circuit port boundary or a coil domain from another physics interface) as a voltage source between two nodes in the electrical circuit. The resulting circuit current from the first node to the second node is typically coupled back as a prescribed current source in the context of the voltage measurement.

DEVICE NAME

Enter a **Device name** for the External I vs. U node.

NODE CONNECTIONS

Specify the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal.

EXTERNAL DEVICE

Enter the source of the **Voltage**. If circuit or current excited terminals or circuit ports are defined on boundaries or domains or a multiturn coil domains is defined in other physics interfaces, these display as options in the **Voltage** list. Also select the **User defined** option and enter your own voltage variable, for example, using a suitable coupling operator. For inductive or electromagnetic wave propagation models, the voltage measurement must be performed as an integral of the electric field because the electric potential only does not capture induced EMF. Also the integration must be performed over a distance that is short compared to the local wavelength.



Except when coupling to a circuit terminal, circuit port, or coil, the current flow variable must be manually coupled back in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention `cirn.IvsUm_i`, where `cirn` is the tag of the Electrical Circuit interface node and `IvsUm` is the tag of the **External I vs. U** node. The tags are typically displayed within curly brackets `{}` in the Model Builder.



[Nonlocal Couplings and Coupling Operators in the COMSOL Multiphysics Reference Manual](#)

External U vs. I

The **External U vs. I** () feature connects an arbitrary current measurement (for example, a coil domain from another physics interface) as a current source between two nodes in the electrical circuit. The resulting circuit voltage between the first node and the second node is typically coupled back as a prescribed voltage source in the context of the current measurement.

DEVICE NAME

Enter a **Device name** for the External U vs. I node.

NODE CONNECTIONS

Specify the two **Node names** for the connecting nodes for the current source. The current flows from the first node to the second node.

EXTERNAL DEVICE

Enter the source of the **Current**. Voltage excited terminals or lumped ports defined on boundaries in other physics interfaces are natural candidates but do not appear as options in the **Voltage** list because those do not have an accurate built-in current measurement variable. A **User defined** option must be selected and a current variable entered, for example, using a suitable coupling operator.



The voltage variable must be manually coupled back in the electrical circuit to the context of the current measurement. This applies also when coupling to a voltage excited terminal or lumped port. The name of this voltage variable follows the convention `cirn.UvsIm_v`, where `cirn` is the tag of the Electrical Circuit interface node and `UvsIm` is the tag of the **External U vs. I** node. The tags are typically displayed within curly brackets `{ }` in the Model Builder.



[Nonlocal Couplings and Coupling Operators in the COMSOL Multiphysics Reference Manual](#)

External I-Terminal

The **External I-Terminal** () feature connects an arbitrary voltage-to-ground measurement (for example, a circuit terminal from another physics interface) as a voltage-to-ground assignment to a node in the electrical circuit. The resulting circuit current from the node is typically coupled back as a prescribed current source in the context of the voltage measurement. This node does not apply when coupling to inductive or electromagnetic wave propagation models because then voltage must be defined as a line integral between two points rather than a single point measurement of electric potential. For such couplings, use the **External I vs. U** node instead.

DEVICE NAME

Enter a **Device name** for the External I-terminal.

NODE CONNECTIONS

Set the **Node name** for the connecting node for the voltage assignment.

EXTERNAL TERMINAL

Enter the source of the **Voltage**. If circuit- or current-excited terminals are defined on boundaries in other physics interfaces, these display as options in the **Voltage** list. Also select the **User defined** option and enter a voltage variable, for example, using a suitable coupling operator.



Except when coupling to a circuit terminal, the current flow variable must be manually coupled back in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention `cirn.termIm_i`, where `cirn` is the tag of the Electrical Circuit interface node and `termIm` is the tag of the **External I-Terminal** node. The tags are typically displayed within curly brackets `{ }` in the Model Builder.



[Nonlocal Couplings and Coupling Operators](#) in the *COMSOL Multiphysics Reference Manual*.

SPICE Circuit Import

Right-click the **Electrical Circuit** () feature node to import an existing SPICE netlist (select **Import Spice Netlist**). A window opens — enter a file location or browse your directories to find one. The default file extension for a SPICE netlist is `.cir`. The SPICE circuit import translates the imported netlist into Electrical Circuit interface nodes so these define the subset of SPICE features that can be imported.



See [SPICE Import and Export](#) about the supported SPICE commands.

SPICE Circuit Export

Right-click the **Electrical Circuit** () feature node to export the current circuit to the SPICE netlist file format (select **Export Spice Netlist** ). A window opens — enter a

file location or browse your directories to find one. The default file extension for a SPICE netlist is `.cir`. The compatible circuit nodes are exported as SPICE devices



See [SPICE Export](#) for more details on the supported SPICE commands.

Theory for the Electrical Circuit Interface

The [Electrical Circuit Interface](#) theory is discussed in this section:

- [Electric Circuit Modeling and the Semiconductor Device Models](#)
- [Bipolar Transistors](#)
- [MOSFET Transistors](#)
- [Diode](#)
- [Reference for the Electrical Circuit Interface](#)



[Connecting to Electrical Circuits](#)

Electric Circuit Modeling and the Semiconductor Device Models

Electrical circuit modeling capabilities are useful when simulating all sorts of electrical and electromechanical devices ranging from heaters and motors to advanced plasma reactors in the semiconductor industry. There are two fundamental ways that an electrical circuit model relates to a physical field model.

- The field model is used to get a better, more accurate description of a single device in the electrical circuit model.
- The electrical circuit is used to drive or terminate the device in the field model in such a way that it makes more sense to simulate both as a tightly coupled system.

The Electrical Circuit interface makes it possible to add nodes representing circuit elements directly to the Model Builder tree in a COMSOL Multiphysics model. The circuit variables can then be connected to a physical device model to perform co-simulations of circuits and multiphysics. The model acts as a device connected to the circuit so that its behavior is analyzed in larger systems.

The fundamental equations solved by the Electrical Circuit interface are Kirchhoff's circuit laws, which in turn can be deduced from Maxwell's equations. The supported study types are Stationary, Frequency Domain, and Time Dependent.

There are three more advanced large-signal semiconductor device features available in the Electrical Circuit interface. The equivalent circuits and the equations defining their non-ideal circuit elements are described in this section. For a more complete treatise on semiconductor device modeling see [Ref. 1](#).

Bipolar Transistors

Figure 5-1 illustrates the equivalent circuit for the npn bipolar junction transistor.

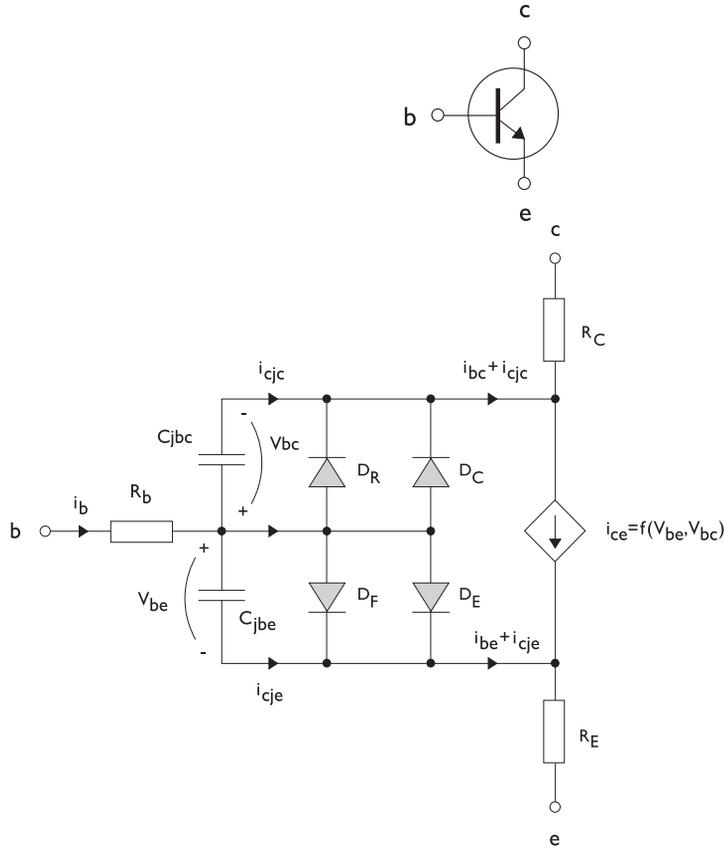


Figure 5-1: A circuit for the bipolar transistor.

The pnp transistor model is similar in all regards to the npn transistor, with the difference that the polarities of the currents and voltages involved are reversed. The following equations are used to compute the relations between currents and voltages in the circuit.

$$v_{rb} = \frac{1}{A} \left(R_{BM} - \frac{R_B - R_{BM}}{f_{bq}} \right) i_b$$

$$f_{bq} = \frac{1}{2 \left(1 - \frac{v_{bc}}{V_{AF}} - \frac{v_{be}}{V_{AR}} \right)} \left(1 + \sqrt{1 + 4I_S \left(\frac{e^{\frac{v_{be}}{N_F V_T}} - 1}{I_{KF} A} + \frac{e^{\frac{v_{bc}}{N_R V_T}} - 1}{I_{KR} A} \right)} \right)$$

$$i_{be} = A \left(\frac{I_S}{B_F} \left(e^{\frac{v_{be}}{N_F V_T}} - 1 \right) + I_{SE} \left(e^{\frac{v_{be}}{N_E V_T}} - 1 \right) \right)$$

$$i_{bc} = A \left(\frac{I_S}{B_R} \left(e^{\frac{v_{bc}}{N_R V_T}} - 1 \right) + I_{SC} \left(e^{\frac{v_{bc}}{N_C V_T}} - 1 \right) \right)$$

$$i_{ce} = A \left(\frac{I_S}{f_{bq}} \left(e^{\frac{v_{be}}{N_F V_T}} + e^{\frac{v_{bc}}{N_C V_T}} \right) \right)$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also two capacitances that use the same formula as the junction capacitance of the diode model. In the parameter names below, replace x with C for the base-collector capacitance and E for the base-emitter capacitance.

$$C_{jbx} = AC_{Jx} \times \begin{cases} \left(1 - \frac{v_{bx}}{V_{Jx}} \right)^{-M_{Jx}} & v_{bx} < F_C V_{Jx} \\ (1 - F_C)^{-1 - M_{Jx}} \left(1 - F_C (1 + M_{Jx}) + M_{Jx} \frac{v_{bx}}{V_{Jx}} \right) & v_{bx} \geq F_C V_{Jx} \end{cases}$$

The model parameters are listed in the table below.

TABLE 5-1: BIPOLAR TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
B_F	100	Ideal forward current gain
B_R	1	Ideal reverse current gain
C_{JC}	0 F/m ²	Base-collector zero-bias depletion capacitance
C_{JE}	0 F/m ²	Base-emitter zero-bias depletion capacitance
F_C	0.5	Breakdown current
I_{KF}	Inf (A/m ²)	Corner for forward high-current roll-off

TABLE 5-1: BIPOLAR TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
I_{KR}	Inf (A/m ²)	Corner for reverse high-current roll-off
I_S	1e-15 A/m ²	Saturation current
I_{SC}	0 A/m ²	Base-collector leakage saturation current
I_{SE}	0 A/m ²	Base-emitter leakage saturation current
M_{JC}	1/3	Base-collector grading coefficient
M_{JE}	1/3	Base-emitter grading coefficient
N_C	2	Base-collector ideality factor
N_E	1.4	Base-emitter ideality factor
N_F	1	Forward ideality factor
N_R	1	Reverse ideality factor
R_B	0 Ωm^2	Base resistance
R_{BM}	0 Ωm^2	Minimum base resistance
R_C	0 Ωm^2	Collector resistance
R_E	0 Ωm^2	Emitter resistance
T_{NOM}	298.15 K	Device temperature
V_{AF}	Inf (V)	Forward Early voltage
V_{AR}	Inf (V)	Reverse Early voltage
V_{JC}	0.71 V	Base-collector built-in potential
V_{JE}	0.71 V	Base-emitter built-in potential

MOSFET Transistors

Figure 5-2 illustrates an equivalent circuit for the n-channel MOSFET transistor. The p-channel MOSFET transistor is treated similarly, but the polarities of the involved voltages are reversed.

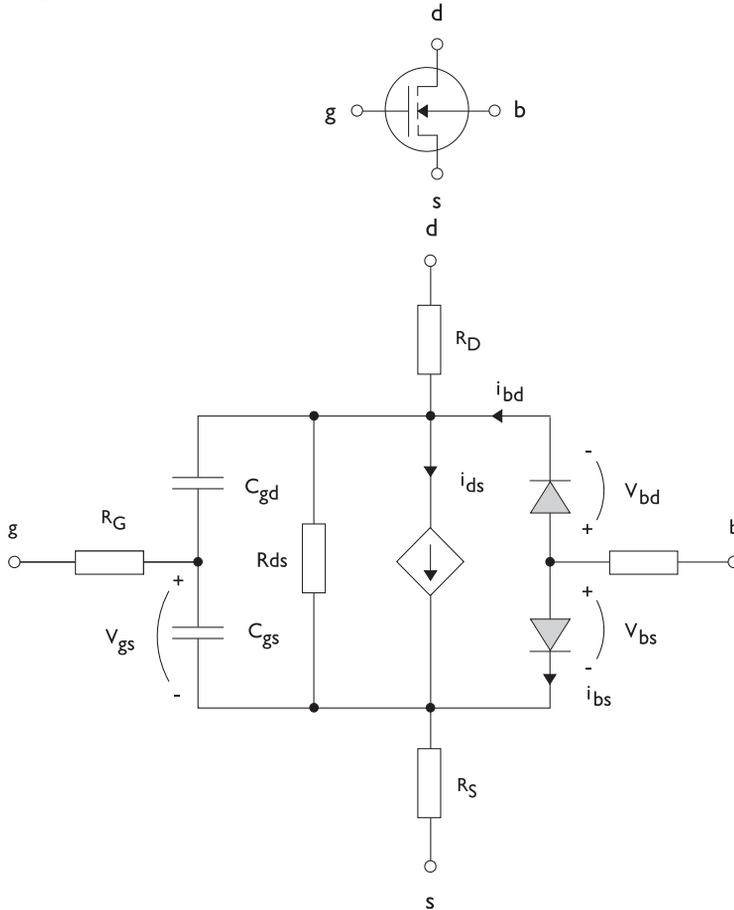


Figure 5-2: A circuit for the MOSFET transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_{ds} = \begin{cases} \frac{WK_P}{L} \frac{1 + \Lambda v_{ds}}{2} v_{ds} (2v_{th} - v_{ds}) & v_{ds} < v_{th} \\ \frac{WK_P}{L} \frac{1 + \Lambda v_{ds}}{2} v_{th}^2 & v_{ds} \geq v_{th} \\ 0 & v_{ds} < v_{th} \leq 0 \end{cases}$$

$$v_{th} = v_{gs} - (V_{TO} + \Gamma(\sqrt{\Phi} - v_{bs} - \sqrt{\Phi}))$$

$$i_{bd} = I_S \left(e^{\frac{v_{bd}}{NV_T}} - 1 \right)$$

$$i_{bs} = I_S \left(e^{\frac{v_{bs}}{NV_T}} - 1 \right)$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also several capacitances between the terminals

$$C_{gd} = C_{gd0} W$$

$$C_{gs} = C_{gs0} W$$

$$C_{jbd} = C_{BD} \times \begin{cases} \left(1 - \frac{v_{bd}}{P_B}\right)^{-M_J} & v_{bx} < F_C P_B \\ (1 - F_C)^{-1 - M_J} \left(1 - F_C(1 + M_J) + M_J \frac{v_{bx}}{P_B}\right) & v_{bx} \geq F_C P_B \end{cases}$$

The model parameters are as follows:

TABLE 5-2: MOSFET TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
C_{BD}	0 F/m	Bulk-drain zero-bias capacitance
C_{GDO}	0 F/m	Gate-drain overlap capacitance
C_{GSO}	0 F/m	Gate-source overlap capacitance
F_C	0.5	Capacitance factor
I_S	1e-13 A	Bulk junction saturation current
K_P	2e-5 A/V ²	Transconductance parameter
L	50e-6 m	Gate length
M_J	0.5	Bulk junction grading coefficient

TABLE 5-2: MOSFET TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
N	1	Bulk junction ideality factor
P_B	0.75 V	Bulk junction potential
R_B	0 Ω	Bulk resistance
R_D	0 Ω	Drain resistance
R_{DS}	Inf (Ω)	Drain-source resistance
R_G	0 Ω	Gate resistance
R_S	0 Ω	Source resistance
T_{NOM}	298.15 K	Device temperature
V_{TO}	0 V	Zero-bias threshold voltage
W	50e-6 m	Gate width
Γ (GAMMA)	1 $\text{V}^{0.5}$	Bulk threshold parameter
Φ (PHI)	0.5 V	Surface potential
Λ (LAMBDA)	0 1/V	Channel-length modulation

Diode

Figure 5-3 illustrates equivalent circuit for the diode.

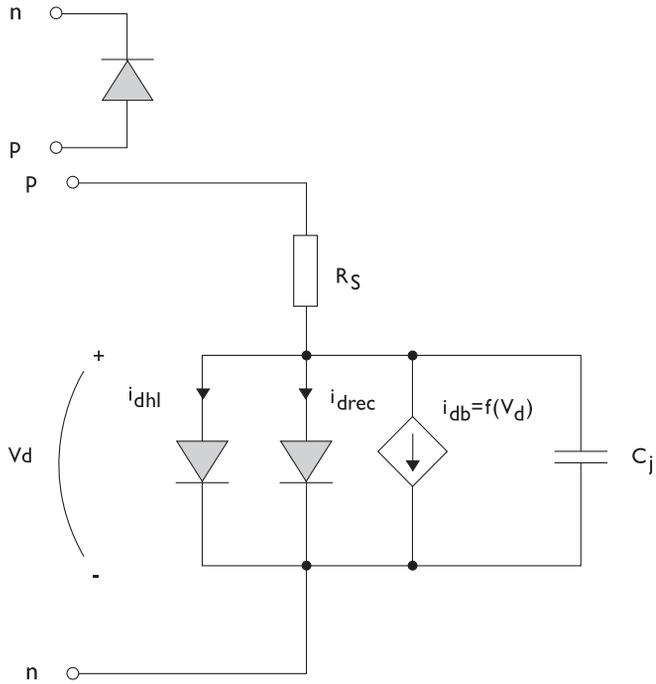


Figure 5-3: A circuit for the diode.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_d = i_{dhl} + i_{drec} + i_{db} + i_c$$

$$i_{dhl} = I_S \left(e^{\frac{v_d}{NV_T}} - 1 \right) \frac{1}{\sqrt{1 + \frac{I_S}{I_{KF}} \left(e^{\frac{v_d}{NV_T}} - 1 \right)}}$$

$$i_{drec} = I_{SR} \left(e^{\frac{v_d}{N_R V_T}} - 1 \right)$$

$$i_{db} = I_{BV} e^{\frac{v_d + B_V}{N_{BV} V_T}}$$

$$C_j = C_{J0} \times \begin{cases} \left(1 - \frac{v_d}{V_J} \right)^{-M} & v_d < F_C V_J \\ (1 - F_C)^{-1-M} \left(1 - F_C (1 + M) + M \frac{v_d}{V_J} \right) & v_d \geq F_C V_J \end{cases}$$

$$V_T = \frac{k_B T_{NOM}}{q}$$

where the following model parameters are required

TABLE 5-3: DIODE TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
B_V	Inf (V)	Reverse breakdown voltage
C_{J0}	0 F	Zero-bias junction capacitance
F_C	0.5	Forward-bias capacitance coefficient
I_{BV}	1 e-09 A	Current at breakdown voltage
I_{KF}	Inf (A)	Corner for high-current roll-off
I_S	1 e-13 A	Saturation current
M	0.5	Grading coefficient
N	1	Ideality factor
N_{BV}	1	Breakdown ideality factor
N_R	2	Recombination ideality factor
R_S	0 Ω	Series resistance
T_{NOM}	298.15 K	Device temperature
V_J	1.0 V	Junction potential

Reference for the Electrical Circuit Interface

1. P. Antognetti and G. Massobrio, *Semiconductor Device Modeling with Spice*, 2nd ed., McGraw-Hill, 1993.

Heat Transfer Interfaces

This chapter describes [The Microwave Heating Interface](#) found under the **Heat Transfer>Electromagnetic Heating** branch () when adding a physics interface.

See [The Heat Transfer Interfaces](#) and [The Joule Heating Interface](#) in the *COMSOL Multiphysics Reference Manual* for other Heat Transfer interface and feature node settings.

The Microwave Heating Interface

The **Microwave Heating** interface () is used to model electromagnetic heating for systems and devices that are on a scale ranging from 1/10 of a wavelength up to, depending on available computer memory, about 10 wavelengths. This multiphysics interface adds an Electromagnetic Waves, Frequency Domain interface and a Heat Transfer in Solids interface. The multiphysics couplings add the electromagnetic losses from the electromagnetic waves as a heat source, and the electromagnetic material properties can depend on the temperature. The modeling approach is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale.

The following table shows what study step combinations of Electromagnetic Waves, Frequency Domain interface and Heat Transfer in Solids interface are supported from the Microwave Heating interface.

TABLE 6-1: STUDY STEP COMBINATIONS OF ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN INTERFACE AND HEAT TRANSFER IN SOLIDS INTERFACE

PRESET STUDIES	ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN INTERFACE	HEAT TRANSFER IN SOLIDS INTERFACE
Frequency-Stationary	Frequency Domain	Stationary
Frequency-Transient	Frequency Domain	Transient
Frequency-Stationary, One-Way Coupled, Electromagnetic Heating	Frequency Domain	Stationary
Frequency-Transient, One-Way Coupled, Electromagnetic Heating	Frequency Domain	Transient

When a predefined **Microwave Heating** interface is added from the **Heat Transfer>Electromagnetic Heating** branch () of the **Model Wizard** or **Add Physics** windows, **Electromagnetic Waves, Frequency Domain** and **Heat Transfer in Solids** interfaces are added to the Model Builder.

In addition, a **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Electromagnetic Heating**.

On the Constituent Physics Interfaces

The Electromagnetic Waves, Frequency Domain interface computes time-harmonic electromagnetic field distributions. To use this physics interface, the maximum mesh element size should be limited to a fraction of the wavelength. Thus, the domain size that can be simulated scales with the amount of available computer memory and the

wavelength. The physics interface solves the time-harmonic wave equation for the electric field.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation. A Heat Transfer in Solids model is active by default on all domains. All functionality for including other domain types, such as a fluid domain, is also available. The temperature equation defined in solid domains corresponds to the differential form of Fourier's law that may contain additional contributions like heat sources.



In previous versions of COMSOL Multiphysics, a specific physics interface called Microwave Heating was added to the Model Builder. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see [Multiphysics Modeling Workflow](#) in the *COMSOL Multiphysics Reference Manual*.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURE

When physics interfaces are added using the predefined couplings, for example **Microwave Heating**, specific settings are included with the physics interfaces and the coupling feature.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Electromagnetic Waves, Frequency Domain** and **Heat Transfer in Solids** interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose **Electromagnetic Heating** from the available coupling features, but the modified settings are not included.



Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

TABLE 6-2: MODIFIED SETTINGS FOR A MICROWAVE HEATING INTERFACE

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Electromagnetic Waves, Frequency Domain	No changes.

TABLE 6-2: MODIFIED SETTINGS FOR A MICROWAVE HEATING INTERFACE

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Solids	No changes.
Electromagnetic Heating	<p>The Domain Selection is the same as that of the participating physics interfaces.</p> <p>The Boundary Selection is the same as the exterior and interior boundaries of the Domain Selection of the participating physics interfaces.</p> <p>The corresponding Electromagnetic Waves, Frequency Domain and Heat Transfer in Solids interfaces are preselected in the Coupled Interfaces section (described in the <i>COMSOL Multiphysics Reference Manual</i>).</p>



A side effect of adding physics interfaces one at a time is that four study types — Frequency-Stationary, Frequency-Transient, Frequency-Stationary, One-Way Coupled, Electromagnetic Heating, and Frequency-Transient, One-Way Coupled, Electromagnetic Heating — are not available for selection until *after* at least one coupling feature is added. In this case, it is better to initially not add any study at all, then add the coupling features to the **Multiphysics** node, and lastly, open the **Add Study** window and add a study sequence below the **Preset Studies for Selected Multiphysics** heading.

PHYSICS INTERFACES AND COUPLING FEATURE



Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature

The [Electromagnetic Heating](#) coupling feature node is described in this section.

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 Electromagnetic Waves, Frequency Domain Interface](#) are listed in the section [Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Frequency Domain Interface](#).
- See [The Heat Transfer Interfaces](#) in the *COMSOL Multiphysics Reference Manual* for information about the available physics features for heat transfer.



If you have an add-on module, such as the Heat Transfer Module, there are additional specialized physics nodes available and described in the individual module documentation.



Microwave Oven: Application Library path **RF_Module/Microwave_Heating/microwave_oven**

Electromagnetic Heating

The **Electromagnetic Heating** node represents the electromagnetic losses, Q_e (SI unit: W/m^3), as a heat source in the heat transfer part of the model. It is given by

$$Q_e = Q_{rh} + Q_{ml}$$

where the resistive losses are

$$Q_{rh} = \frac{1}{2} \text{Re}(\mathbf{J} \cdot \mathbf{E}^*)$$

and the magnetic losses are

$$Q_{ml} = \frac{1}{2} \text{Re}(i\omega \mathbf{B} \cdot \mathbf{H}^*)$$

In addition, it maps the electromagnetic surface losses as a heat source on the boundary (SI unit: W/m^2) in the heat transfer part of the model.

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 `emh`.

DOMAIN SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific domains to define the electromagnetic heat source or select **All domains** as needed.

When **Electromagnetic Heating** is added as an effect of adding a **Microwave Heating** interface, the selection is the same as for the participating interfaces.

Only domains that are active in the physics interfaces selected in the **Coupled Interfaces** section can be selected.

BOUNDARY SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific boundaries to define the electromagnetic boundary heat source or select **All boundaries** as needed.

When **Electromagnetic Heating** is added as an effect of adding a **Microwave Heating** interface, the selection is the same as the exterior and interior boundaries of the **Domain Selection** of the participating physics interfaces.

Only boundaries that are active in the physics interfaces selected in the **Coupled Interfaces** section can be selected.

COUPLED INTERFACES

This section defines the physics involved in the **Electromagnetic Heating** multiphysics coupling. By default, the applicable physics interface is selected in the **Electromagnetic** list to apply the **Heat transfer** to its physics interface to establish the coupling.

You can also select **None** from either list to uncouple the **Electromagnetic Heating** node from a physics interface. If the physics interface is removed from the **Model Builder**, for example **Heat Transfer in Solids** is deleted, then the **Heat transfer** list defaults to **None** as there is nothing to couple to.



If a physics interface is deleted and then added to the model again, and in order to re-establish the coupling, you need to choose the physics interface again from the **Heat transfer** or **Electromagnetic** lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present physics interface. See [Multiphysics Modeling Workflow](#) in the *COMSOL Multiphysics Reference Manual*.

Glossary

This [Glossary of Terms](#) contains finite element modeling terms in an electromagnetic waves 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

absorbing boundary A boundary that lets an electromagnetic wave propagate through the boundary without reflections.

anisotropy Variation of material properties with direction.

constitutive relation The relation between the **D** and **E** fields and between the **B** and **H** fields. These relations depend on the material properties.

cutoff frequency The lowest frequency for which a given mode can propagate through, for example, a waveguide or optical fiber.

edge element See *vector element*.

eigenmode A possible propagating mode of, for example, a waveguide or optical fiber.

electric dipole Two equal and opposite charges $+q$ and $-q$ separated a short distance d . The electric dipole moment is given by $\mathbf{p} = q\mathbf{d}$, where \mathbf{d} is a vector going from $-q$ to $+q$.

gauge transformation A variable transformation of the electric and magnetic potentials that leaves Maxwell's equations invariant.

lumped port A type of port feature. Use the lumped port to excite the model with a voltage, current, or circuit input. The lumped port must be applied between two metallic objects, separated by much less than a wavelength.

magnetic dipole A small circular loop carrying a current. The magnetic dipole moment is $\mathbf{m} = IA\mathbf{e}$, where I is the current carried by the loop, A its area, and \mathbf{e} a unit vector along the central axis of the loop.

Maxwell's equations A set of equations, written in differential or integral form, stating the relationships between the fundamental electromagnetic quantities.

Nedelec's edge element See *vector element*.

perfect electric conductor (PEC) A material with high electrical conductivity, modeled as a boundary where the electric field is zero.

perfect magnetic conductor A material with high permeability, modeled as a boundary where the magnetic field is zero.

phasor A complex function of space representing a sinusoidally varying quantity.

quasi-static approximation The electromagnetic fields are assumed to vary slowly, so that the retardation effects can be neglected. This approximation is valid when the geometry under study is considerably smaller than the wavelength.

surface current density Current density defined on the surface. The component normal to the surface is zero. The unit is A/m.

vector element A finite element often used for electromagnetic vector fields. The tangential component of the vector field at the mesh edges is used as a degree of freedom. Also called *Nedelec's edge element* or just *edge element*.

I n d e x

- 2D
 - wave equations 100
- 2D axisymmetry
 - wave equations 100
- 2D modeling techniques 28, 30
- 3D modeling techniques 30

A absorbing boundary (node) 156
AC/DC Module 13
ammeter (node) 203
anisotropic materials 76
antiperiodicity, periodic boundaries and 33
application libraries examples

- polarization plot 52
- S-parameter calculations 45

Application Libraries window 20
application library examples

- analyze as a tem field 113
- axial symmetry 29
- background field 169
- Cartesian coordinates 29
- connecting electrical circuits to physics interfaces 58
- diffraction order 120
- Drude-Lorentz polarization 152
- electrical circuits 58
- electromagnetic waves, frequency domain interface 97
- electromagnetic waves, transient 146
- far field plots 42
- far-field calculation 107
- far-field calculations 37
- far-field domain and far-field calculation 106
- hexagonal periodic port 115
- impedance boundary condition 133

- lossy eigenvalue calculations 54
- lumped element 127
- lumped port 50, 126
- microwave heating 231
- perfect electric conductor 108
- perfect magnetic conductor 109
- periodic boundary condition 138
- periodic boundary conditions 33
- periodic port reference point 123
- port 118
- port sweeps 46
- reduced order modeling 66–67
- scattered fields 36
- scattering boundary condition 130
- transmission line 154

applying electromagnetic sources 31
Archie’s law (node) 142
attenuation constant 188
axisymmetric models 29
axisymmetric waves theory

- frequency domain 175
- time domain 179

B background field (node) 168
backward-propagating wave 188
base node 209
bipolar junction transistor 209
Bloch-Floquet periodicity 138
boundary conditions

- nonlinear eigenfrequency problems and 55
- perfect electric conductor 108
- perfect magnetic conductor 109
- periodic 33
- theory 78
- using efficiently 31

boundary nodes

- electromagnetic waves, frequency domain interface 97
 - electromagnetic waves, time explicit 161
 - electromagnetic waves, transient 147
 - transmission line 154
- bulk node 210
- C**
 - calculating
 - S-parameters 45
 - capacitor (node) 203
 - Cartesian coordinates 28
 - cementation exponent 142, 186
 - circuit import, SPICE 215
 - circular port reference axis (node) 119
 - collector node 209
 - common settings 16
 - complex permittivity, electric losses and 175
 - complex propagation constant 188
 - complex relative permeability, magnetic losses and 177
 - constitutive relations 192
 - constitutive relations, theory 73
 - continuity, periodic boundaries and 33
 - coupling, to the electrical circuits interface 59
 - curl-curl formulation 192
 - current source (node) 205
 - current-controlled current source (node) 208
 - current-controlled voltage source (node) 207
 - cutoff frequency 82
 - cylindrical coordinates 29
 - cylindrical waves 129
- D**
 - Debye dispersion model 103
 - device models, electrical circuits 218
 - dielectric medium theory 79
 - diffraction order (node) 119
 - diode (node) 212
 - diode transistor model 224
 - dispersive materials 76
 - divergence constraint (node) 104
 - documentation 19
 - domain nodes
 - electromagnetic waves, frequency domain interface 97
 - electromagnetic waves, time explicit 161
 - drain node 210
 - Drude-Lorentz dispersion model 102
 - Drude-Lorentz polarization (node) 151
- E**
 - E (PMC) symmetry 37
 - edge current (node) 140
 - effective medium (node) 143
 - eigenfrequency analysis 54
 - eigenfrequency calculations theory 180
 - eigenfrequency study 172
 - eigenmode analysis 81
 - eigenvalue (node) 56
 - electric current density (node) 165
 - electric field (node) 127, 165
 - electric losses theory 175
 - electric point dipole (node) 141
 - electric potential (node) 118
 - electric scalar potential 75
 - electric susceptibility 193
 - electrical circuit interface 200
 - theory 217
 - electrical circuits
 - modeling techniques 58
 - electrical conductivity 73
 - electrical conductivity, porous media 186
 - electrical size, modeling 13
 - electromagnetic energy theory 75
 - electromagnetic heating (node) 231

- electromagnetic quantities 86
- electromagnetic sources, applying 31
- electromagnetic waves, frequency domain interface 90, 92
 - theory 171
- electromagnetic waves, time explicit interface 160
 - theory 191
- electromagnetic waves, transient interface 145
 - theory 171
- emailing COMSOL 21
- emitter node 209
- error message, electrical circuits 59
- exponential filter, for wave problems 163
- exporting
 - SPICE netlists 63
- external current density (node) 105
- external I vs. U (node) 212
- external I-terminal (node) 214
- external U vs. I (node) 213
- F** far field variables 39
 - Faraday's law 191
 - far-field calculation (node) 106, 169
 - far-field calculations 84
 - far-field domain (node) 105, 107, 169
 - far-field variables 37
 - file, Touchstone 97, 154
 - Floquet periodicity 33, 138
 - fluid saturation 143
 - flux/source (node) 168
 - formation factor 186
 - forward-propagating wave 188
 - free-space variables 99
 - frequency domain equation 172
 - Frequency-Domain Modal Method 66
- G** gate node 210
 - Gauss' law 191
- geometry, simplifying 28
- ground (node) 118
- ground node (node) 202
- H** H (PEC) symmetry 37
 - high frequency modeling 13
 - hybrid-mode waves
 - axisymmetric, frequency domain 175
 - axisymmetric, time domain 179
 - in-plane, frequency domain 174
 - in-plane, time domain 178
 - perpendicular 173
- I** impedance boundary condition (node) 132
 - importing
 - SPICE netlists 62, 215
 - incoming wave (node) 156
 - inductor (node) 204
 - inhomogeneous materials 76
 - initial values (node)
 - electromagnetic waves, frequency domain interface 105
 - electromagnetic waves, time explicit interface 164
 - electromagnetic waves, transient 151
 - transmission line 156
 - in-plane TE waves theory
 - frequency domain 174
 - time domain 178
 - in-plane TM waves theory
 - frequency domain 174
 - time domain 178
 - inports 111
 - integration line for current (node) 118
 - integration line for voltage (node) 118
 - internet resources 18
- K** Kirchhoff's circuit laws 217
 - knowledge base, COMSOL 21

- L**
 - line current (out-of-plane) (node) 142
 - linearization point 56
 - listener ports 111
 - losses, electric 175
 - losses, magnetic 177
 - lossy eigenvalue calculations 54
 - lumped element (node) 126
 - lumped port (node) 123, 158
 - lumped ports 47–48
- M**
 - magnetic current (node) 139
 - magnetic current density (node) 165
 - magnetic field (node) 128, 166
 - magnetic losses theory 177
 - magnetic point dipole (node) 141
 - magnetic susceptibility 74, 193
 - material properties 76
 - materials 77
 - Maxwell's equations 72
 - electrical circuits and 217
 - Maxwell-Ampere's law 191
 - mesh resolution 32
 - microwave heating interface 228
 - mode analysis 56, 173
 - mode phase
 - for Port and Diffraction Order 113
 - modeling tips 27
 - MPH-files 20
 - multiphysics couplings
 - microwave heating 228
 - mutual inductance (node) 211
- N**
 - n-Channel MOS transistor 210, 221
 - n-Channel MOSFET (node) 210
 - netlists, SPICE 62, 215
 - nodes, common settings 16
 - nonlinear materials 76
 - NPN bipolar junction transistor 218
 - NPN BJT (node) 209
 - numeric modes 110
- O**
 - open circuit (node) 157
 - orthogonal polarization (node) 121
- P**
 - p-Channel MOS transistor 210
 - p-Channel MOSFET (node) 210
 - PEC. see perfect electric conductor
 - perfect conductors theory 78
 - perfect electric conductor (node) 166
 - boundaries 107
 - perfect magnetic conductor (node) 108, 166
 - periodic boundary conditions 33
 - periodic condition (node) 137
 - periodic port reference point (node) 122
 - permeability
 - anisotropic 173
 - permeability of vacuum 73
 - permittivity
 - anisotropic 173
 - permittivity of vacuum 73
 - phasors theory 79
 - physics interfaces, common settings 16
 - PMC. see perfect magnetic conductor
 - PNP BJT (node) 209
 - polarization, 2D and 2D axisymmetry 30
 - port (node) 109
 - port boundary conditions 45
 - ports, lumped 47–48
 - potentials theory 75
 - power law, porous media
 - conductivity 183
 - permeability 185
 - permittivity 184
 - Poynting's theorem 75
 - predefined couplings, electrical circuits 59
 - propagating waves 188
 - propagation constant 188
- Q**
 - quality factor (Q-factor) 54, 172

- quasi-static modeling 13
- R**
 - reciprocal permeability, volume average 185
 - reciprocal permittivity, volume average 184
 - reference point (node) 131
 - refractive index 100
 - refractive index theory 176
 - relative electric field 36
 - relative permeability 74
 - relative permittivity 74
 - remanent displacement 74
 - resistor (node) 203
- S**
 - saturation coefficient 186
 - saturation exponent 142
 - scattered fields, definition 35
 - scattering boundary condition (node) 128, 167
 - scattering parameters. *see* S-parameters
 - selecting
 - mesh resolution 32
 - solver sequences 32
 - study types 13, 16
 - semiconductor device models 218
 - short circuit (node) 158
 - SI units 86
 - simplifying geometries 28
 - skin effect, meshes and 32
 - solver sequences, selecting 32
 - source node 210
 - space dimensions 14, 28
 - S-parameter calculations
 - electric field, and 44
 - port node and 109
 - theory 81
 - spherical waves 129
 - SPICE
 - exporting 63
 - SPICE netlists 62, 215
 - standard settings 16
 - study types 13
 - boundary mode analysis 110
 - eigenfrequency 54, 172
 - frequency domain 172
 - mode analysis 56, 173
 - subcircuit definition (node) 208
 - subcircuit instance (node) 209
 - surface charge density 78
 - surface current density (node) 133, 167
 - surface magnetic current density (node) 134
 - surface roughness (node) 135
 - symbols for electromagnetic quantities 86
 - symmetry in E (PMC) or H (PEC) 37
 - symmetry planes, far-field calculations 37
 - symmetry, axial 29
 - T**
 - TE axisymmetric waves theory
 - frequency domain 175
 - time domain 179
 - TE waves theory 83
 - technical support, COMSOL 21
 - TEM waves theory 83
 - terminating impedance (node) 157
 - theory
 - constitutive relations 73
 - dielectrics and perfect conductors 78
 - electric and magnetic potentials 75
 - electrical circuit interface 217
 - electromagnetic energy 75
 - electromagnetic waves 171
 - electromagnetic waves, time explicit interface 191
 - far-field calculations 84
 - lumped ports 48
 - Maxwell equations 72

- phasors 79
- S-parameters 81
- surface charges 78
- transmission line 187
- time domain equation, theory 177
- TM waves
 - axisymmetric 172
- TM waves theory 83
- Touchstone file 97, 154
- transformer (node) 211
- transition boundary condition (node)
 - 136
- transmission line equation (node) 155
- transmission line interface 153
 - theory 187
- TW axisymmetric waves theory
 - frequency domain 175
 - time domain 179
- two-port network (node) 139
- two-port network port (node) 140

U uniform element (node) 127
units, SI 86

V variables

- eigenfrequency analysis and 55
- far-field 37
- for far fields 39
- Jones vectors 52
- lumped ports 50
- mode analysis 57, 173
- S-parameters 45

vector elements theory 179
voltage input, ports 47
voltage source (node) 204
voltage-controlled current source
(node) 207
voltage-controlled voltage source (node)
206
voltmeter (node) 202

- volume average, permeability 185
- volume average, permittivity 184
- volume averages, porous media 183

W wave equation, electric (node) 99, 148
wave equations (node) 162
wave excitation 111
wave impedance theory 83
wave number, free-space 99
wavelength, meshes and 32
websites, COMSOL 21