COMSOL MULTIPHYSICS

RF MODULE

USER'S GUIDE

VERSION 4.0a

COMSOL

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

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

Finland
COMSOL OY
Arabiankatu 12
FIN-00560 Helsinki
Phone: +358 9 2510 400
Fax: +358 9 2510 4010
info@comsol.fi
www.comsol.fi

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

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

India
COMSOL Multiphysics Pvt. Ltd.
Esquire Centre
C-Block, 3rd Floor
No. 9, M. G. Road
Bangalore 560001
Karnataka
Phone: +91-80-4092-3859
Fax: +91-80-4092-3856
info@comsol.co.in

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

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

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

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

United Kingdom
COMSOL Ltd.
UH Innovation Centre
College Lane
Hatfield
Hertfordshire AL10 9AB
Phone: +44-(0)-1707 636020
Fax: +44-(0)-1707 284746
info.uk@comsol.com
www.uk.comsol.com

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

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

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

info@comsol.com
www.comsol.com

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

Company home page
www.comsol.com

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

COMSOL RF Module User's Guide

Version: June 2010 COMSOL 4.0a

Part number: CM021001

CONTENTS

Chapter 1: Introduction to the RF Module

Chapter 2: RF Modeling

Chapter 3: Review of Electromagnetics

Chapter 4: The Radio Frequency Interfaces

Chapter 5: The Electrical Circuit Interface

Chapter 6: The Heat Transfer Interface

Chapter 7: Glossary

1

Introduction to the RF Module

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

This section introduces you to the capabilities of the RF Module including a summary of the physics interfaces and where you can find documentation and model examples. Click on the links for more information.

Also see RF Modeling for an extensive oveview of the modeling stages and some realistic and illustrative models.

About the RF Module

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 physics interfaces for electromagnetic waves included here are fully multiphysics enabled, making it possible to couple them to any other physics interface in COMSOL Multiphysics or the other modules.

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 contains a set of physics interfaces adapted to a broad category of electromagnetic simulations. Those who are not familiar with computational techniques but have a solid background in electromagnetics should find this module extremely beneficial. It can serve equally well as an excellent tool for educational purposes.

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

What Problems Can You Solve?

The RF Module is a collection of physics interfaces for COMSOL Multiphysics that handles time-harmonic, time-dependent, and eigenfrequency/eigenmode problems. The physics interfaces fall into two main categories:

- Frequency domain electromagnetic waves (see The Electromagnetic Waves Interface)
  - Time-harmonic
  - Eigenfrequency/eigenmode

- Transient electromagnetic waves (see The Transient Electromagnetic Waves Interface)
  - Time-dependent

In addition there is a predefined multiphysics interface for **Microwave Heating** (see The Microwave Heating Interface) that involves frequency domain (time-harmonic) electromagnetic waves in conjunction with stationary or transient heat transfer. This interface is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale (adiabatic assumption).

All categories are available in both 2D and 3D. In 2D the package offers in-plane formulations for problems with a planar symmetry as well as axisymmetric formulations for problems with a cylindrical symmetry. It further supports 2D mode analysis of waveguide cross sections with out-of-plane propagation.

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 RF 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.

Examples of applications you can successfully simulate with the RF Module include waveguides, photonic crystals, antennas, and transmission lines.

Note: If you are working directly from a PDF on your computer, the blue underlined links do not work to open a model or documentation referenced in a different user guide. However, if you are using the online help desk in COMSOL Multiphysics, these links work to other modules, model examples, and documentation sets.

The Documentation

The *COMSOL Multiphysics User's Guide* and *COMSOL Multiphysics Reference Guide* describe all the interfaces included with the basic COMSOL license. These guides also have instructions about how to use COMSOL Multiphysics, and how to access the documentation electronically through the COMSOL Multiphysics help desk.

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree. This documentation is specific to the **RF** interfaces.

The Model Library

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

To open any model in COMSOL, select **File>Open Model Library** from the main menu, and then search either by name or browse by module name. If you also want to review the documentation explaining how to build a model, select **Help>Documentation** in COMSOL and again, search by name or browse by module.

If you have feedback or suggestions for additional models for the library (including those developed by you), feel free to contact us at suggest@comsol.com.

Typographical Conventions

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

- A **boldface** font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL Desktop (or, for toolbar buttons, in the corresponding tooltip). For instance, we often refer to the **Model Builder** window, which is the window that contains the model tree. As another example, the instructions might say to click the **Zoom Extents** button, and the boldface font indicates that you can expect to see a button with that exact label on the COMSOL Desktop.
- Click text highlighted in blue and underlined to go to other information in the PDF. When you are using the online help desk in COMSOL Multiphysics, these links also work to other modules, model examples, and documentation sets.
- The names of other items on the COMSOL Desktop that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Main toolbar; this horizontal bar containing many icons appears on top of the user interface. However, nowhere on the screen will you see the term "Main" referring to this toolbar.
- The symbol **>** indicates a menu item. For example, **Options>Results** is equivalent to: From the **Options** menu, choose **Results**.
- A `Code` (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as "Type `1.25` in the **Current density** edit field." The monospace font also indicates code. This font also indicates variable names. An italic *`Code`* (monospace) font indicates user inputs and parts of names that can vary or be defined by the user.
- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

Overview of the Physics Interfaces

The physics interfaces in the RF Module form a complete set of simulation tools for electromagnetic wave simulations. To select the right physics interface for describing the real-life physics you need to consider the material properties and the time variations of the fields.

You select the physics interface and study type from the **Model Wizard** when starting a new model. You can also add interfaces and add or change studies to an existing model to create a multiphysics model.

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

The RF Module Interfaces

The physics interfaces in the RF Module are listed below followed by a table specifying which space dimension each one is available in.

ELECTROMAGNETIC WAVES

For wave problems, there are two physics interfaces:

- The Electromagnetic Waves Interface

  This interface supports the study types frequency domain (time-harmonic), eigenfrequency, mode (2D only), boundary mode analysis and frequency domain modal.

- The Transient Electromagnetic Waves Interface

  This interface supports the time dependent study type, only.

MICROWAVE HEATING

The Microwave Heating Interface predefined multiphysics interface combines the features of an **Electromagnetic Waves** interface from the RF Module with those of the **Heat Transfer** interface. The predefined interaction adds the electromagnetic losses from the electromagnetic waves as a heat source.

PHYSICS INTERFACE	ICON	TAG	1D	1D AXI	2D	2D AXI	3D
AC/DC							
Electrical Circuit		cir	not space dependent				
Heat Transfer							
Microwave Heating		mh			√	√	√
Radio Frequency							
Electromagnetic Waves		emw			√	√	√
Transient Electromagnetic Waves		temw			√	√	√

Selecting the Study Type—Time-Dependent or Frequency Domain

When you have selected the physics interface you can also choose a study type. However, you can also change this later in the COMSOL Multiphysics user interface. The available study types for electromagnetic waves are frequency domain, eigenfrequency, mode, boundary mode analysis and frequency domain modal. The only available study type for transient electromagnetic waves is time-dependent (transient). Below you find an explanation of the difference between the time dependent study type offered by the transient electromagnetic waves interface and the frequency domain study type offered by the electromagnetic waves interface. This is followed by a more general short introduction to the two physics interfaces in the RF Module.

Table 1-1 lists the available physics interfaces in the RF Module. For a descriptive illustration and more details on each of these interfaces, see the corresponding section in the table's **Page** column.

In the **Name** column you find the default name that is given to the physics interface. This name appears as a label on the physics interface when you use it and is of special importance when performing multiphysics simulations in order to distinguish between different physics interfaces in the model. The variables defined by the physics interfaces get an underscore plus the physics interface name appended to their names.

The **Dependent Variables** column contains the variables that the PDEs are formulated for. For 2D modeling, the PDEs solved in the simulations are formulated for an appropriate subset of variables or field components (selected by the user).

The **Field Components** columns list the nonzero field components. In 3D and planar 2D, Cartesian components are used; in 2D axisymmetry, cylindrical components are used.

Finally, the **Study Type** columns indicate the study types that the physics interface supports.

TABLE 1-1: RF MODULE PHYSICS INTERFACES

PHYSICS INTERFACE	NAME	DEP. VAR.	FIELD COMPONENTS		STUDY TYPES				
			MAGNETIC FIELD	ELECTRIC FIELD	TIME DOMAIN	FREQUENCY DOMAIN	EIGENFREQUENCY	MODE ANALYSIS	BOUNDARY MODE ANALYSIS
ELECTROMAGNETIC WAVES									
Electromagnetic Waves	emw	**E**	all 3	all 3		√	√	√	√
Transient Electromagnetic Waves	temw	**A**	all 3	all 3	√				
MULTIPHYSICS									
Microwave Heating	mh	T, J, E	all 3	all 3	T, J	E			E

To carry out different kinds of simulations for a given set of parameters in a physics interface, you only have to change the *solver type* or specify the *study type*. The available study types are *Time domain*, *Frequency domain*, *Eigenfrequency*, *Mode analysis*, and *Boundary mode analysis*. Not all study types are available in all physics interfaces.

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 type. However, this approach can be time consuming if small time steps are necessary for the desired accuracy. It is necessary when your inputs are transients like turn-on and turn-off sequences.

An efficient simplification is to 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.

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

Worth mentioning here are some special predefined study types for the multiphysics interface for **Microwave Heating**. This interface is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale (adiabatic assumption). Thus, there are two optional study types:

- **Frequency-Stationary**
  - Time-harmonic electromagnetic waves
  - Stationary heat transfer
- **Frequency-Transient**
  - Time-harmonic electromagnetic waves
  - Transient heat transfer

Material Properties

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; for details on how to enter anisotropic material properties, see Modeling Anisotropic Materials in the *COMSOL Multiphysics User's Guide* (or see Where Do I Access the Documentation and Model Library?).

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.

MATERIAL BROWSER

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

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

See Materials in the *COMSOL Multiphysics User's Guide* for details about the material libraries.

2

RF Modeling

The goal of this section is to familiarize you with the modeling procedure in the RF Module. Because this module is totally integrated with COMSOL Multiphysics, the modeling process is similar. This section also shows a number of models illustrating different aspects of the simulation process. It steps you through all the stages of modeling, from geometry creation to postprocessing.

The Modeling Process

The modeling process in the RF Module consists of the following main steps, which, except the first step, correspond to branches in the Model Tree:

1. Selecting the appropriate physics interface or predefined multiphysics coupling in the *Model Wizard*.
2. Defining model parameters and variables, if needed, in the *Definitions* branch
3. Drawing or importing the model 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.

The steps listed above are all available in the COMSOL Desktop environment. Once you have defined a model, you can go back and make changes in input data, equations, boundary conditions, and mesh. You can also restart the solver, for example, using the existing solution as the initial condition or initial guess. You can even alter the geometry—the equations and boundary conditions are still available through associative geometry. By adding another physics interface, you can account for a phenomenon not previously described in a model. To do this, simply right-click the **Model** node in the **Model Builder**. This action still retains the existing geometry, equations, boundary conditions, and current solution, which you can build upon for further development of the model.

Preparing for Modeling

This section is intended to guide you through the selection process among the physics interfaces in the RF Module. Several topics in the art of modeling are covered here that you may not find in ordinary textbooks on electromagnetic theory. You get help in answering questions like:

- 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?

The intention of this section is not to give detailed descriptions about each physics interface but to give references to the information elsewhere in this manual. First you get a few general tips about modeling, helping you to decide what to include in your simulation. The next topic is related to the geometry, what you can do to minimize the size of your problem, and which spatial dimension (2D or 3D) that suits your model. This section also includes some tips about boundary conditions and how you can use these to minimize the geometry. Then the issues regarding the numerical part of your model are discussed (that is, meshing and solving). The final topics cover more specific issues about the physics interfaces, the study types, and how to treat the fields and sources.

GENERAL TIPS

Before you start modeling, try first to answer the following questions:

- What is the purpose of the model?
- What information do you want to extract from the model?

It is important to remember that a model never captures all the details of reality. 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.

Simplifying Geometries

Most of the problems that you solve 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 your 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 you have verified your 2D model, you are in a much better position to build a 3D model.

2D MODELS

The text below guides you through 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.

Cartesian Coordinates

In this case you view a cross section 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. 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 you know there is no variation of the solution in one particular dimension. This is shown in the model H-Bend Waveguide 2D in the RF and Microwave Engineering section of the RF Module Model Library (or see Where Do I Access the Documentation and Model Library?), where the electric field only has one component in the *z* direction and is constant along that axis. The second approach

is when you have a problem where you can neglect the influence of the finite extension in the third dimension.

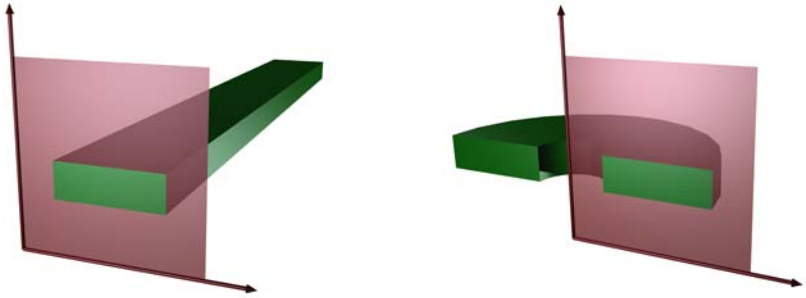


Figure 2-1: The cross sections and their real geometry for Cartesian coordinates and cylindrical coordinates (axial symmetry).

Axial Symmetry (Cylindrical Coordinates)

If you can construct the 3D geometry by revolving a cross section around an axis, and if no variations in any variable occur when going around the axis of revolution, you can 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 you have to multiply all flows with αr, where α is the revolution angle (for example, 2π for a full turn). See the model Conical Antenna in the RF and Microwave Engineering section of the RF Module Model Library for an example (or see Where Do I Access the Documentation and Model Library?).

Polarization in 2D

In addition to selecting 2D or 2D axisymmetry when you start building the model, the main physics interface node (Electromagnetic Waves or Transient Electromagnetic Waves) in the model tree offers a choice in the **Settings** section of **Electric field components to solve for**. The available choices are **Out-of-plane vector**, **In-plane vector** and **Three-component vector**. This choice determines what polarizations you can handle. 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 easily get large and

require more computer power, memory, and time to solve. The extra time you spend on simplifying your model is probably well spent when solving it. Below are a few issues that you need to address before starting to implement a 3D model in the RF 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 you can use a much denser mesh.
- 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. You can then remove the geometry below the plane if you do not see any differences 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, so you can replace it by an analytical function. You can use this approach either to convert 3D to 2D or to convert a layer to a boundary condition (see also the section Boundary Conditions below).

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 may have regions where the solution only undergoes small changes. This problem is addressed in two related steps. First, you need to truncate the geometry in a suitable position. Second, you need to apply a suitable boundary condition 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, you must apply special low-reflecting boundary conditions. This boundary should be in the order of a few wavelengths away from any source, see the model Conical Antenna in the RF and Microwave Engineering section of the RF Module Model Library. A more accurate option is to use Perfectly Matched Layers, which is a layer that absorbs all radiated waves with small reflections, see the RF Module Model Library “Radar Cross Section” (see Where Do I Access the Documentation and Model Library?).

- Replace thin layers with boundary conditions where possible. There are several types of boundary conditions in COMSOL Multiphysics suitable for such replacements. You can, 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.

SOURCES

You can apply electromagnetic sources 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 you can compute 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, 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. You need at least two linear elements per skin depth to capture the variation of the fields. If you do not study the skin depth or need a very accurate measure of the dissipation loss profile, you can replace regions with a small skin depth with a boundary condition, thereby saving elements. If you find it necessary to resolve the skin depth, the boundary layer meshing technique can be a convenient way to get a dense mesh near

a boundary. See the *COMSOL Multiphysics User's Guide*, Creating Boundary Layer Meshes (or see Where Do I Access the Documentation and Model Library?).

- 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

You can, in most cases, use the solver sequence that COMSOL Multiphysics generates for you. The choice of solver is optimized for the typical case for each physics interface and study type in the RF Module. However, in special cases you might need to tune the solver settings. This is especially important for 3D problems because they can require a large amount of memory. For large 3D problems, you may need a 64-bit platform. You can find a more detailed description on the solver settings in the section Solving in the *COMSOL Multiphysics User's Guide* (or see Where Do I Access the Documentation and Model Library?).

Periodic Boundary Conditions

The RF Module has a dedicated **Periodic Condition** feature. 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. The model "Periodic Boundary Condition" in the Tutorial Models section of the RF Module Model Library shows how to use the Copy Mesh feature to ensure that the mesh on the destination boundary is identical to that on the source boundary.

If more advanced periodic boundary conditions are required, for example, when there is a known rotation of the polarization from one boundary to another, the section Model Couplings in the *COMSOL Multiphysics User's Guide* presents tools for defining more general mappings between boundaries. Also see Where Do I Access the Documentation and Model Library?.

Perfectly Matched Layers (PMLs)

Introduction

One of the challenges in finite element modeling is how to treat open boundaries in radiation problems. The RF Module offers two closely related types of absorbing boundary conditions, the *scattering boundary condition* and the *port boundary condition*. The former is a first order absorbing boundary condition for a plane wave or (optionally) a cylindrical or spherical wave, whereas the latter is a perfectly absorbing condition for general modes of a known shape, provided that the correct mode shape and the propagation constant are supplied. Several port boundary condition features representing an expansion into mutually orthogonal modes is also allowed and can be used to account for higher diffraction orders from a grating or be used to truncate a waveguide operated in the multimode regime. However, in many scattering and antenna-modeling problems, you cannot describe the outgoing radiation as a plane wave with a well-known direction of propagation or as a known, finite modal expansion. In such situations, consider using *perfectly matched layers* (PMLs). A PML is strictly speaking not a boundary condition but an additional domain that absorbs the incident radiation without producing reflections. It provides good performance for a wide range of incidence angles and is not particularly sensitive to the shape of the wave fronts. The PML formulation can be deduced from Maxwell's equations by introducing a complex-valued coordinate transformation under the additional requirement that the wave impedance should remain unaffected, see Ref. 1. The following section describes how to use the semiautomatic frequency domain PMLs in the RF Module to create planar, cylindrical, and spherical PMLs. Transient PMLs are not supported by the RF Module.

PML Implementation

This RF Module uses the following coordinate transform for the general coordinate variable t.

$$t' = \left(\frac{t}{\Delta_w}\right)^n (1-i)\lambda F \tag{2-1}$$

The coordinate, t, and the width of the PML region, Δ_w, are geometrical parameters that are automatically extracted for each region. The other parameters are the PML scaling factor F and the PML order n that can be modified in the PML feature (both

default to unity). To avoid a non-linear dependence in the eigenvalue, the wavelength, λ, is removed from the scaling expression when computing an eigenfrequency study. The software automatically computes the value for Λ_w and the orientation of the transform for PML regions that are Cartesian, cylindrical, or spherical. However, there is no check that the geometry of the region is correct, so it is important to draw a proper geometry and select the corresponding region type. Typical examples of PML regions that work nicely are shown in the following figures for the three first PML types below. The supported PML types are:

- **Cartesian**—PMLs absorbing in Cartesian coordinate directions. It is available in 2D and 3D.
- **Cylindrical**—PMLs absorbing in cylindrical coordinate directions from a specified axis. It is available in 2D and 2D axisymmetry. In axisymmetry, the cylinder axis is the z-axis.
- **Spherical**—PMLs absorbing in the radial direction from a specified center point. It is available in 2D axisymmetry and 3D.
- **General**—General PMLs or domain scaling with user-defined coordinate transformations.

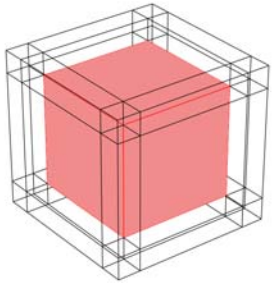


Figure 2-2: A cube surrounded by typical PML regions of the type "Cartesian."

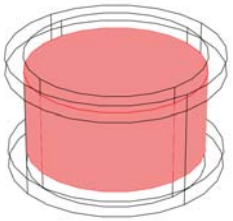


Figure 2-3: A cylinder surrounded by typical cylindrical PML regions. Cylindrical PMLs are only supported in 2D axisymmetry.

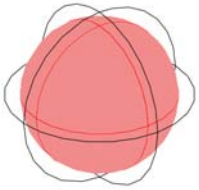


Figure 2-4: A sphere surrounded by a typical spherical PML region.

GENERAL SCALING

With manual control of the scaling, the geometrical parameters that defines the scaling are added as sub-nodes, labeled as **Manual scaling**. These sub-nodes has no effect unless the type of the PML node is set to **General**. Each sub-node has three parameters, **Scaling direction**, **Geometrical width**, and **Coordinate at interface**. The first parameter sets the direction from the interface to the outer boundary, the second parameter sets the width of the region, and the last parameter sets an arbitrary coordinate at the interface. When going from any of the other types to the **General** type, sub-nodes are automatically added that represent scaling of the previous type.

Known Issues When Modeling Using PMLs

When modeling with PMLs you should be aware of the following:

- The coordinate scaling resulting from PMLs also yields an equivalent scaling of the mesh that may effectively result in a poor element quality. (The element quality displayed by the mesh statistics feature does not account for this effect.) This typically happens when the geometrical thickness of the PML deviates much from

one wavelength (local wavelength rather than free space wavelength). The poor element quality causes poor convergence for iterative solvers and make the problem ill-conditioned in general. Especially vector element formulations (the ones using two or more components of a vector field variable) are sensitive to low element quality. For this reason, it is strongly recommended to use swept meshing in the PML domains. The sweep direction should be selected the same as the direction of scaling. For Cartesian PMLs and regions with more than one direction of scaling it is recommended to first sweep the mesh in the domains with only one direction of scaling, then sweep the domains with scaling in two directions, and finish by sweeping the mesh in the domains with PML scaling in all three directions.

- The expressions resulting from the stretching get quite complicated for spherical PMLs in 3D. This increases the time for the assembly stage in the solution process. After the assembly, the computation time and memory consumption are comparable to a problem without PMLs. The number of iterations for iterative solvers might increase if the PML regions have a coarse mesh.
- PML regions deviating significantly from the typical configurations shown in the beginning of this section can cause the automatic calculation of the PML parameter to give erroneous result. Enter the parameter values manually if you find that this is the case.
- The PML region is designed to model uniform regions extended toward infinity. Avoid using objects with different material parameters or boundary conditions that influence the solution inside an PML region.

You can also use parts of the shapes shown, but the PML scaling does probably not work for complex shapes that deviate significantly from these shapes.

REFERENCE

1. Jianming Jin, *The Finite Element Method in Electromagnetics*, 2nd Edition, Wiley-IEEE Press, May 2002.

Example Models

For examples on using perfectly matched layers, use any of the following models in the the RF Module Model Library (or see Where Do I Access the Documentation and Model Library?):

- Tutorial Models/Radar Cross Section (2D, cylindrical PML), and RF Coil (3D, spherical PML with swept mesh).
- RF and Microwave Engineering/Balanced Patch Antenna for 6 GHz (3D, spherical PML).

Scattered-Field Formulation

Introduction

For many problems, it is the scattered field that is the interesting quantity. Such models usually have a known incident field that you really do not need to compute the solution for, so there are several benefits to reduce the formulation and only solve for the scattered field. If the incident field is much larger in magnitude than the scattered field, the accuracy of the simulation improves if you solve for the scattered field. Furthermore, a plane wave excitation is easier to set up, because for scattered-field problems you specify it as a global plane wave. Otherwise you have to set up matched boundary conditions around your structure, which can be rather complicated for nonplanar boundaries. Especially when using perfectly matched layers (PMLs), the advantage of using the scattered-field formulation becomes clear. With a full-wave formulation, you have to take the damping in the PML into account when exciting the plane wave, because the excitation appears outside the PML. With the scattered-field formulation you specify the plane wave for all non-PML regions, so it is not at all affected by the PML design.

Physics Interfaces with Scattered Fields

The scattered-field formulation is available for the Electromagnetic Waves interfaces. It is accessible via the **Solve for** option in the **Settings** section of the main physics interface node in the model tree. The available options are **Full field** an **Scattered field**. You need to specify the incident field in the **Background electric field** input field that becomes available when choosing **Scattered field**.

The scattered field is in postprocessing called **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.

Example Model Scattered Field Formulation

For an example on using the scattered-field formulation, see the model Radar Cross Section in the Tutorial Models section in the RF Module Model Library (or see Where Do I Access the Documentation and Model Library?).

Far-Field Postprocessing

The far electromagnetic field from, for example, antennas can be calculated from the calculated near field on a boundary using far-field postprocessing. 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.

For the theory behind the far-field postprocessing, see Far-Field Calculations.

Far-Field Support in the Electromagnetic Waves Interface

The Electromagnetic Waves interface supports far-field postprocessing. To define the far-field variables use the **Far Field Calculation** feature. Select the boundaries where the algorithm then integrates the near field, and enter a name for the far electric field. You can also specify if you want to use symmetry planes in your model when calculating the far-field variable. The symmetry planes have to coincide with one of the Cartesian coordinate planes. For each of these planes it is possible to select the type of symmetry you use, which can be of either **Symmetry in E (PMC)** or **Symmetry in H (PEC)**. Make the choice here match the boundary condition you used for the symmetry boundary. Using these settings, you can include the parts of the geometry that are not in the model for symmetry reasons in the far-field analysis.

For each variable name that you enter, 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 that you enter in the field. For example, if you enter the name `Efar` 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 may vary but typically look something like `emw.Efarz` etc.

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. Note that 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 you want to evaluate the far electric field. The expression

```
Efarx(dx,dy,dz)
```

gives the value of the far electric field in this direction. If you instead want to give the direction as an angle, use the expression

```
Efarx(sin(theta)*cos(phi),sin(theta)*sin(phi),cos(theta))
```

where you define the variables `theta` and `phi` 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 `normEfardB`, respectively.

Example Model Far-Field

For an example on far-field postprocessing, see the model Radar Cross Section, in the Tutorial Models section in the RF Module Model Library (or see Where Do I Access the Documentation and Model Library?).

S-Parameters and Ports

S-Parameters in Terms of Electric Field

Scattering parameters or S-parameters are complex-valued, frequency dependent matrices describing the transmission and reflection of electromagnetic energy measured 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, that is, there is no reflection directly at a port.

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

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

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 in COMSOL Multiphysics: Ports

The Electromagnetic Waves physics interfaces have built-in support for S-parameter calculations. To set up an S-parameter study use a **Port** boundary feature for each port in the model. You can also use lumped ports that approximate connecting transmission lines. The lumped ports should only be used when the port width is much smaller than the wavelength for more details on lumped ports, see Lumped Ports with Voltage Input. Below, the more general (wave type) of port excitation is treated.

THE PORT PROPERTIES SECTION

In the **Port Properties** section of the port feature you specify 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, see the note below. You also set the **Type of Port** which could be **User defined**, **Numeric**, or (analytical modes) **Rectangular** (TE or TM) or **Coaxial** (only TEM). The **Wave excitation at this port** setting decides whether it is an inport or a listener port when **On** the **Port input power** and **Port phase** can also be specified.

Note: 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 if you excite several ports, the S-parameter output is turned off.

The **Port Sweep Settings** section in the Electromagnetic Waves interface node 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.

Coaxial Modes

Analytical coaxial modes are available in 3D (see note below) and for axisymmetric TM waves. Only the fundamental (TEM) mode is available as a predefined mode.

Note: For analytical coaxial ports to work well in 3D the connecting boundaries should be assigned the impedance boundary condition rather than the default PEC condition. Alternatively use the lumped coaxial port as it works with connecting PEC boundaries in 3D.

Numeric Modes

The numeric option works only if the Boundary Mode Analysis study type has been selected. It should appear before the frequency domain study node in the study branch of the model tree. If you need to use more than one numeric port, you must 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.

THE PORT MODE SETTINGS SECTION

The **Port Mode Settings** section has controls for specifying the details of a rectangular or user defined mode. User-defined modes, let you enter the expressions for the fields manually. The fields can be complex-valued if you like. The **Port phase** edit field in the previous section has no impact for this mode type because the phase is determined by the entered fields.

Rectangular Modes

To specify a unique rectangular mode you need to set a mode type and a mode number.

- Establish whether the mode is a transverse electric (TE) or a transverse magnetic (TM) mode using the **Mode type** list.
- Enter the **Mode number**, for example, 10 for a TE_{10} mode, or 11 for a TM_{11} mode.

User-Defined Modes

Using the user-defined mode type you can manually specify the eigenmode of the port. To fully specify the mode enter the following data:

- Enter the amplitude of the **Electric field**.
- Enter the **Propagation constant** β. This is frequency dependent for all but TEM modes and a correct frequency-dependent expression must be used.

S-Parameter Variables

The RF Module automatically generates variables for the S-parameters. The port names (use numbers for port sweeps to work correctly) determine the variable names. If you, for example, have 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:

$$S_{11\mathrm{dB}} = 20\log 10(|S_{11}|)$$

The model and physics interface names also appear in front of the variable names so they may vary but typically look something like `mod1.rf.S11dB` etc. The S-parameter variables are added to the predefined quantities in appropriate plot lists. To see the various ways of displaying and exporting S-parameters, see the model examples in the section “Model with S-Parameter Calculations” below.

Port Sweeps and Touchstone Export

The **Port Sweep Settings** section in the Electromagnetic Waves interface node cycles through the ports, computes the entire S-matrix and exports it to a Touchstone file, for an example see the model H-Bend Waveguide 3D in the RF and Microwave Engineering section in the RF Module Model Library (or see Where Do I Access the Documentation and Model Library?).

Model with S-Parameter Calculations

For examples on S-parameter analysis, see the RF Module Model Library models (or see Where Do I Access the Documentation and Model Library?):

- RF and Microwave Engineering/H-Bend Waveguide 3D
- RF and Microwave Engineering/Waveguide Adapter

The former illustrates Port Sweeps and Touchstone file export whereas the latter has a detailed description how to model numerical ports with a boundary mode analysis.

Lumped Ports with Voltage Input

The ports described in the previous section require a detailed specification of the mode, including the propagation constant and field profile. In situations when the mode is difficult to calculate or when there is an applied voltage to the port, a *lumped port* might be a better choice. This is also the appropriate choice when connecting your RF model to an electrical circuit. The lumped port is not as accurate as the ordinary port in terms of calculating S-parameters, but it is easier to use. You can, for example, attach a lumped port as an internal port directly to a printed circuit board or to the transmission line feed of a patch antenna. 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

$$Z_{\text{port}} = \frac{V_{\text{port}}}{I_{\text{port}}}$$

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

where V_{port} is the extracted voltage for the port given by the 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. For more details, see also Lumped Port Parameters.

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 supports the lumped port, you find the **Lumped Port** available as a boundary feature.

THE PORT PROPERTIES SECTION

In the **Port Properties** section of the **Lumped port** feature you specify 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, see the note below. You also set the **Type of Port** which could be **Uniform** or **Coaxial**. Under **Terminal type**, you specify if it is a voltage driven transmission line (**Cable**) port or a **Current** driven port or a **Circuit** port. For the cable terminal type, the **Wave excitation at this port** setting decides whether it is an inport or a listener port. The **Port Voltage** and **Port phase** can also be specified.

Note: 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 if you excite several ports, the S-parameter output is turned off.

The **Port Sweep Settings** section in the Electromagnetic Waves interface node 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.

THE SETTINGS SECTION

In the **Settings** section you specify the **Characteristic impedance** for the cable terminal type or the **Terminal current** for the current cable terminal type.

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 physics interface with the tag `emw`, defines the port impedance variable `emw.Zport_1`.

Example Model—RF Coil

For an example of how to use the lumped port boundary condition, see the model RF Coil in the Tutorial Models section in RF Module Model Library (or see Where Do I Access the Documentation and Model Library?).

ECAD Import

Overview of the ECAD Import

This section explains how to import ECAD files into COMSOL Multiphysics. An ECAD file can, for example, be a 2D layout of a printed circuit board (PCB) that COMSOL Multiphysics imports and converts to a 3D geometry.

EXTRUDING LAYERS

A PCB layout file holds information about all traces in several 2D drawings or layers. During import, each 2D layer is extruded to a 3D object so that all traces get a valid thickness. An standard extrude operation requires that the source plane is identical to the destination plane. This makes it impossible to extrude an entire PCB with several layers, where the source and destination planes in almost all cases simply do not match. It is possible to do several extrude operations, one for each layer, and there is an option to do so. For complex PCBs it is not trivial to put these layers together, and it might therefore take a very long time to go from the geometry node to the Material node or a Physics node in the model tree. In some situations this operation might fail.

As a result of these performance issues, the ECAD Import has its own extrude operation that automatically connects nonmatching planes. In one operation this functionality extrudes and connects all layers, so there is only one geometry object after the import. With only one object, it is trivial to switch to the physics modes. You use this special extrude operation when you use the grouping option **All**.

The special extrude operation is bound to certain rules that the 2D layout must fulfill. If the 2D layout does not comply with these rules, the operation might fail. You can then switch to one of the other grouping options to import the geometry into COMSOL Multiphysics.

Importing ODB++(X) Files

This section describes how to import ODB++(X) files into COMSOL Multiphysics. If your ECAD software supports this format we recommend that you use it, because it usually gives the most efficient geometry model of the layout.

FILE INFORMATION

The ODB++ file format is developed and maintained by Valor (www.valor.com). It is a sophisticated format that handles most of the information needed to manufacture a

PCB. Some of the information is not needed when importing the file into COMSOL Multiphysics, and the program ignores such information during import.

ODB++ exists in two different format versions:

- A single XML file containing all information organized in a hierarchy of XML tags. This file format is usually referred to as ODB++(X), and it is the only format that you currently can import into COMSOL Multiphysics.
- A directory structure with several files, each containing parts of information about the PCB. An entire PCB layout is often distributed as zipped or unzipped tar archives. This version is currently not possible to import into COMSOL Multiphysics.

The ODB++ import reads the layer list and the first step in the file. Multiple step files are not yet supported. From the first step it reads all the layer features and the board outline but currently skips all the package information.

EXTRACTING LAYER STACKUP

The import can read stackup information from the ODB++ file, such as thickness for metal layers and dielectric layers. It is quite common that the layer thickness is not included in the export from the ECAD program, so the layers only get a default thickness. You always have the possibility to change the thickness prior to import on the **Layers** page in the **ECAD Import Options** dialog box, so it is recommended that you check this page before importing.

Importing GDS-II Files

This section describes how to import GDS-II files into COMSOL Multiphysics.

FILE INFORMATION

The GDS-II file format is commonly used for mask layout production used in the manufacturing process of semiconductor devices and MEMS devices. The file is a binary file, containing information about drawing units, geometry objects, and object drawing hierarchy. The drawing hierarchy is made up of a library of cell definitions, where each cell can be instantiated (drawn several times) with scaling, translation, mirroring, and rotation. It is also possible to repeat a cell as an array of drawn objects. This is very useful for mask layouts of integrated circuits, which often consist of millions of transistors. There are usually only a few transistor configurations present on the layout, and each transistor configuration only has to be defined once.

File Extension

The file extension of the GDS-II format is usually `.gds`, and the ECAD import requires it to be so, otherwise it cannot identify the file as a GDS-II file. If the file has a different extension, you must changed it to `.gds` before importing the file.

Supported Features

There are several record types in a GDS file that are of no interest in a geometry import, and these are ignored. There are also a few record types that actually could be imported as a geometry object, but are also ignored. One such example is the Text record, which produce a lot of mesh elements and is usually of no interest in a simulation. Below is a list of the supported record types.

- **Boundary**. A closed polyline object.
- **Box**. A box object.
- **Path**. A path with a thickness.
- **Sref**. An instance of a cell that can be translated, rotated, scaled, and mirrored.
- **Aref**. An n-by-m array of Sref objects.
- **Element**. Specification of a cell.

3D IMPORT OF GDS-II FILES

The GDS-II format does not contain any information about layer thickness and layer position, so any such information has to be supplied by the user. When importing a GDS-II file with the ECAD import, it creates a table for all layers included in the file. In that table it is possible to specify a thickness for each layer and thereby get a 3D structure. This procedure has a few limitations regarding how the GDS layers are organized:

- One layer represents one position in height, so if the file contains two GDS layers that define two objects on the same height, the ECAD import still positions the layers with one layer on top of the other. Several GDS layers on the same height is common for semiconductor layouts, where the fabrication process includes deposition followed by etching and then redepositing of a different layer. Such advanced process schemes cannot be automatically handled correctly by the ECAD import.
- With the grouping option **All**, objects on adjacent layers must not cross each other, because the original edge of the objects must be kept unchanged when two adjacent

layers are merged to form the interface between them. You can get around this by selecting a different grouping option (see ECAD Import).

- Use the 3D GDS-II import with the ECAD import. The standard CAD import of COMSOL Multiphysics does not support pre-reading of the file, so it is not possible to specify any properties the layers (like thickness for example). The ECAD import always reads the file before displaying the import options.

The best way to solve any of these issues is to do the import with the grouping option **By layer**, and manually rearrange the layers by simple move operations so the elevation of the layers are correct. You can do etching by removing a layer from other objects, using the **Difference** toolbar button or the **Boolean Operations** geometry feature.

Importing NETEX-G Files

FILE INFORMATION

The NETEX-G file format is a special format produced by the application NETEX-G by Artwork (www.artwork.com). NETEX-G can read Gerber and drill files that almost any ECAD software can export to because those formats are used when sending the layout to manufacturing. The output file is an ASCII file with a GDS-like structure, containing information about the layout of each layer, the layer thickness, vias, and dielectric layers. The geometry objects are defined and instantiated in the same way as in a GDS file; see the corresponding section in Importing GDS-II Files for a more detailed description.

File Extension

The file extension of the NETEX-G format is not set, but the ECAD import requires it to be `.asc`, otherwise it cannot identify the file as a NETEX-G file. If the file has a different extension, you have to change it before importing it. Throughout the rest of this section, files of this type are referred to as a Netex file.

USING NETEX-G

This section contains a brief description of the main steps to produce a Netex file for import into COMSOL Multiphysics. For specific details on how to use NETEX-G, the user is referred to the NETEX-G manual.

GERBER Layer Files

The first type of input files to NETEX-G is a collection of Gerber files, one for each layer. The ECAD software generates these files when the PCB layout is sent to manufacturing, but they can also be used for interfacing to other programs like

COMSOL Multiphysics. The layer files do not contain any information about layer thickness, layer materials, dielectrics, and electrical connectivity (nets). Furthermore, a standard PCB layout usually consists of a large number of conductors, vias, and symbols printed in metal that are not important for a finite element simulation. With NETEX-G you can reduce the size of the exported layout in the following ways:

- Defining a region to include in the export. This region is drawn directly on a top view of the layout.
- Exclude entire layers from the layout.
- Selecting electrical nets to include in the export in addition to the selected region.
- It is also possible to let NETEX-G include nets in the proximity of the selected nets.

Because the Gerber layer files do not contain any physical information about the layer and dielectrics, you also need to specify this information in NETEX-G.

Some of these steps can also be done during import to COMSOL Multiphysics, for example, excluding layers from the import and changing thickness of the layers.

Drill Files

The connectivity between the layers is defined through drilled holes, known as *vias*. A via can go through the entire circuit board or just between certain layers. Most ECAD programs use the Excellon drill file format to specify the vias, which contains information about via diameter and position. Before generating the final output file from NETEX-G, it is necessary to convert all drill files to Gerber format and include them to the export project in NETEX-G. For each drill file, it is also necessary to specify between which layers the hole goes. Within NETEX-G you can call a tool that directly converts the Excellon drill format into Gerber. After the conversion you also specify the source and destination layers for the drill file.

NETEX-G Export Settings

To reduce the complexity of the output file it is recommended that vias are exported as circles and not as polygon chains. Although the arc recognition utility can detect these polygons, the former option is a bit more robust.

IMPORTING WIREBONDS

The Netex file can contain information about wirebonds or bond wires. Including wirebonds in the geometry often increases the problem size significantly. To get more control over the problem size, you can control the complexity of the imported wires.

Types of Wirebonds

The ECAD import can model the wirebond at three different complexity levels:

- As geometrical edges. This is the simplest form, which works well when the current in the wires is known.
- As solids with a square-shaped cross section. This cross section often produces fewer mesh elements than when using a circular cross section and is also easier for the geometry engine to analyze.
- As solids with a circular cross section.

Wirebonds Models

The Netex file format supports wirebonds models according to the JEDEC standard. It is possible to either define the wirebond as a JEDEC3 or a JEDEC4 model. These models define the bond wire as 3 or 4 segment paths with user-supplied coordinates and elevations. In a Netex file the bond wire goes from a layer to a special die layer, representing the semiconductor die.

Note: Wirebonds are currently not supported with the grouping option set to **All**. Using this option ignores all wirebonds.

ECAD Import Options

ECAD IMPORT

Most PCB layout files mainly contain definitions of 2D objects. The Netex file also contains information about wirebonds. The ECAD import engine first creates the 2D objects for each layer, possibly grouped as one object. Then it extrudes all the objects in each layer according to the information in the file. GDS files contain no information about thickness, so a default value of 100 µm is used for all layers. The ECAD Import allows you to change the layer thickness prior to import. Another alternative is to first import the objects into 2D and then manually extrude them to 3D.

To use this import, add an **Import** node by right-clicking on the **Geometry** node in the model tree. The **Import** section under **Geometry import**, you decide what type of CAD file to import. In the list there are two options for ECAD import: **ECAD file (GDS/ NETEX-G)** and **ECAD file (ODB++)**. Next step is to enter the path to the wanted file or click the **Browse** button that opens a window where you select the file you want to

import. Before clicking the **Import** button you should consider the import options described below.

THE ECAD IMPORT OPTIONS

There are a number of settings that control how to treat the information in the layout file. The content of this section depends on the file type you import.

For GDS and NETEX-G files you can enter a *net* name in the **Net to import** edit field if you want to import a single electrical net beneath the top net in the hierarchy. Leave this edit field empty to import the entire file. (In GDS files, the standard terminology is *cell* instead of net, but structurally they mean the same thing.)

The **Grouping of geometries** list specifies how the imported geometry objects are grouped in the final geometry. The choices for 3D import are:

- **All**. Groups all objects into one single object. This selection makes use of a more efficient extrude algorithm that extrudes and combines all layers directly. Because the import results in only one geometry object, COMSOL Multiphysics does not need to do a complicated analysis of several geometry objects.
- **By layer**. Groups all objects in one layer into one geometry object. The final geometry contains one object for each layer.
- **No grouping**. No grouping of objects is performed. This can be useful for debugging purposes when the other choices fail for some reason. This selection returns all the primitive objects found in the file, so objects with negative polarity are not drawn correctly.

The **Type of import** list specifies how to treat metal layers. The **Full 3D** option imports all metal layers with a thickness. Select the **Metal shell** options if you want to import all metal layers as an embedded boundary between dielectric regions.

For NETEX-G files, bond wires or wirebonds can be imported in three different complexity levels. You choose the level from the **Type of bond wires** list:

- **Edges**. The path of the bond wire is represented only as a geometrical edge. This option has the least complexity and does not produce a large number of mesh elements. There might be some limitations when using these edges in modeling.
- **Blocks**. The bond wire is modeled as a solid with a square cross section.
- **Cylinders**. Same as above but with a circular cross section.

The layer information from the file is shown in the **Layers to import** table. In addition to the layer **Name**, the following columns are provided:

- The **Type** column. This column declares the type of layer. The import treats layers of different types differently. For example, a layer of type **Metal** converts to faces if the option **Type of import** is set to **Metal shell**. The **Outline** type uses a union of the objects in the selected layer as a PCB outline. For ODB++ files, the **Drill** type means that the objects in the layer define drilled via holes through the PCB. For NETEX-G files, the vias are defined within each metal and dielectric layer.
- The numbers in the **Thickness** column can be changed. This column is especially important when importing GDS files because that format does not contain any thickness information, so all layers get a default thickness that you probably want to change.
- The **Import** column. Here you can clear the check box for layers that you do not want to import. Note that if you use the **Metal shells** import type, you cannot import isolated boundaries if the import also includes another solid layer. Then you must perform two imports. The only exception to this rule is when the import results in only face objects.

In most electromagnetic simulations the material between the metal layers is important for the simulation result. For NETEX-G/GDS import, the **Import dielectric regions** check box controls if the import engine also includes the dielectric layers, which in most cases are the actual PCB materials. An ODB++ file usually has the outline of the PCB board defined in the file. If you import a NETEX-G file or a GDS file, it is possible to define the PCB outline using margins for the dielectric material. They define the distance between the exterior of the PCB and the bounding box of all metal layers.

With the **Keep interior boundaries** check box cleared, all interior boundaries of the imported nets are deleted. This keeps the geometry complexity to a minimum and can also make the import more robust in some situations.

Clearing the **Ignore text objects** check box tells the importer to skip all objects in an ODB++ file that have the `TEXT` tag set. It is common that PCB layouts have text written in copper. Such objects increase the problem size and are usually of no interest in a physical simulation.

For NETEX-G/GDS import, other options that can significantly reduce the complexity of imported layouts are the recognition of arcs and straight lines. With the **Recognize arcs** set to **Automatic**, all polygon chains that represent arcs are identified and replaced with more efficient curve objects. With the edit fields appearing when setting

this to **Manual**, you can fine tune the arc recognition. The **Find straight lines** check box also controls whether to convert several polygon segments that lie on a single straight line into a single straight segment. This option uses the number in the **Minimum angle between segments** edit field to determine if a group of segments lies on the same straight line.

Geometry repair is controlled via the **Repair imported data** check box and the **Relative repair tolerance** edit field.

Postimport Operations

MESHING IMPORTED GEOMETRY

The imported geometry often consists of objects with very high aspect ratios, which are hard to mesh with a free tetrahedron mesh generator. As a result, it is often necessary to use interactive meshing of the imported geometry in a by-layer fashion.

The following section describes this procedure in general terms.

This procedure assumes that the top and bottom layers are metal layers. All metal layers can often be meshed using swept meshing, but dielectric layers usually cannot be meshed that way. You begin by meshing from the bottom or top layer, starting with a boundary mesh. Then you mesh layer by layer, where each metal layer gets a swept mesh, and each dielectric layer (with vias) gets a free mesh.

The dielectric layers cannot use a swept mesh because the source and target boundaries usually do not look the same. If there is a surrounding air domain it is usually not possible to use swept meshes for the metal layers either. You must then use tetrahedrons or convert the swept mesh to tetrahedrons before meshing the surrounding domain.

For more details on meshing operations, see Creating Meshes and Creating 3D Swept Meshes in the *COMSOL Multiphysics User's Guide* and Convert in the *COMSOL Multiphysics Reference Guide* (or see Where Do I Access the Documentation and Model Library?).

TUNING IMPORT SETTINGS

Delete Interior Edges

A complex layout produces a large number of faces that can be hard to render. A simple way to reduce the number of faces is to clear the **Keep interior boundaries** check box in the ECAD Import Options. This removes all faces internal to the nets within a layer.

Removing Features

You can remove all features that are not important for your simulation. This is usually best to do before the import in NETEX-G or in the ECAD software. When importing with **Grouping of geometries** set to **None** it is possible to manually delete certain objects after import, but it is recommended to do this only for relatively simple geometries.

PROBLEMS WHEN EXTRUDING LAYERS

Most ECAD or EDA programs support design rule checks (DRC), which test the entire layout and check that all features (vias, conductors, and components) are separated according to certain rules. With such checks the layout is free from overlapping vias and conductors touching other conductors or vias. This also ensures that the special extrude functionality of the ECAD import works properly. If the file contains such design-rule violations, the extrude might fail and throw an error message stating that it could not handle the topology of the layout.

The best approach to handle such problems is to perform a DRC with your ECAD software and produce new layout files. If this is not possible, you can import the layout in 2D and try to identify the problematic features. They can either be in a single layer or at the interface between two adjacent layers. When identified, it is possible to remove them manually using a text editor if you are importing a NETEX-G file or an ODB++ file. It can be hard to find a certain feature, but you can use either the coordinate or the net information to find it. The GDS format is a binary file format so it is very difficult to edit the file manually.

PROBLEMS WITH SEVERAL GEOMETRY OBJECTS

If you do not use the special extrude functionality you get several geometry objects, for example, one for each layer if you choose **By layer** from the **Grouping of geometries** list. After a CAD import COMSOL Multiphysics is in the Geometry branch of the model tree. When you continue to the **Materials** branch if the model tree or to a physics interface node or the **Mesh** branch, the program tries to combine all the objects into one geometry, and this operation might fail if the objects are very complex and have

high aspect rations. You can resolve this either by trying the option **All** in the **Grouping of geometries** list. This creates one combined geometry object by using the special extrude functionality, and with only one object this.

Another possibility is to use assemblies, because then COMSOL Multiphysics does not have to combine the objects (parts). This is controlled by the **Finalize** node in the Geometry branch of the model tree. When using an assembly, you have to use identity pairs to connect the interfaces between the layers.

As a final option, you can choose to not import the dielectric layers. The import then leaves you with isolated metal layers that you have to connect with coupling variables.

Circuit Modeling

Introduction

Electrical circuit modeling capabilities are useful when simulating all sorts of electrical and electro-mechanical 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. Either the field model is used to get a better, more accurate description of a single device in the electrical circuit model or 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 is possible to add features representing circuit elements directly to the model 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 you can analyze its behavior 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.

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

SEMICONDUCTOR DEVICE MODELS

There are three more advanced large signal semiconductor device features available in the **Electrical Circuit** interface. Below, the equivalent circuits and the equations defining their non-ideal circuit elements are given. For a more detailed account on semiconductor device modeling, see Ref. 2.

NPN Bipolar Transistor

Figure 2-5 illustrates the equivalent circuit for the bipolar transistor.

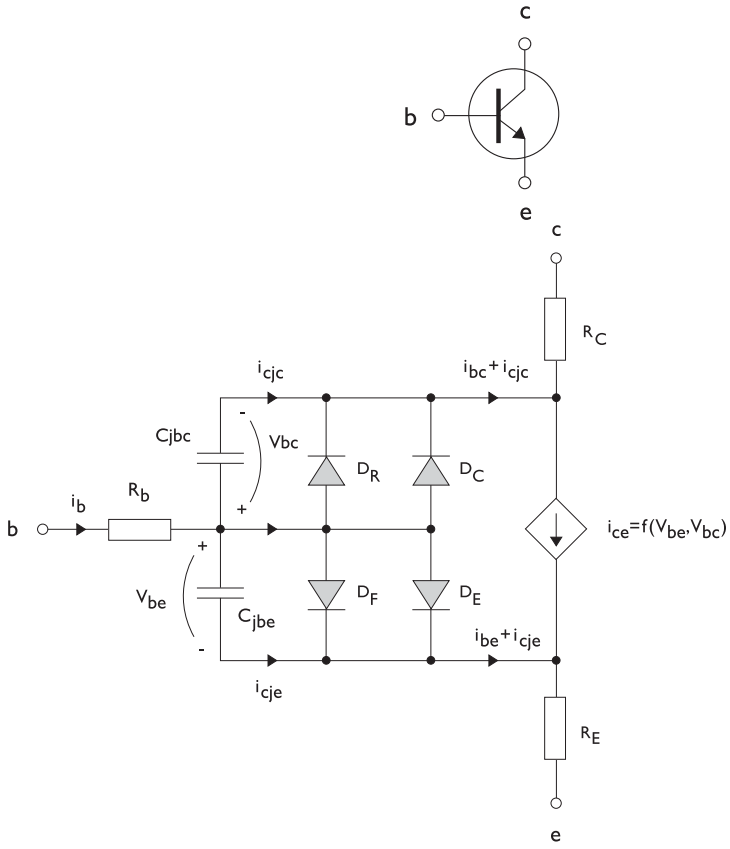


Figure 2-5: A circuit for the bipolar transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$v_{rb} = \frac{1}{A}\left(R_{BM} - \frac{R_B - R_{BM}}{f_{bq}}\right) i_b$$

$$f_{bq} = \frac{1}{2\left(1 - \frac{v_{bc}}{V_{AF}} - \frac{v_{be}}{V_{AR}}\right)}\left(1 + \sqrt{1 + 4I_S\left(\frac{e^{-\frac{v_{be}}{N_F V_T}} - 1}{I_{KF}A} + \frac{e^{-\frac{v_{bc}}{N_R V_T}} - 1}{I_{KR}A}\right)}\right)$$

$$i_{be} = A\left(\frac{I_S}{B_F}\left(e^{-\frac{v_{be}}{N_F V_T}} - 1\right) + I_{SE}\left(e^{-\frac{v_{be}}{N_E V_T}} - 1\right)\right)$$

$$i_{bc} = A\left(\frac{I_S}{B_R}\left(e^{-\frac{v_{bc}}{N_R V_T}} - 1\right) + I_{SC}\left(e^{-\frac{v_{bc}}{N_C V_T}} - 1\right)\right)$$

$$i_{ce} = A\left(\frac{I_S}{f_{bq}}\left(e^{-\frac{v_{be}}{N_F V_T}} + e^{-\frac{v_{bc}}{N_C V_T}}\right)\right)$$

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 2-1: 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
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

TABLE 2-1: BIPOLAR TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
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

n-Channel MOS Transistor

Figure 2-5 illustrates an equivalent circuit for the MOS transistor.

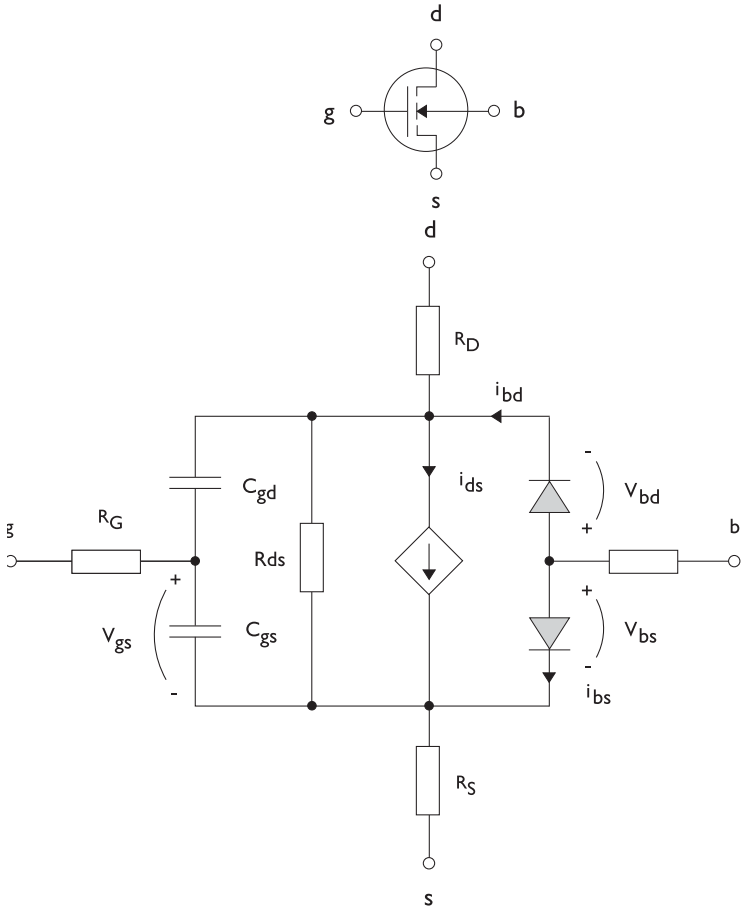


Figure 2-6: A circuit for the MOS transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_{ds} = \begin{cases} \frac{W}{L}\frac{K_P}{2}(1+\Lambda v_{ds})v_{ds}(2v_{th}-v_{ds}) & v_{ds} < v_{th} \\ \frac{W}{L}\frac{K_P}{2}(1+\Lambda v_{ds})v_{th}^2 & v_{ds} \geq v_{th} \\ 0 & v_{ds} < v_{th} \leq 0 \end{cases}$$

$$v_{th} = v_{gs} - (V_{TO} + \Gamma(\sqrt{\Phi - v_{bs}} - \sqrt{\Phi}))$$

$$i_{bd} = I_S\left(e^{-\frac{v_{bd}}{NV_T}} - 1\right)$$

$$i_{bs} = I_S\left(e^{-\frac{v_{bs}}{NV_T}} - 1\right)$$

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 2-1: MOS 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
N	1	Bulk junction ideality factor
P_B	0.75 V	Bulk junction potential

TABLE 2-1: MOS TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
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)	0 $V^{0.5}$	Bulk threshold parameter
Φ (PHI)	0.5 V	Surface potential
Λ (LAMBDA)	0 1/V	Channel-length modulation

Diode

Figure 2-7 illustrates equivalent circuit for the diode.

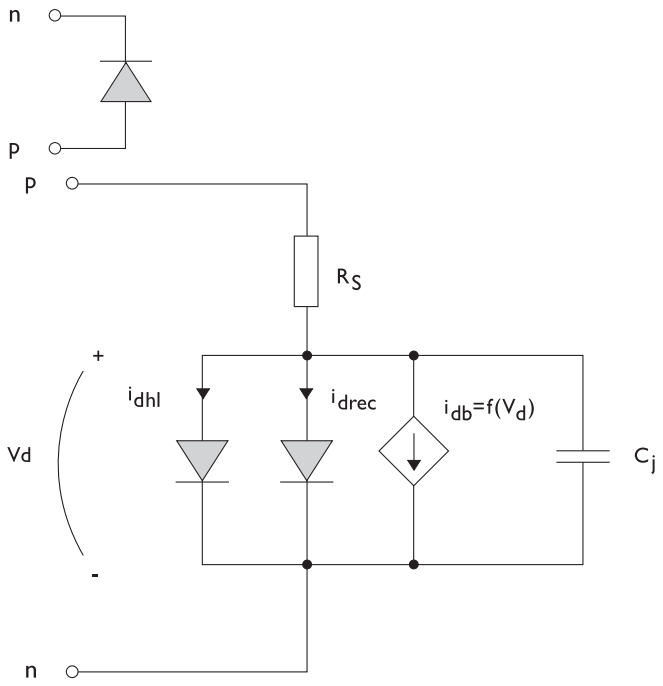


Figure 2-7: A circuit for the diode.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_d = i_{dhl} + i_{drec} + i_{db} + i_c$$

$$i_{dhl} = I_S\left(e^{-\frac{v_d}{NV_T}} - 1\right)\frac{1}{\sqrt{1 + \frac{I_S}{I_{KF}}\left(e^{-\frac{v_d}{NV_T}} - 1\right)}}$$

$$i_{drec} = I_{SR}\left(e^{-\frac{v_d}{N_R V_T}} - 1\right)$$

$$i_{db} = I_{BV}e^{-\frac{v_d + B_V}{N_{BV}V_T}}$$

$$C_j = C_{J0} \times \begin{cases} \left(1 - \frac{v_d}{V_J}\right)^{-M} & v_d < F_C V_J \\ (1 - F_C)^{-1-M}\left(1 - F_C(1+M) + M\frac{v_d}{V_J}\right) & v_d \geq F_C V_J \end{cases}$$

where we need the following model parameters

TABLE 2-1: 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}	1e-09 A	Current at breakdown voltage
I_{KF}	Inf (A)	Corner for high-current roll-off
I_S	1e-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

Example Model Using The Electrical Circuit Interface

The RF Module Model Library model Conical Antenna (or see Where Do I Access the Documentation and Model Library?) contains a transient analysis involving an external circuit, which is added to the model using the SPICE circuit editor. See The Electrical Circuit Interface and Theory For the Electrical Circuit Interface for more information.

REFERENCES

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

2. P. Antognetti and G. Massobrio, Semiconductor Device Modeling with Spice, 2nd Ed. McGraw-Hill, Inc., 1993.

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:

- The propagation constant or eigenfrequency
- The damping in space and time

Eigenfrequency Analysis

The eigenfrequency analysis solves for the eigenfrequency of a model. This study type is available for all physics interfaces in the RF Module. The time-harmonic representation of the fields is more general and includes a complex parameter in the phase

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

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

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

VARIABLES

The following list shows the variables that the eigenvalues 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 subdomain	Scattering boundary condition
E	Nonzero conductivity at adjacent subdomain	Scattering boundary condition
E	Analytical ports	Port boundary condition

These situations require special treatment, especially since it can lead to "singular matrix" or "undefined value" messages if not treated correctly. 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

$$-(-\lambda)\frac{\sqrt{\varepsilon_0\mu_0}\sqrt{\mu_{\mathrm{rbnd}}}}{\sqrt{\varepsilon_{\mathrm{rbnd}}+\dfrac{\sigma_{\mathrm{bnd}}}{(-\lambda)\varepsilon_0}}}(\mathbf{n}\times(\mathbf{n}\times\mathbf{H}))$$

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, see Solving in the *COMSOL Multiphysics User's Guide* for more information. Also see Where Do I Access the Documentation and Model Library?. By default this linearization point is zero, which leads to a division by zero in the expression above. To avoid this problem and also to give a suitable initial guess for the nonlinear eigenvalue problem, it is necessary to provide a "good" linearization point for the eigenvalue solver. You can do this in the **Eigenvalue** node (not the Eigenfrequency node) under the **Solver Sequence** node in the **Study** branch of the model tree. You may first have to generate a solver sequence. In the **Linearization Point** section you check the **Transform point** box and enter a suitable value in the **Point** field.

For example, if you know that the eigenfrequency is close to 1 GHz, enter the eigenvalue `-i*2*pi*1e9` in the edit field.

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

1. Specify that the eigenvalue solver only search for one eigenvalue. You can 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 eigenvalues shift, use the same value with the real part removed.
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.

Mode Analysis

In mode analysis COMSOL Multiphysics solves for the propagation constant, which is possible for the Perpendicular Waves and Boundary-Mode Analysis problem types. The time-harmonic representation is almost the same as for the eigenfrequency analysis, but with a know propagation in the out-of-plane direction

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

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

The following table lists the variables that are influenced by the eigenfrequency 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

3

Review of Electromagnetics

This section 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 postprocessing. There is also a list of electromagnetic quantities with their SI units and symbols.

Note: Also see Theory for the Heat Transfer Interfaces in the *COMSOL Multiphysics User's Guide* for more information (or see Where Do I Access the Documentation and Model Library?).

In this section:

Maxwell's Equations

The problem of electromagnetic analysis on a macroscopic level is the problem of 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* $\mathbf{E}$, the *electric displacement* or *electric flux density* $\mathbf{D}$, the *magnetic field intensity* $\mathbf{H}$, the *magnetic flux density* $\mathbf{B}$, the *current density* $\mathbf{J}$ and the *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

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t}$$

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t}$$

$$\nabla \cdot \mathbf{D} = \rho$$

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

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

$$\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P}$$
$$\mathbf{B} = \mu_0 (\mathbf{H} + \mathbf{M})$$
$$\mathbf{J} = \sigma \mathbf{E}$$

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

$$\varepsilon_0 = \frac{1}{c_0^2 \mu_0} = 8.854 \cdot 10^{-12} \text{ F/m} \approx \frac{1}{36\pi} \cdot 10^{-9} \text{ F/m}$$

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 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, for instance, 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, $\mathbf{P} = \varepsilon_0 \chi_e \mathbf{E}$, 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

$$\mathbf{D} = \varepsilon_0 (1 + \chi_e) \mathbf{E} = \varepsilon_0 \varepsilon_r \mathbf{E} = \varepsilon \mathbf{E}$$
$$\mathbf{B} = \mu_0 (1 + \chi_m) \mathbf{H} = \mu_0 \mu_r \mathbf{H} = \mu \mathbf{H}$$

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

$$\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E} + \mathbf{D}_r$$

The field $\mathbf{D}_\mathrm{r}$ 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

$$\mathbf{B} = \mu_0 \mu_\mathrm{r} \mathbf{H} + \mathbf{B}_\mathrm{r}$$

where $\mathbf{B}_\mathrm{r}$ 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 $\mathbf{J}^\mathrm{e}$. The resulting constitutive relation is

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

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

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

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

In the magnetostatic case where there are no currents present, Maxwell-Ampère's law reduces to $\nabla \times \mathbf{H} = \mathbf{0}$. When this holds, it is also possible to define a *magnetic scalar potential* V_m by the relation

$$\mathbf{H} = -\nabla V_\mathrm{m}$$

Electromagnetic Energy

The electric and magnetic energies are defined as

$$W_\mathrm{e} = \int_V \left(\int_0^D \mathbf{E} \cdot d\mathbf{D} \right) dV = \int_V \left(\int_0^T \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} dt \right) dV$$
$$W_\mathrm{m} = \int_V \left(\int_0^B \mathbf{H} \cdot d\mathbf{B} \right) dV = \int_V \left(\int_0^T \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dt \right) dV$$

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

$$P_{\mathrm{e}} = \int_V \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} dV$$

$$P_{\mathrm{m}} = \int_V \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dV$$

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,

$$P_{\mathrm{h}} = \int_V \mathbf{J} \cdot \mathbf{E} dV$$

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,

$$P_{\mathrm{r}} = \oint_S (\mathbf{E} \times \mathbf{H}) \cdot \mathbf{n} dS$$

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

$$\mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} = \varepsilon \mathbf{E} \cdot \frac{\partial \mathbf{E}}{\partial t} = \frac{\partial}{\partial t} \left(\frac{1}{2} \varepsilon \mathbf{E} \cdot \mathbf{E} \right)$$

$$\mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} = \frac{1}{\mu} \mathbf{B} \cdot \frac{\partial \mathbf{B}}{\partial t} = \frac{\partial}{\partial t} \left(\frac{1}{2\mu} \mathbf{B} \cdot \mathbf{B} \right)$$

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), you get

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

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

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

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 consideration. 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.

Boundary and Interface Conditions

To get a full description of an electromagnetic problem, you also need to 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 electric 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.

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

Phasors

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

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

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

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

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 $j\omega$,

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

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 the RF 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 $\mathrm{Re}(\tilde{\mathbf{E}}(\mathbf{r}))$ by default, which is $\mathbf{E}$ at time $t = 0$. To obtain the solution at a given time, you can specify a phase factor when evaluating and visualizing the results.

Special Calculations

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 you need to perform an eigenmode expansion of the electromagnetic fields on the ports. Assume that you have performed eigenmode analyses on the ports 1, 2, 3, … and that you know the electric field patterns $\mathbf{E}_1, \mathbf{E}_2, \mathbf{E}_3, \ldots$ of the fundamental modes on these ports. Further, assume that the fields are normalized with respect to the integral of the power flow across each port cross section, respectively. Note that this normalization is frequency dependent unless you are dealing with TEM modes. The port excitation is applied using the fundamental eigenmode. The computed electric field $\mathbf{E}_c$ on the port consists of the excitation plus the reflected field. The S-parameters are given by

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

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

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

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

S-PARAMETERS IN TERMS OF POWER FLOW

For a guiding structure in single mode operation, it is also possible to interpret the S-parameters in terms of the power flow through the ports. Such a definition is only

the absolute value of the S-parameters defined in the previous section and does not have any phase information.

The definition of the S-parameters in terms of the power flow is

$$S_{11} = \sqrt{\frac{\text{Power reflected from port 1}}{\text{Power incident on port 1}}}$$

$$S_{21} = \sqrt{\frac{\text{Power delivered to port 2}}{\text{Power incident on port 1}}}$$

$$S_{31} = \sqrt{\frac{\text{Power delivered to port 3}}{\text{Power incident on port 1}}}$$

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

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

and ε is the permittivity. The power flow then becomes

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

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

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

The power flow then becomes

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

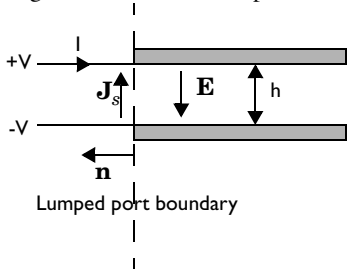
where the last equality holds because the electric field is tangential to the port.

Lumped Port Parameters

In transmission line theory you deal with voltages and currents 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

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

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 you can define an equivalent surface impedance 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 we integrate along the width, w, of the transmission line

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

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

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

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

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 is used in a slightly modified transition boundary condition.

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

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.

Note: 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.

Far-Field Calculations

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

$$\mathbf{E}_p = \frac{jk_0}{4\pi} \mathbf{r}_0 \times \int [\mathbf{n} \times \mathbf{E} - \eta_0 \mathbf{r}_0 \times (\mathbf{n} \times \mathbf{H})] \exp(jk_0 \mathbf{r} \cdot \mathbf{r}_0) dS$$

and in 2D it looks slightly different:

$$\mathbf{E}_p = \sqrt{\lambda_0}\frac{jk_0}{4\pi}\mathbf{r}_0 \times \int [\mathbf{n} \times \mathbf{E} - \eta_0 \mathbf{r}_0 \times (\mathbf{n} \times \mathbf{H})] \exp(jk_0 \mathbf{r} \cdot \mathbf{r}_0) dS$$

In both cases, for scattering problems, the far field in COMSOL 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_0 is the free-space impedance, $\eta_0 = \sqrt{\mu_0 / \varepsilon_0}$.
- k_0 is the free-space wave number.
- λ_0 is the free-space wavelength.
- $\mathbf{r}$ is the radius vector (not a unit vector) of the surface S.
- $\mathbf{E}_p$ is the calculated far field at point p.

The unit vector $\mathbf{r}_0$ can also be interpreted as the direction defined by the angular position (θ, φ), so $\mathbf{E}_p$ is the far field for 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:

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

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

Electromagnetic Quantities

The table below shows the symbol and SI unit for most of the physical quantities that appear in the RF Module. The default values for the permittivity of vacuum, $\varepsilon_0 = 8.854187817 \cdot 10^{-12}$ F/m, and for the permeability of vacuum, $\mu_0 = 4\pi \cdot 10^{-7}$ H/m, require that you provide all other quantities in SI units and that you use meter for the length scale of the geometry. If you draw the geometry using another length scale, it becomes necessary to change the numerical values for the physical quantities accordingly. For example, if you draw the geometry using μm as the length scale, you need to have $\varepsilon_0 = 8.854187817 \cdot 10^{-18}$ F/μm and $\mu_0 = 4\pi \cdot 10^{-13}$ H/μm.

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)	–
Electric 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

TABLE 3-1: ELECTROMAGNETIC QUANTITIES

QUANTITY	SYMBOL	UNIT	ABBREVIATION
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
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

REFERENCES

1. D.K. Cheng, *Field and Wave Electromagnetics*, Addison-Wesley, 1991, 2nd edition.

2. Jianming Jin, *The Finite Element Method in Electromagnetics*, 2nd Edition, Wiley-IEEE Press, May 2002.

3. A. Kovetz, *The Principles of Electromagnetic Theory*, Cambridge University Press, 1990.

4. O. Wilson, *Introduction to Theory and Design of Sonar Transducers*, Peninsula Publishing, 1988.

5. R.K. Wangsness, *Electromagnetic Fields*, John Wiley & Sons, 1986, 2nd edition.

4

The Radio Frequency Interfaces

This section reviews the physics interfaces in the RF Module. For each physics interface, you find information about its applications, fundamental equations, available boundary conditions, and other related information.

In this section:

- The Electromagnetic Waves Interface 
- The Transient Electromagnetic Waves Interface 

Also included is the underlying theory for these interfaces:

- Theory for the RF Interfaces

The Electromagnetic Waves Interface

Note: Also refer to the *COMSOL Multiphysics User's Guide* for detailed information about selecting geometric entities (domains, boundaries, edges, and points) (or see Where Do I Access the Documentation and Model Library?).

With the **Electromagnetic Waves** interface () you can solve the electric field based time harmonic wave equation which strictly is valid for linear media only. The physics interface supports the study types *Frequency domain*, *Eigenfrequency*, *Mode analysis*, and *Boundary mode analysis*. 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 cavity problems. The Mode analysis study type is applicable only for 2D 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 3D and applies to boundaries representing waveguide ports. Only the electric field variant of the time harmonic equation is supported.

INTERFACE IDENTIFIER

This is the physics interface identifier, which you use to reach the fields and variables in expressions, for example.

The identifier appears in the **Identifier** edit field, and you can change it to any unique string that is a valid identifier. The default identifier (for the first interface in the model) is `emw`.

DOMAINS

Select the domains where you want the interface to apply. The default setting is to include all domains in the model.

SETTINGS

Select whether to solve for the **Full field** or the **Scattered field** from the **Solve for** list.

PORT SWEEP SETTINGS

When activated this invokes a parametric sweep over the ports/terminals in addition to the automatically generated frequency sweep. Select the **Activate port sweep** check

box to switch on the port sweep. The generated lumped parameters are in the form of an impedance or admittance matrix depending on the port/terminal settings which consistently must be of either fixed voltage or fixed current type. The **Port parameter name** input field assigns a specific name to the variable that controls the port number solved for during the sweep. The lumped parameters are subject to **Touchstone file export**. File name and path are entered in an input field.

ADVANCED SETTINGS AND DISCRETIZATION

Normally these settings do not need to be changed. See Show More Options: Advanced Settings and Discretizationin the *COMSOL Multiphysics User's Guide* (or see Where Do I Access the Documentation and Model Library?).

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Electric field** $\boldsymbol{E}$. You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.

Wave Equation, Electric

This is the main feature. The governing equation can be written in the form

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \varepsilon_{rc} \mathbf{E} = \mathbf{0}$$

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

$$k_0 = \omega \sqrt{\varepsilon_0 \mu_0} = \frac{\omega}{c_0}$$

where c_0 is the speed of light in vacuum.

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 $\varepsilon_r = n^2$, 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.

DOMAINS

Select the domains where you want the feature to apply. The default setting is to include all domains in the model.

DISPLACEMENT FIELD

Select the **Displacement field model**. The options are:

- **Relative permittivity** - to specify the relative permittivity or take it from the material.
- **Refractive index** - to specify the real and imaginary parts of the refractive index or take them from the material. Note that this assumes a relative permeability of unity and zero conductivity. 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** - to specify a loss tangent for dielectric losses or take it from the material. Note that this assumes zero conductivity.
- **Dielectric loss** - to specify the real and imaginary parts of the relative permittivity or take them from the material. 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.

MAGNETIC FIELD

Select the **Magnetic field model**. The options are:

- **Relative permeability** - to specify the relative permittivity or take it from the material.
- **Magnetic losses** - to specify the real and imaginary parts of the relative permeability or take them from the material. 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.

CONDUCTION CURRENT

Select the **Electric conductivity**. The options are:

- **Conductivity** from the material.
- **Linearized resistivity** specifies a linear temperature dependence that can be taken from the material or be user defined.
- **User defined** conductivity specified by the user. Here optionally anisotropic conductivity can be defined.

Gauge Fixing

This feature is available as a sub feature to the wave equation feature. 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.

DOMAINS

Select the domains where you want to use perfectly matched layers.

Perfectly Matched Layers

The **Perfectly Matched Layers** feature is used to set up perfectly absorbing domains as an alternative to low-reflecting boundary conditions. It imposes a complex-valued coordinate transformation to the selected domain that effectively makes it absorbing at a maintained wave impedance thus eliminating reflections at the interface.

DOMAINS

Select the domains where you want to use perfectly matched layers.

GEOMETRIC SETTINGS

Select the type of perfectly matched layer scaling to use. The options are **Cartesian**, **Cylindrical**, **Spherical**, and **General** (may be less depending on the spatial dimensions modeled); see Perfectly Matched Layers (PMLs).

- For Cartesian and General types of PMLs, no additional user entries are necessary

- For Cylindrical PMLs, you enter the components for the center coordinate $\mathbf{r}_0$ and the center axis direction $\mathbf{r}_{\text{axis}}$.
- For Spherical PMLs, you enter the components for the center coordinate $\mathbf{r}_0$.
- For Cartesian and General types of PMLs, no additional user entries are necessary
- For Cylindrical PMLs, you enter the components for the center coordinate $\mathbf{r}_0$ and the center axis direction $\mathbf{r}_{\text{axis}}$.

PARAMETERS

Tweak two parameters affecting the coordinate transformation: The first one is a **PML scaling factor**, which should be changed from its default of unity if the domain thickness is much different from one wavelength. Also the **PML order** can be specified. For more details, see Perfectly Matched Layers (PMLs).

Initial Values

The **Initial Values** feature adds an initial value for the electric field that can serve as an initial guess for a nonlinear solver. If you need to specify more than one set of initial values, you can add additional **Initial Values** feature from the **Other** menu when you right-click the main feature for the physics interface.

DOMAINS

Select the domains where you want to define an initial value.

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.

Boundary and Interface Conditions

With no surface currents present the interface 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. As we are solving for $\mathbf{E}$, 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.

The following boundary features are available:

- Electric Field
- Perfect Electric Conductor
- Surface Current
- Perfect Magnetic Conductor
- Scattering Boundary Condition
- Impedance Boundary Condition
- Transition Boundary Condition
- Port
- Lumped Port
- Far Field Calculation
- Periodic Boundary Condition

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

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 conditions 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 (perfect electric conductor) is described in the next section.

BOUNDARIES

Select the boundaries where you want to specify the electric field.

ELECTRIC FIELD

Enter the value or expression for the components of the electric field $\mathbf{E}_0$.

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 "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.

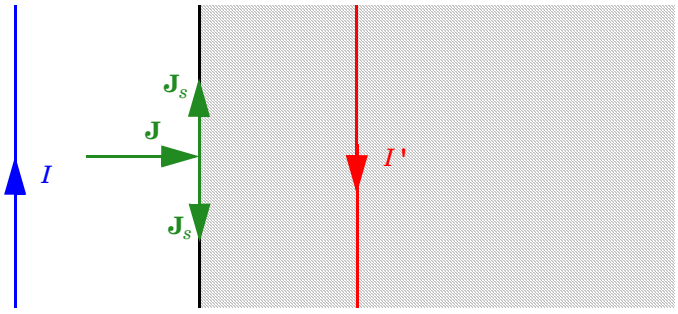


Figure 4-1: The perfect electric conductor boundary condition is used on exterior and interior boundaries representing the surface of a lossless metallic conductor or (on exterior boundaries) representing a symmetry cut. The shaded (metallic) region is not part of the model but still carries effective 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.

BOUNDARIES

Select the boundaries where you want to specify a perfect electric conductor.

Magnetic Field

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

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

BOUNDARIES

Select the boundaries where you want to specify the magnetic field.

MAGNETIC FIELD

Enter the value or expression for the components of the magnetic field $\mathbf{H}_0$.

Surface Current

The surface current boundary condition

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

lets you specify 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.

BOUNDARIES

Select the boundaries where you want to specify a surface current.

SURFACE CURRENT

Enter values or expressions for the components of the surface current density in the J_s edit fields.

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.

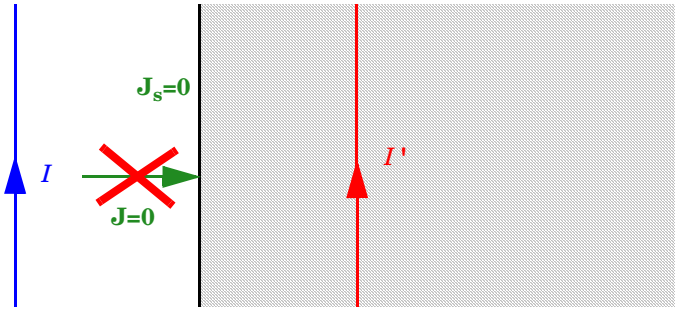


Figure 4-2: The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut. The shaded (high impedance) region is not part of the model but nevertheless carries effective mirror images of the sources. Note also that 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 discontinuous.

BOUNDARIES

Select the boundaries that you want to model as perfect magnetic conductors.

Scattering Boundary Condition

Use the scattering boundary condition when you want a boundary to be 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

$$\mathbf{E} = \mathbf{E}_{sc} e^{-jk(\mathbf{n}\cdot\mathbf{r})} + \mathbf{E}_0 e^{-jk(\mathbf{k}\cdot\mathbf{r})} \quad \text{Plane scattered wave}$$

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

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

The field $\mathbf{E}_0$ is the incident plane wave that travels in the direction $\mathbf{k}$. Note that the boundary condition is transparent for incoming (but not outgoing) plane waves with any angle of incidence.

Note: 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 a more general way of modeling an open boundary, see Perfectly Matched Layers.

For cylindrical waves you have to 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 wave you have to specify the center of the sphere around which the wave is spherical.

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

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

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

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

BOUNDARIES

Select the boundaries where you want to specify the scattering boundary condition.

SCATTERING BOUNDARY CONDITION

Specify the **Incident field** as **Wave given by E field** or as **Wave given by H field** and enter the expressions for the components of the incident field in the $\mathbf{E}_0 / \mathbf{H}_0$ edit field. You can also specify the wave type for which the boundary is absorbing. Here the choices for **Wave type** are **Spherical wave**, **Cylindrical wave** and **Plane wave**. You also need to specify the **Wave direction**.

Impedance Boundary Condition

The impedance boundary condition

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

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 reflective index

$$N\;=\;\sqrt{\frac{\mu\varepsilon_c}{\mu_1\varepsilon_1}}$$

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

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

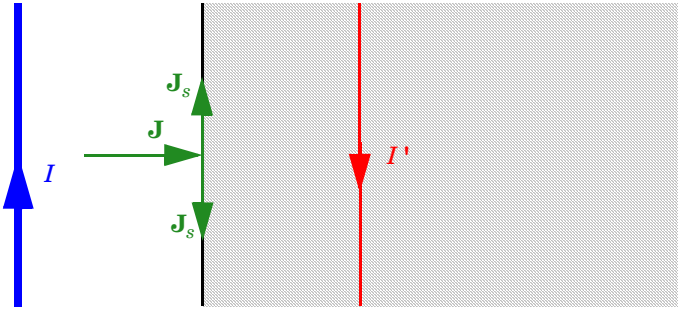


Figure 4-3: The impedance boundary condition is used on exterior boundaries representing the surface of a lossy domain. The shaded (lossy) region is not part of the model. The effective induced image currents are of reduced magnitude due to losses. Any current flowing into the boundary is perfectly balanced by induced surface currents as for the perfect electric conductor boundary condition. The tangential electric field is generally small but non zero at the boundary.

BOUNDARIES

Select the boundaries where you want to specify the impedance boundary condition.

IMPEDANCE BOUNDARY CONDITION

This section has the following material properties for the domain outside the boundary, which this boundary condition approximates:

- The relative permeability, μ_r
- The relative permittivity, ε_r
- The electric conductivity, σ

For all of these properties, the default setting in the corresponding **Relative permeability**, **Relative permittivity**, and **Electric conductivity** lists is **From material**, which means that the values are taken from the material. To specify another values for any of these properties, select **User-defined** from the corresponding list and then enter a value or expression in the edit field that appears.

In addition, you can enter the values of expressions for the components of a source electric field in the $\mathbf{E}_s$ edit fields.

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 it has not 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:

$$\mathbf{J}_{s1} = \frac{(Z_S\mathbf{E}_{t1} - Z_T\mathbf{E}_{t2})}{Z_S^2 - Z_T^2}$$

$$\mathbf{J}_{s2} = \frac{(Z_S\mathbf{E}_{t2} - Z_T\mathbf{E}_{t1})}{Z_S^2 - Z_T^2}$$

$$Z_S = \frac{-j\omega\mu}{k}\frac{1}{\tan(kd)}$$

$$Z_T = \frac{-j\omega\mu}{k}\frac{1}{\sin(kd)}$$

$$k = \omega\sqrt{(\varepsilon + (\sigma/(j\omega)))\mu}$$

Where indices 1 and 2 refer to the different sides of the layer.

BOUNDARIES

Select the boundaries where you want to specify the transition boundary condition.

TRANSITION BOUNDARY CONDITION

This section has the following material properties for the thin layer which this boundary condition approximates:

- The relative permeability, μ_r
- The relative permittivity, ε_r
- The thickness, d
- The electric conductivity, σ

For all of these properties, he default setting in the corresponding **Relative permeability**, **Relative permittivity**, and **Electric conductivity** lists is **From material**, which means that the values are taken from the material. To specify another values for any of these properties, select **User-defined** from the corresponding list and then enter a value or expression in the edit field that appears.

Port

Use the port boundary condition to specify wave type ports. Ports support S-parameter calculations but can be used just for exciting the model. See S-Parameter Calculations in COMSOL Multiphysics: Ports for more information.

BOUNDARIES

Select the boundaries where you want to specify the port.

PORT PROPERTIES

Specify 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. Also set the **Type of Port** which could be **User defined**, **Numeric**, or (analytical modes) **Rectangular** (TE or TM) or **Coaxial** (only TEM).

The **Numeric** option only works if the Boundary Mode Analysis study type has been selected. For an example, see the model Waveguide Adapter in the RF and Microwave Engineering section in the RF Module Model Library (also see Where Do I Access the Documentation and Model Library?).

Note: For **Coaxial** ports to work well in 3D the connecting boundaries should be assigned the impedance boundary condition rather than the default PEC condition. Alternatively use the lumped coaxial port as that works with connecting PEC boundaries in 3D.

The **Wave excitation at this port** setting decides whether it is an in-port or a listener port, when **On** the **Port input power** and **Port phase** can also be specified.

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 if you excite several ports, the S-parameter output is turned off. The **Port Sweep Settings** section in the Electromagnetic Waves interface node 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.

THE PORT MODE SETTINGS SECTION

Specify the details of a rectangular or user defined mode. User-defined modes, let you enter the expressions for the fields manually. The fields can be complex-valued if you like. The **Port phase** edit field in the previous section has no impact for this mode type because the phase is determined by the entered fields.

Rectangular Modes

To specify a unique rectangular mode you need to set a mode type and a mode number.

- Establish whether the mode is a transverse electric (TE) or a transverse magnetic (TM) mode using the **Mode type** list.
- Enter the **Mode number**, for example, 10 for a TE_{10} mode, or 11 for a TM_{11} mode.

User-Defined Modes

Using the user-defined mode type you can manually specify the eigenmode of the port. To fully specify the mode enter the following data:

- Enter the amplitude of the **Electric field**.
- Enter the **Propagation constant** β. This is frequency dependent for all but TEM modes and a correct frequency-dependent expression must be used.

Lumped Port

The **Lumped Port** feature is used 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 general rule is that the lumped port must be applied between two metallic boundaries (PEC or Impedance type boundaries) separated by a distance much smaller than the wavelength.

BOUNDARIES

Select the boundaries where you want to specify the lumped port.

PORT PROPERTIES

Specify 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. Also set the **Type of Port** which could be **Uniform** or **Coaxial**. Under **Terminal type**, you specify if it is a voltage driven transmission line (**Cable**) port or a **Current** driven port or a **Circuit** port. For the cable terminal type, the **Wave excitation at this port** setting decides whether it is an inport or a listener port. The **Port Voltage** and **Port phase** can also be specified.

Note: 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 if you excite several ports, the S-parameter output is turned off.

The **Port Sweep Settings** section in the Electromagnetic Waves interface node 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

Specify the **Characteristic impedance** for the cable terminal type or the **Terminal current** for the current cable terminal type.

Far Field Calculation

To define far-field variables use the **Far Field Calculation** feature. This is explained in more detail in Far-Field Support in the Electromagnetic Waves Interface.

BOUNDARIES

Select the boundaries where you want to specify the source aperture for the far field.

FAR FIELD CALCULATION

Enter a name for the far electric field. You can also specify if you want to use symmetry planes in your model when calculating the far-field variable. The symmetry planes have to coincide with one of the Cartesian coordinate planes. For each of these planes it is possible to select the type of symmetry you use, which can be of either **Symmetry in E (PMC)** or **Symmetry in H (PEC)**. Make the choice here match the boundary condition you used for the symmetry boundary. Using these settings, you can include the parts of the geometry that are not in the model for symmetry reasons in the far-field analysis.

Periodic Boundary Condition

The **Periodic Boundary Condition** sets up a periodicity between the selected boundaries. See Periodic Boundary Conditions for more details on this boundary condition.

BOUNDARIES

Select the boundaries where you want to define a periodic condition. The software automatically identifies the boundaries as either source boundaries or destination boundaries. This works fine for cases like opposing parallel boundaries. In other cases you can use the Destination Selection subnode (see below) to control the destination (right-click **Periodic Condition** in the Model Tree and choose **Destination Selection**). By default it contains the selection that COMSOL Multiphysics has identified.

PERIODIC CONDITION

From the **Type of periodicity** list, select **Continuity** (the default) to make the electric field periodic (equal on the source and destination), select **Antiperiodicity** to make it antiperiodic, or select **Floquet periodicity** (Electromagnetic Waves interface only) to use a Floquet periodicity (Bloch-Floquet periodicity). For the **Floquet periodicity** you are also prompted for the **k-vector for Floquet periodicity**.

Magnetic Current

The **Magnetic Current** feature specifies a magnetic line current along one or more edges.

EDGES

Select the edges that you want to carry a magnetic current.

MAGNETIC CURRENT

Specify the **Magnetic current**.

Edge Current

The **Edge Current** feature specifies an electric line current along one or more edges.

EDGES

Select the edges that you want to carry an electric edge current.

EDGE CURRENT

Specify the **Edge current**.

Perfect Electric Conductor, Edges

The **Perfect Electric Conductor, Edges** condition

$$\mathbf{t} \cdot \mathbf{E} = 0$$

sets the tangential component of the electric field to zero.

EDGES

Select the edges that you want to be a perfect electric conductor.

Line Current (Out of Plane)

The **Line Current (Out of Plane)** feature specifies a line current out of the modeling plane. In axially symmetric geometries this is the rotational direction, in 2D geometries this is the z-direction.

POINTS

Select the points where you want to add a line current.

LINE CURRENT (OUT OF PLANE)

Specify the **Out of plane current**.

Electric Point Dipole

It is possible to add point dipoles to both 2D and 3D models. To add this feature, right-click the **Electromagnetic Waves** node and choose **Electric Point Dipole** from the **Points** menu. The **Electric Point Dipole** form contains the following sections:

POINTS

Select the points where you want to add an electric point dipole.

DIPOLE SPECIFICATION

Apply a point dipole $\mathbf{P}$ (SI unit: Am) to points. This represents the limiting case of when the length d of a current filament carrying uniform current I approaches zero at while maintaining the product between I and d. The dipole moment is a vector entity with the positive direction set by the current flow. You select a **Dipole specification** either as **Magnitude and direction** or as **Dipole moment**.

SETTINGS

Enter the numbers according to what has been selected for the **Dipole specification**.

Magnetic Point Dipole

It is possible to add point dipoles to both 2D and 3D models. To add a Magnetic Point Dipole feature, right-click the **Electromagnetic Waves** node and choose **Magnetic Point Dipole** from the **Points** menu.

POINTS

Select the points where you want to add an electric point dipole.

DIPOLE SPECIFICATION

Apply a point dipole $\mathbf{M}$ (SI unit: Am^2) to points. This represents the limiting case of when the cross-section area a of a circular current loop carrying uniform current I approaches zero at while maintaining the product between I and a. The dipole moment is a vector entity with the positive direction set by the curl of the current. You select a **Dipole specification** either as **Magnitude and direction** or as **Dipole moment**.

SETTINGS

Enter the numbers according to what has been selected for the **Dipole specification**.

The Transient Electromagnetic Waves Interface

With the **Transient Electromagnetic Waves** interface ( ) you can solve a transient wave equation for the magnetic vector potential.

INTERFACE IDENTIFIER

This is the physics identifier, which you use to reach the fields and variables in expressions, for example.

The identifier appears in the **Identifier** edit field, and you can change it to any unique string that is a valid identifier. The default identifier (for the first interface in the model) is `temw`.

DOMAINS

Select the domains where you want the interface to apply. The default setting is to include all domains in the model.

SETTINGS

Select whether to solve for the **Full field** or the **Scattered field** from the **Solve for** list.

ADVANCED SETTINGS AND DISCRETIZATION

Normally these settings do not need to be changed. See Show More Options: Advanced Settings and Discretization in the *COMSOL Multiphysics User's Guide* (or see Where Do I Access the Documentation and Model Library?).

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Magnetic vector potential** A. You can change the name in the corresponding edit field, but the names of fields and dependent variables must be unique within a model.

Wave Equation, Electric

This is the main feature. The governing equation can be written in the form

$$\mu_0\sigma\frac{\partial \mathbf{A}}{\partial t} + \mu_0\varepsilon_0\frac{\partial}{\partial t}\left(\varepsilon_r\frac{\partial \mathbf{A}}{\partial t}\right) + \nabla \times (\mu_r^{-1}\nabla \times \mathbf{A}) = 0$$

for transient problems with the constitutive relations $\mathbf{B} = \mu_0\mu_r\mathbf{H}$ and $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E}$. Other constitutive relations can also be handled for transient problems.

DOMAINS

Select the domains where you want the feature to apply. The default setting is to include all domains in the model.

DISPLACEMENT FIELD

Select the **Displacement field model**. The options are:

- **Relative permittivity** - to specify the relative permittivity or take it from the material.
- **Refractive index** - to specify the real and imaginary parts of the refractive index or take them from the material. Note that this assumes a relative permeability of unity and zero conductivity. 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.
- **Polarization** - to input an explicit electric polarization vector.
- **Remanent displacement** to input an explicit remanent displacement vector.

MAGNETIC FIELD

Select the **Magnetic field model**. The options are:

- **Relative permeability** - to specify the relative permittivity or take it from the material.
- **Remanent flux density** - to input an explicit remanent flux density vector.
- **Magnetization** - to input an explicit magnetic polarization vector.

CONDUCTION CURRENT

Select the **Electric conductivity**. The options are:

- **Conductivity** from the material.
- **Linearized resistivity** specifies a linear temperature dependence that can be taken from the material or be user defined.
- **User defined** conductivity specified by the user. Here optionally anisotropic conductivity can be defined.

Gauge Fixing

This feature us available as a subfeature to the wave equation feature. 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.

DOMAINS

Select the domains where you want to use perfectly matched layers.

Initial Values

The **Initial Values** feature adds an initial value for the magnetic vector potential and its time derivative that can serve as an initial condition for a transient simulation. If you need to specify more than one set of initial values, you can add additional **Initial Values** feature from the **Other** menu when you right-click the main feature for the physics interface.

DOMAINS

Select the domains where you want to define an initial value.

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: Wb/m/s). The default values are 0.

Boundary and Interface Conditions

With no surface currents present the interface 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 which field you are solving 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.

The following boundary features are available:

- Electric Field
- Magnetic Field
- Surface Current
- Perfect Magnetic Conductor
- Scattering Boundary Condition
- Lumped Port
- Periodic Boundary Condition
- Perfect Electric Conductor, Edges

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

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 conditions 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 (perfect electric conductor) is described in the next section.

BOUNDARIES

Select the boundaries where you want to specify the electric field.

ELECTRIC FIELD

Enter the value or expression for the components of the electric field $\mathbf{E}_0$.

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 "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.

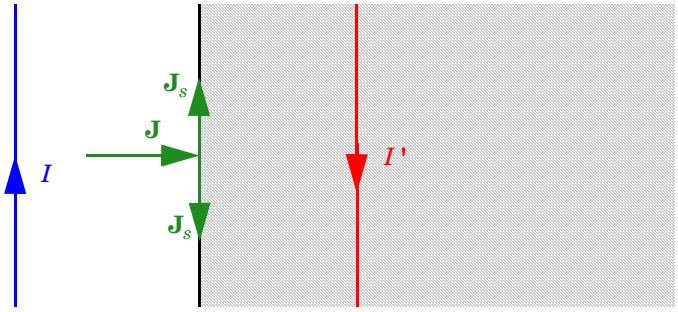


Figure 4-4: The perfect electric conductor boundary condition is used on exterior and interior boundaries representing the surface of a lossless metallic conductor or (on exterior boundaries) representing a symmetry cut. The shaded (metallic) region is not part of the model but still carries effective 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.

BOUNDARIES

Select the boundaries where you want to specify a perfect electric conductor.

Magnetic Field

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

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

BOUNDARIES

Select the boundaries where you want to specify the magnetic field.

MAGNETIC FIELD

Enter the value or expression for the components of the magnetic field $\mathbf{H}_0$.

Surface Current

The surface current boundary condition

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

lets you specify 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.

BOUNDARIES

Select the boundaries where you want to specify a surface current.

SURFACE CURRENT

Enter values or expressions for the components of the surface current density in the J_s edit fields.

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.

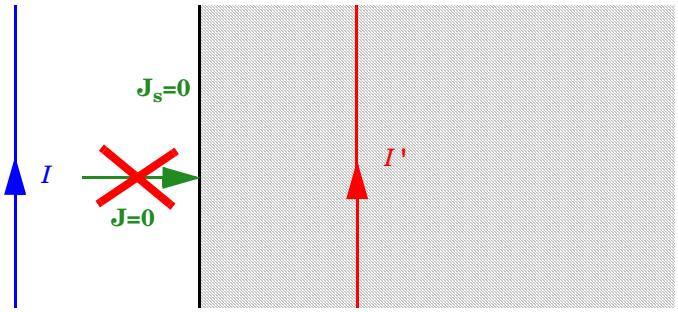


Figure 4-5: The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut. The shaded (high impedance) region is not part of the model but still carries effective mirror images of the sources. Note also that any 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 magnetic vector potential (and in dynamics the tangential electric field) discontinuous.

BOUNDARIES

Select the boundaries that you want to model as perfect magnetic conductors.

Scattering Boundary Condition

Use the scattering boundary condition when you want a boundary to be transparent for a scattered wave. The boundary condition is also transparent for an incoming plane wave. The scattered (outgoing) wave types which the boundary condition is perfectly transparent for are

$$\mathbf{E} = \mathbf{E}_{sc} e^{-jk(\mathbf{n}\cdot\mathbf{r})} + \mathbf{E}_0 e^{-jk(\mathbf{k}\cdot\mathbf{r})} \quad \text{Plane scattered wave}$$

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

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

The field $\mathbf{E}_0$ is the incident plane wave which travels in the direction $\mathbf{k}$. Note that the boundary condition is transparent for incoming (but not outgoing) plane waves with any incidence angle.

Note: 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 cylindrical waves you have to 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 wave you have to specify the center of the sphere around which the wave is spherical.

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

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

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

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

BOUNDARIES

Select the boundaries where you want to specify the scattering boundary condition.

SCATTERING BOUNDARY CONDITION

Specify the **Incident field** as **Wave given by E field** or as **Wave given by H field** and enter the expressions for the components of the incident field in the $\mathbf{E}_0$ / $\mathbf{H}_0$ edit field. You can also specify the wave type for which the boundary is absorbing. Here the choices for **Wave type** are **Spherical wave**, **Cylindrical wave** and **Plane wave**. You also need to specify the **Wave direction**.

Lumped Port

The **Lumped Port** feature is used 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 general rule is that the lumped port must be applied between two metallic boundaries (PEC) separated by a distance much smaller than the wavelength.

BOUNDARIES

Select the boundaries where you want to specify the lumped port.

PORT PROPERTIES

Specify a unique **Port Name**. You also set the **Type of Port** which could be **Uniform** or **Coaxial**. Under **Terminal type**, you specify if it is a voltage driven transmission line (**Cable**) port or a **Current** driven port or a **Circuit** port. For the cable terminal type, the **Wave excitation at this port** setting decides whether it is an inport or a listener port. The **Port Voltage** and **Port phase** can also be specified.

SETTINGS

Specify the **Characteristic impedance** for the cable terminal type or the **Terminal current** for the current cable terminal type.

Periodic Boundary Condition

The **Periodic Boundary Condition** sets up a periodicity between the selected boundaries. See Periodic Boundary Conditions for more details on this boundary condition.

BOUNDARIES

Select the boundaries where you want to define a periodic condition. The software automatically identifies the boundaries as either source boundaries or destination boundaries. This works fine for cases like opposing parallel boundaries. In other cases you can use the Destination Selection subnode (see below) to control the destination (right-click **Periodic Condition** in the Model Tree and choose **Destination Selection**). By default it contains the selection that COMSOL Multiphysics has identified.

PERIODIC CONDITION

From the **Type of periodicity** list, select **Continuity** (the default) to make the electric field periodic (equal on the source and destination), select **Antiperiodicity** to make it antiperiodic, or select **Floquet periodicity** (Electromagnetic Waves interface only) to use a Floquet periodicity (Bloch-Floquet periodicity). For the **Floquet periodicity** you are also prompted for the **k-vector for Floquet periodicity**.

Edge Current

The **Edge Current** feature specifies a line current along one or more edges.

EDGES

Select the edges that you want to an edge current.

EDGE CURRENT

Specify the **Edge current**.

Perfect Electric Conductor, Edges

The perfect electric conductor edge condition

$$\mathbf{t} \cdot \mathbf{E} = 0$$

sets the tangential component of the electric field to zero.

EDGES

Select the edges that you want to be a perfect electric conductor.

Line Current (Out of Plane)

The **Line Current (Out of Plane)** feature specifies a line current out of the modeling plane. In axially symmetric geometries this is the rotational direction, in 2D geometries this is the z-direction.

POINTS

Select the points where you want to add a line current.

LINE CURRENT (OUT OF PLANE)

Specify the **Out of plane current**.

Electric Point Dipole

It is possible to add point dipoles to both 2D and 3D models. To add an Electric Point Dipole feature, right-click the **Electromagnetic Waves** node and choose **Electric Point Dipole** from the **Points** menu.

POINTS

Select the points where you want to add an electric point dipole.

DIPOLE SPECIFICATION

Apply a point dipole $\mathbf{P}$ (SI unit: Am) to points. This represents the limiting case of when the length d of a current filament carrying uniform current I approaches zero at while maintaining the product between I and d. The dipole moment is a vector entity with the positive direction set by the current flow. You select a **Dipole specification** either as **Magnitude and direction** or as **Dipole moment**.

SETTINGS

Enter the numbers according to what has been selected for the **Dipole specification**.

Magnetic Point Dipole

It is possible to add point dipoles to both 2D and 3D models. To add a Magnetic Point Dipole feature, right-click the **Electromagnetic Waves** node and choose **Magnetic Point Dipole** from the **Points** menu.

POINTS

Select the points where you want to add an electric point dipole.

DIPOLE SPECIFICATION

Apply a point dipole $\mathbf{M}$ (SI unit: Am^2) to points. This represents the limiting case of when the cross-section area a of a circular current loop carrying uniform current I approaches zero at while maintaining the product between I and a. The dipole moment is a vector entity with the positive direction set by the curl of the current. You select a **Dipole specification** either as **Magnitude and direction** or as **Dipole moment**.

SETTINGS

Enter the numbers according to what has been selected for the **Dipole specification**.

Theory for the RF Interfaces

Formulations for high-frequency waves can be derived from Maxwell-Ampère's and Faraday's laws,

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t}$$

$$\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t}$$

Using the constitutive relations for linear materials $\mathbf{D} = \varepsilon\mathbf{E}$ and $\mathbf{B} = \mu\mathbf{H}$ as well as a current $\mathbf{J} = \sigma\mathbf{E}$, these two equations become

$$\nabla \times \mathbf{H} = \sigma\mathbf{E} + \frac{\partial \varepsilon\mathbf{E}}{\partial t}$$

$$\nabla \times \mathbf{E} = -\mu\frac{\partial \mathbf{H}}{\partial t}$$

Time Domain Equation

The relations $\mu\mathbf{H} = \nabla \times \mathbf{A}$ and $\mathbf{E} = -\partial\mathbf{A}/\partial t$ make it possible to rewrite Maxwell-Ampère's law using the magnetic potential.

$$\mu_0\sigma\frac{\partial \mathbf{A}}{\partial t} + \mu_0\frac{\partial}{\partial t}\varepsilon\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_r^{-1}\nabla \times \mathbf{A}) = 0$$

This is the equation used by the *Transient Electromagnetic Waves* interface. It is suitable for the simulation of non-sinusoidal waveforms or non linear media.

Frequency Domain Equation

Writing the fields on a time-harmonic form, assuming a sinusoidal excitation and linear media,

$$\mathbf{E}(x, y, z, t) = \mathbf{E}(x, y, z)e^{j\omega t}$$

$$\mathbf{H}(x, y, z, t) = \mathbf{H}(x, y, z)e^{j\omega t}$$

the two laws can be combined into a time harmonic equation for the electric field, or a similar equation for the magnetic field

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

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

The first of these, based on the electric field is used in the *Electromagnetic Waves* interface.

INTRODUCING LOSSES IN THE FREQUENCY DOMAIN

Both time harmonic equations contain the *complex permittivity*,

$$\varepsilon_c = \varepsilon - j\frac{\sigma}{\omega}$$

The conductivity gives rise to ohmic losses in the medium. Other types of losses, for example polarization losses, are customarily given as an imaginary part of a complex permittivity,

$$\varepsilon_c = \varepsilon' - j\varepsilon''$$

where ε' is the real part of ε, and all losses are given by ε''.

Alternatively, it is possible to define an equivalent conductivity representing all losses:

$$\sigma = \omega\varepsilon''$$

Note that the time harmonic form, especially in the physics and optics literature often is written with a minus sign (and "i" instead of "j").

$$\mathbf{E}(x, y, z, t) = \mathbf{E}(x, y, z)e^{-i\omega t}$$

$$\mathbf{H}(x, y, z, t) = \mathbf{H}(x, y, z)e^{-i\omega t}$$

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 optics literature usually must be conjugated before using in a COMSOL model; see also below for refractive index.

REFRACTIVE INDEX IN OPTICS AND PHOTONICS

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

$$\varepsilon_{rc} = \bar{n}^2$$

that is

$$\varepsilon'_r = n^2 - \kappa^2$$
$$\varepsilon''_r = 2n\kappa$$

The inverse relations are

$$n^2 = \frac{1}{2}(\varepsilon'_r + \sqrt{\varepsilon'^2_r + \varepsilon''^2_r})$$
$$\kappa^2 = \frac{1}{2}(-\varepsilon'_r + \sqrt{\varepsilon'^2_r + \varepsilon''^2_r})$$

The parameter κ represents a damping of the electromagnetic wave.

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 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 physics 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 condition and the electric field condition, allow you to 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.

5

The Electrical Circuit Interface

This section summarizes the functionality of the electrical circuit interface.

In this section:

- The Electrical Circuit Interface 

Also included is the underlying theory for the interface:

- Theory For the Electrical Circuit Interface

The Electrical Circuit Interface

Note: Also refer to the *COMSOL Multiphysics User's Guide* for detailed information about editing node properties, names, and identifiers, as well as about selecting geometric entities (domains, boundaries, edges, and points), for example (or see Where Do I Access the Documentation and Model Library?).

The **Electrical Circuit** interface () provides the equations for modeling electrical circuits with or without connections to a distributed fields model, solving for the voltages, currents and charges associated with the circuit elements.

For more details on the equations solved by this physics interface, see the Theory For the Electrical Circuit Interface.

When you add an Electrical Circuit interface, it adds a default **Ground Node** feature and associates that with node zero in the electrical circuit.

Note that circuit nodes are nodes in the electrical circuit and should not be confused with nodes in the model tree of COMSOL Multiphysics. Circuit node names are not restricted to numerical values but can be arbitrary character strings.

Ground Node

The **Ground Node** feature adds a ground node with the default node number zero to the electrical circuit. This is the default feature in the **Electrical Circuit** interface.

GROUND CONNECTION

Set the **Node name** for the ground node in the circuit. The convention is to use zero for the ground node.

Resistor

The **Resistor** feature connects a resistor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Capacitor

The **Capacitor** feature connects a capacitor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the capacitor. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Capacitance** of the capacitor.

Inductor

The **Inductor** feature connects an inductor between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the inductor. If the ground node is involved, the convention is to use zero for this.

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.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Source type** which should be adapted to the selected study type. It can be **DC-source**, **AC-source** or a time-dependent **Sine source**. Depending on the choice of source, you also specify the **Voltage**, $\mathbf{V_{src}}$, the offset **Voltage**, $\mathbf{V_{off}}$, the **Frequency** and the

Source phase. All values are peak values rather than RMS. Note that for the AC source, the frequency is a global input set by the solver so 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.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the current source. The first node represents the positive reference terminal from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the **Source type** which should be adapted to the selected study type. It can be **DC-source**, **AC-source** or a time-dependent **Sine source**. Depending on the choice of source, you also specify the **Current**, $\mathbf{I_{src}}$, the offset **Current**, $\mathbf{I_{off}}$, the **Frequency** and the **Source phase**. All values are peak values rather than RMS. Note that for the AC source, the frequency is a global input set by the solver so 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.

NODE CONNECTIONS

Set 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. If the ground node is involved, the convention is to use zero for this.

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.

NODE CONNECTIONS

Set 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 which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the voltage **Gain**. The resulting current is this number multiplied by the control voltage. Thus it formally has the unit of conductance.

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 named device that must be a two-pin device.

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. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the voltage **Gain** and the **Device** (any two-pin device) name. 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.

NODE CONNECTIONS

Set two **Node names** for the connection nodes for the current source. The first node in a pair represents the positive reference terminal from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this.

DEVICE PARAMETERS

Enter the current **Gain** and the **Device** (any two-pin-device) name. The resulting current is this number multiplied by the control current through the named **Device** (any two-pin device).

Subcircuit Definition

The **Subcircuit Definition** feature is used to define subcircuits. By right-clicking on a subcircuit definition node, you can under that the node add all circuit features available under the main **Electrical Circuit** node except for the subcircuit definition feature itself. By right-clicking on a subcircuit definition node, you can also **Rename** it to something more descriptive than its default name.

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** feature. 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 feature.

Subcircuit Instance

The **Subcircuit Instance** feature is used to refer to defined subcircuits.

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

The **NPN BJT** device model is a large signal model for an NPN Bipolar Junction Transistor. It is an advanced device model and no thorough description and motivation of the many input parameters is attempted here. The interested reader is referred to

Ref. 2 for more details on semiconductor modeling within circuits. 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.

NODE CONNECTIONS

Set three **Node names** for the connection nodes for the **NPN BJT** device. These represent the *collector*, *base* and *emitter* nodes respectively. If the ground node is involved, the convention is to use zero for this.

MODEL PARAMETERS

Set the **Model Parameters**. Reasonable defaults are provided but for any particular BJT, the device manufacturer should be the primary source of information. For an explanation of the **Model Parameters** see NPN Bipolar Transistor.

n-Channel MOSFET

The **n-Channel MOSFET** device model is a large signal model for an n-Channel MOS Transistor. It is an advanced device model and no thorough description and motivation of the many input parameters is attempted here. The interested reader is referred to Ref. 2 for more details on semiconductor modeling within circuits. Many device manufacturers provide model parameters for this MOSFET model. For any particular make of MOSFET, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Set four **Node names** for the connection nodes for the **n-Channel MOSFET** device. These represent the *drain*, *gate*, *source* and *bulk* nodes respectively. If the ground node is involved, the convention is to use zero for this.

MODEL PARAMETERS

Set the **Model Parameters**. Reasonable defaults are provided but for any particular MOSFET, the device manufacturer should be the primary source of information. For an explanation of the **Model Parameters** see n-Channel MOS Transistor.

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 is attempted here. The interested reader is referred to Ref. 2 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.

NODE CONNECTIONS

Set two **Node names** for the positive and negative nodes for the **Diode** device. If the ground node is involved, the convention is to use zero for this.

MODEL PARAMETERS

Set 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 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.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this.

EXTERNAL DEVICE

Enter the source of the **Voltage**. If you have circuit or current excited terminals or circuit ports defined on boundaries or a multi-turn coil domains in other physics interfaces, these display as options in the drop-down list for the **Voltage**. You can also select the **User defined** option and type in your own voltage variable, for example using a suitable coupling operator, see Model Couplings in the *COMSOL Multiphysics User's Guide* (or see Where Do I Access the Documentation and Model Library?). For inductive or electromagnetic wave propagation models, the voltage measurement must be performed as an integral of the electric field as 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. Note that except for when coupling to a circuit terminal or circuit port, you must manually couple back the current flow variable 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 circuits node in the model tree, `IvsUm` is the tag of the **External I vs. U** node in the model tree. The mentioned tags are typically displayed within curly braces `{}` in the model tree.

External U vs. I

The **External U vs. I** feature connects an arbitrary current measurement, for example from another physics interface, as a 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.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the current source. The current flows from the first node to the second node. If the ground node is involved, the convention is to use zero for this.

EXTERNAL DEVICE

Enter the source of the **Current**. Voltage excited terminals or lumped ports defined on boundaries in other interfaces are natural candidates but do not appear as options in the drop-down list for the **Voltage**, as those do not have an accurate built-in current measurement variable. You must select the **User defined** option and type in you own current variable, for example using a suitable coupling operator, see Model Couplings in the *COMSOL Multiphysics User's Guide* (or see Where Do I Access the Documentation and Model Library?). Note that you must manually couple back the voltage variable 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 circuits node in the model tree, `UvsIm` is the tag of the **External U vs. I** node in the model tree. The mentioned tags are typically displayed within curly braces `{}` in the model tree.

External I-Terminal

The **External I-Terminal** feature connects an arbitrary voltage-to-ground measurement, for example a circuit terminal boundary from another 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 feature does not apply when coupling to inductive or

electromagnetic wave propagation models as 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** feature instead.

NODE CONNECTIONS

Set the **Node name** for the connecting node for the voltage assignment.

EXTERNAL TERMINAL

Enter the source of the **Voltage**. If you have circuit or current excited terminals defined on boundaries in other interfaces, these display as options in the drop-down list for the **Voltage**. You can also select the **User defined** option and type in your own voltage variable, for example using a suitable coupling operator, see Model Couplings in the *COMSOL Multiphysics User's Guide* (or see Where Do I Access the Documentation and Model Library?). Note that except for when coupling to a circuit terminal, you must manually couple back the current flow variable 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 circuits node in the model tree, `termIm` is the tag of the **External I-Terminal** node in the model tree. The mentioned tags are typically displayed within curly braces `{}` in the model tree.

SPICE Circuit Import

By right-clicking on the **Electrical Circuit** interface node you can import an existing SPICE netlist by selecting **Import Spice Netlist**. A window opens where you can 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.

Theory For the Electrical Circuit Interface

Introduction

Electrical circuit modeling capabilities are useful when simulating all sorts of electrical and electro-mechanical 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. Either the field model is used to get a better, more accurate description of a single device in the electrical circuit model or 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 is possible to add features representing circuit elements directly to the model 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 you can analyze its behavior 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.

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

SEMICONDUCTOR DEVICE MODELS

There are three more advanced large signal semiconductor device features available in the **Electrical Circuit** interface. Below, the equivalent circuits and the equations defining their non-ideal circuit elements are given. For a more detailed account on semiconductor device modeling, see Ref. 2.

NPN Bipolar Transistor

Figure 5-1 illustrates the equivalent circuit for the bipolar transistor.

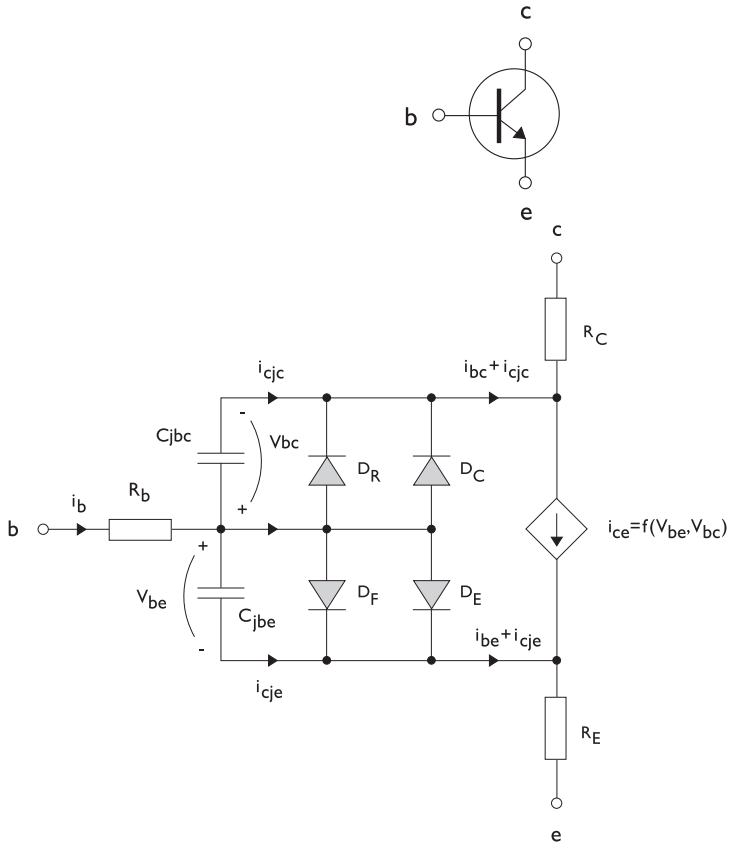


Figure 5-1: A circuit for the bipolar transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$v_{rb} = \frac{1}{A}\left(R_{BM} - \frac{R_B - R_{BM}}{f_{bq}}\right) i_b$$

$$f_{bq} = \frac{1}{2\left(1 - \frac{v_{bc}}{V_{AF}} - \frac{v_{be}}{V_{AR}}\right)}\left(1 + \sqrt{1 + 4I_S\left(\frac{e^{-\frac{v_{be}}{N_F V_T}} - 1}{I_{KF}A} + \frac{e^{-\frac{v_{bc}}{N_R V_T}} - 1}{I_{KR}A}\right)}\right)$$

$$i_{be} = A\left(\frac{I_S}{B_F}\left(e^{-\frac{v_{be}}{N_F V_T}} - 1\right) + I_{SE}\left(e^{-\frac{v_{be}}{N_E V_T}} - 1\right)\right)$$

$$i_{bc} = A\left(\frac{I_S}{B_R}\left(e^{-\frac{v_{bc}}{N_R V_T}} - 1\right) + I_{SC}\left(e^{-\frac{v_{bc}}{N_C V_T}} - 1\right)\right)$$

$$i_{ce} = A\left(\frac{I_S}{f_{bq}}\left(e^{-\frac{v_{be}}{N_F V_T}} + e^{-\frac{v_{bc}}{N_C V_T}}\right)\right)$$

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.

$$C_{jbx} = AC_{Jx} \times \begin{cases} \left(1 - \frac{v_{bx}}{V_{Jx}}\right)^{-M_{Jx}} & v_{bx} < F_C V_{Jx} \\ \left(1 - F_C\right)^{-1 - M_{Jx}}\left(1 - F_C(1 + M_{Jx}) + M_{Jx}\frac{v_{bx}}{V_{Jx}}\right) & v_{bx} \geq F_C V_{Jx} \end{cases}$$

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
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

TABLE 5-1: BIPOLAR TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
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

n-Channel MOS Transistor

Figure 5-1 illustrates an equivalent circuit for the MOS transistor.

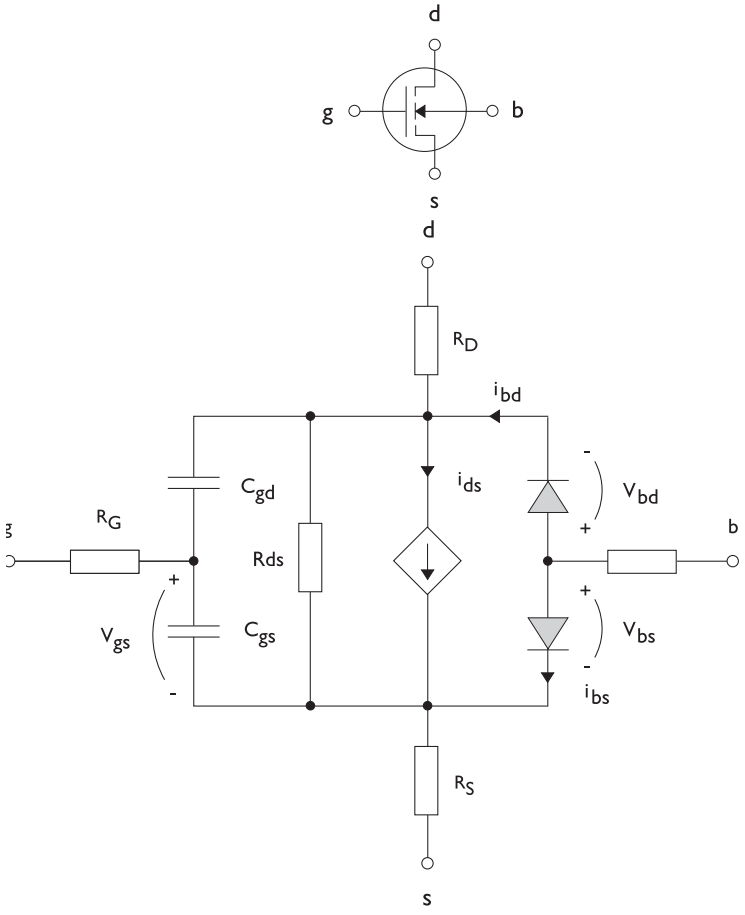


Figure 5-2: A circuit for the MOS transistor.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_{ds} = \begin{cases} \frac{W}{L}\frac{K_P}{2}(1+\Lambda v_{ds})v_{ds}(2v_{th}-v_{ds}) & v_{ds} < v_{th} \\ \frac{W}{L}\frac{K_P}{2}(1+\Lambda v_{ds})v_{th}^2 & v_{ds} \geq v_{th} \\ 0 & v_{ds} < v_{th} \leq 0 \end{cases}$$

$$v_{th} = v_{gs} - (V_{TO} + \Gamma(\sqrt{\Phi - v_{bs}} - \sqrt{\Phi}))$$

$$i_{bd} = I_S\left(e^{-\frac{v_{bd}}{NV_T}} - 1\right)$$

$$i_{bs} = I_S\left(e^{-\frac{v_{bs}}{NV_T}} - 1\right)$$

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 5-1: MOS 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
N	1	Bulk junction ideality factor
P_B	0.75 V	Bulk junction potential

TABLE 5-1: MOS TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
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)	0 $V^{0.5}$	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.

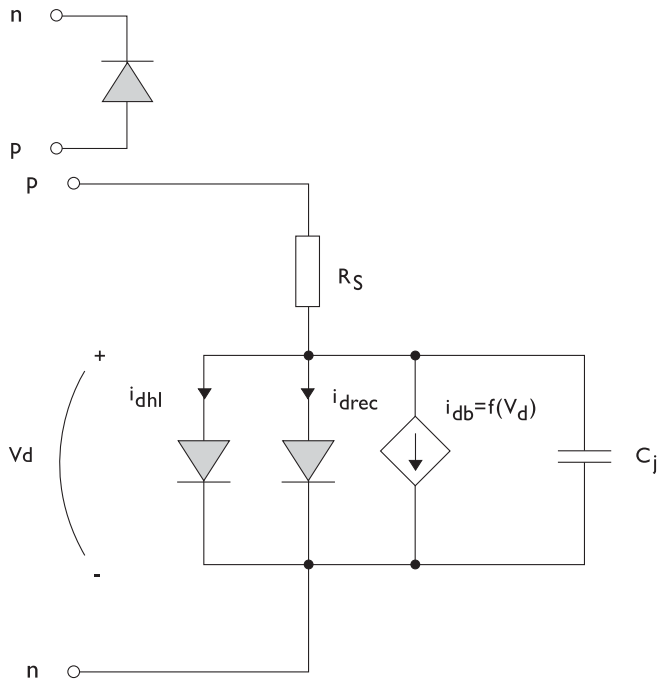


Figure 5-3: A circuit for the diode.

The following equations are used to compute the relations between currents and voltages in the circuit.

$$i_d = i_{dhl} + i_{drec} + i_{db} + i_c$$

$$i_{dhl} = I_S\left(e^{-\frac{v_d}{NV_T}} - 1\right)\frac{1}{\sqrt{1 + \frac{I_S}{I_{KF}}\left(e^{-\frac{v_d}{NV_T}} - 1\right)}}$$

$$i_{drec} = I_{SR}\left(e^{-\frac{v_d}{N_R V_T}} - 1\right)$$

$$i_{db} = I_{BV}e^{-\frac{v_d + B_V}{N_{BV}V_T}}$$

$$C_j = C_{J0} \times \begin{cases} \left(1 - \frac{v_d}{V_J}\right)^{-M} & v_d < F_C V_J \\ (1 - F_C)^{-1-M}\left(1 - F_C(1+M) + M\frac{v_d}{V_J}\right) & v_d \geq F_C V_J \end{cases}$$

where we need the following model parameters

TABLE 5-1: 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}	1e-09 A	Current at breakdown voltage
I_{KF}	Inf (A)	Corner for high-current roll-off
I_S	1e-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

Example Model Using The Electrical Circuit Interface

There is one model in the model library that uses the **Electrical Circuit** interface via SPICE netlist import, the AC/DC Module Model Library model Inductor in an Amplifier Circuit in the Electrical Components section.

REFERENCES

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

2. P. Antognetti and G. Massobrio, Semiconductor Device Modeling with Spice, 2nd Ed. McGraw-Hill, Inc., 1993.

6

The Heat Transfer Interface

This section the **Microwave Heating** multiphysics interface, which combines the features of an **Electromagnetic Waves** interface from the RF Module with those of the **Heat Transfer** interface.

In this section:

Location of Other Heat Transfer Documentation

Heat transfer through conduction and convection (both non-isothermal flow and conjugate heat transfer) in solid and free media is supported by physics interfaces shipped with the basic COMSOL Multiphysics license. The **Heat Transfer** interface is described in the *COMSOL Multiphysics User's Guide*. See The Heat Transfer Interface, The Joule Heating Interface and Theory for the Heat Transfer Interfaces for more information (or see Where Do I Access the Documentation and Model Library?).

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under **Heat Transfer Module** in the documentation tree.

The Microwave Heating Interface

Note: Also refer to the *COMSOL Multiphysics User's Guide* for detailed information about selecting geometric entities (domains, boundaries, edges, and points) (or see Where Do I Access the Documentation and Model Library?).

The **Microwave Heating** multiphysics interface ( ) combines the features of an **Electromagnetic Waves** interface from the RF Module with those of the **Heat Transfer** interface. The predefined interaction adds the electromagnetic losses from the electromagnetic waves as a heat source. This interface is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale (adiabatic assumption). Thus, it is associated with two predefined study types:

- **Frequency-Stationary**
  - Time-harmonic electromagnetic waves
  - Stationary heat transfer
- **Frequency-Transient**
  - Time-harmonic electromagnetic waves
  - Transient heat transfer

The following default nodes are added when using this interface—**Microwave Heating Model**, **Electromagnetic Heat Source**, **Thermal Insulation**, **Perfect Electric Conductor**, and **Initial Values** nodes.

Right-click the **Microwave Heating** node to add other features that implement, for example, boundary conditions and volume forces.

INTERFACE IDENTIFIER

This is the physics interface identifier, which you use to reach the fields and variables in expressions, for example.

The identifier appears in the **Identifier** edit field, and you can change it to any unique string that is a valid identifier. The default identifier (for the first interface in the model) is `mh`.

DOMAINS

Select the domains where you want the interface to apply. The default setting is to include all domains in the model.

SURFACE-TO-SURFACE RADIATION

This section requires the **Heat Transfer Module**.

Select the **Surface-to-surface radiation** check box to enable the **Radiation Settings** section.

RADIATION SETTINGS

To display this section select the **Surface-to-surface radiation** check box. Select a **Surface-to-Surface radiation method**—**Hemicube** (the default) or **Direct area integration**.

Hemicube

The more sophisticated and general hemicube method uses a z-buffered projection on the sides of a hemicube (with generalizations to 2D and 1D) to account for shadowing effects. You can think of the method as rendering digital images of the geometry in five different directions (in 3D; in 2D only three directions are needed), and counting the pixels in each mesh element to evaluate its view factor.

Influence its accuracy by setting the **Radiation resolution** of the virtual snapshots. The number of z-buffer pixels on each side of the 3D hemicube equals the specified **Radiation resolution** squared. Thus the time required to evaluate the irradiation increases quadratically with resolution. In 2D, the number of z-buffer pixels is proportional to the resolution property, and thus the time is, as well.

For an axisymmetric geometry, G_{m} and F_{amb} must be evaluated in a corresponding 3D geometry obtained by revolving the 2D boundaries about the axis. COMSOL Multiphysics creates this virtual 3D geometry by revolving the 2D boundary mesh into a 3D mesh. You control the resolution in the azimuthal direction by setting the number of **azimuthal sectors**, which is the same as the number of elements to a full revolution. Try to balance this number against the mesh resolution in the rz-plane.

Direct Area Integration

COMSOL Multiphysics evaluates the integrals directly, without considering which face elements are obstructed by others. This means that shadowing effects (that is, surface elements being obstructed in nonconvex cases) are not taken into account. Elements facing away from each other are, however, excluded from the integrals.

Direct area integration is fast and accurate for simple geometries with no shadowing, or where the shadowing can be handled by manually assigning boundaries to different

groups. Note that if shadowing is ignored, global energy is not conserved. You can control the accuracy by specifying an **Radiation integration order**. Sharp angles and small gaps between surfaces may require a higher integration order for accuracy but also more time to evaluate the irradiation.

Use Radiation Groups

To speed up the radiation calculations in many cases, select the **Use radiation groups** check box. This adds a **Radiation Group** node to the **Model Builder**, where the selection of boundaries can be made.

SETTINGS

Select whether to **Solve for** the **Full field** or the **Scattered field**.

If **Scattered field** is selected, enter expressions for the **Background electric field** $\boldsymbol{E}_\mathrm{b}$ (SI unit: V/m). The defaults are 0.

PORT SWEEP SETTINGS

To invoke a parametric sweep over the ports/terminals in addition to the automatically generated frequency sweep, select the **Activate port sweep** check box. The generated lumped parameters are in the form of an impedance or admittance matrix depending on the port/terminal settings which consistently must be of either fixed voltage or fixed current type.

Enter a **Port parameter name**. A specific name is assigned to the variable that controls the port number solved for during the sweep.

The lumped parameters are subject to **Touchstone file export**. Enter or **Browse** for a file name and path in the **Touchstone file export** field.

Select an **Output format**—**Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

Enter a **Reference impedance** Z_ref (SI unit: Ω). The default is 50 Ω .

DEPENDENT VARIABLES

This interface defines these dependent variables (fields), **Temperature** T, the **Surface radiosity** J and the **Electric field** $relE$. You can change the name but the names of fields and dependent variables must be unique within a model.

ADVANCED SETTINGS AND DISCRETIZATION

Normally these settings do not need to be changed. See Show More Options: Advanced Settings and Discretization in the *COMSOL Multiphysics User's Guide* (or see Where Do I Access the Documentation and Model Library?).

INCONSISTENT AND CONSISTENT STABILIZATION

To display this section, select **Show More Options** from the **View** menu in the **Model Builder**. See Show More Options: Consistent and Inconsistent Stabilization in the *COMSOL Multiphysics User's Guide* for information about these settings.

Microwave Heating Model

The **Microwave Heating Model** feature has settings to define the **Displacement Field**, **Magnetic Field**, **Conduction Current**, **Heat Conduction**, and **Thermodynamics**.

DOMAINS

Select the domains where you want to apply the model. The default setting is to include all domains in the model.

MODEL INPUTS

In this section you can, for example, define the temperature field to use for a temperature-dependent material property. It is initially empty.

DISPLACEMENT FIELD

Select the **Displacement field model**. The options are:

- **Relative permittivity** - to specify the relative permittivity or take it from the material.
- **Refractive index** - to specify the real and imaginary parts of the refractive index or take them from the material. Note that this assumes a relative permeability of unity and zero conductivity. 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** - to specify a loss tangent for dielectric losses or take it from the material. Note that this assumes zero conductivity.
- **Dielectric loss** - to specify the real and imaginary parts of the relative permittivity or take them from the material. 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.

MAGNETIC FIELD

Select the **Magnetic field model**. The options are:

- **Relative permeability** - to specify the relative permittivity or take it from the material.
- **Magnetic losses** - to specify the real and imaginary parts of the relative permeability or take them from the material. 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.

CONDUCTION CURRENT

Select the **Electric conductivity**. The options are:

- **Conductivity** from the material.
- **Linearized resistivity** specifies a linear temperature dependence that can be taken from the material or be user defined.
- **User defined** conductivity specified by the user. Here optionally anisotropic conductivity can be defined.

HEAT CONDUCTION

The default uses the **Thermal conductivity** k (SI unit: W/(m·K)) **From material**. Select **User-defined** to enter another value. Select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** depending on the characteristics of the thermal conductivity.

THERMODYNAMICS

The default uses the **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)) and **Density** ρ (SI unit: kg/m^3) values **From material**. Select **User-defined** to enter other values or expressions for one or both variables.

Electromagnetic Heat Source

The **Electromagnetic Heat Source** feature maps the electromagnetic losses as a heat source in the heat transfer part of the model. It appears as a default feature.

DOMAINS

Select the domains where you want to apply the model. The default feature setting is hard-coded to include all domains in the model.

Initial Values

The **Initial Values** feature adds initial values for the temperature, surface radiosity and electric field.

DOMAINS

Select the domains where you want to apply the initial values. The default setting is to include all domains in the model.

INITIAL VALUES

Enter values or expressions for the **Temperature** T (SI unit: K), **Surface radiosity** J (SI unit W/m^2) and **Electric field** E (SI unit: V/m). The default temperature is 293.15 K.

Shared Features

The **Microwave Heating** interface shares most of its settings windows with the **Electromagnetic Waves** and the **Heat Transfer** interfaces. For more information on electromagnetic waves, see The Electromagnetic Waves Interface.

Location of Other Heat Transfer Documentation

The **Heat Transfer in Solids, Heat Transfer in Fluids** (general convection and conduction, non-isothermal flow, and conjugate heat transfer), and **Joule Heating** interfaces all belong to the COMSOL Multiphysics base package. See The Heat Transfer Interface, The Joule Heating Interface and Theory for the Heat Transfer Interfaces in the *COMSOL Multiphysics User's Guide* for more information (or see Where Do I Access the Documentation and Model Library?).

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

7

Glossary

This glossary 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, please see the glossary in the *COMSOL Multiphysics User's Guide*. For references to more information about a term, see the index.

Glossary of Terms

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 $\mathbf{D}$ and $\mathbf{E}$ fields and between the $\mathbf{B}$ and $\mathbf{H}$ 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.

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.

Nedelec's edge element See *vector element*.

perfect electric conductor A material with high electric 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.

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*.

INDEX