COMSOL
MULTIPHYSICS

# RF MODULE

**VERSION 3.5a**

COMSOL

*RF Module User's Guide*
© COPYRIGHT 1998–2008 by COMSOL AB. All rights reserved

Patent pending

Version:          November 2008      COMSOL 3.5a

Part number: CM021001

# C O N T E N T S

# Chapter 3: Review of Electromagnetics

# Chapter 4: The Application Modes

# Chapter 5: Glossary

# Introduction

$T$he RF Module 3.5a is an optional package that extends the COMSOL Multiphysics® modeling environment with customized user interfaces and functionality optimized for the analysis of electromagnetic waves. Like all modules in the COMSOL Multiphysics family it provides a library of prewritten ready-to-run models that make it quicker and easier to analyze discipline-specific problems.

This particular module solves problems in the general field of electromagnetic waves, such as RF and microwave applications, optics, and photonics. The application modes (modeling interfaces) included here are fully multiphysics enabled, making it possible to couple them to any other physics application mode in COMSOL Multiphysics or the other modules. For example, to analyze stress-optical effects in a waveguide, you would first do a plane strain analysis using the Structural Mechanics Module followed by an optical mode analysis that shows the resulting split of the fundamental modes.

The underlying equations for electromagnetics are automatically available in all of the application modes—a feature unique to COMSOL Multiphysics. This also makes nonstandard modeling easily accessible.

The documentation set for the RF Module consists of three books. The one you are reading, the *RF Module User's Guide*, introduces you to the basic functionality in the module, reviews new features in the version 3.5a release, reviews basic modeling techniques, and includes reference material of interest to those working in electromagnetics. The second book in the set, the *RF Module Model Library*, contains a large number of ready-to-run models that illustrate real-world uses of the module. Each model comes with an introduction covering basic theory, the modeling purpose, and a discussion about the results, as well as step-by-step instructions that illustrate how to set it up. Further, we supply these models as COMSOL Multiphysics Model MPH-files so you can import them into COMSOL Multiphysics for immediate execution. This way you can follow along with the printed discussion as well as use them as a jumping-off point for your own modeling needs. A third book, the *RF Module Reference Guide*, contains reference material about application-mode implementations and command-line functions and programming.

---

**Note:** The full documentation set is available in electronic formats—PDF and HTML—through the COMSOL Help Desk after installation.

---

*Typographical Conventions*

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

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

- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user interface during geometry modeling. However, nowhere on the screen will you see

the term "Draw" referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).

- The symbol **>** indicates a menu item or an item in a folder in the **Model Navigator**. For example, **Physics>Equation System>Subdomain Settings** is equivalent to: On the **Physics** menu, point to **Equation System** and then click **Subdomain Settings**. **COMSOL Multiphysics>Heat Transfer>Conduction** means: Open the **COMSOL Multiphysics** folder, open the **Heat Transfer** folder, and select **Conduction**.

- A `Code` (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as "Type `1.25` in the **Current density** edit field." The monospace font also indicates code.

- 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 RF Module

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

## What Can the RF Module Do?

The RF Module contains a set of application modes 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 application modes. This forms a powerful *multiphysics* model that solves all the equations simultaneously.

You can transform any model developed with the RF Module into a model described by the underlying partial differential equations. This offers a unique way to see the underlying physical laws of a simulation.

COMSOL Multiphysics also provides an interface to the MATLAB technical computing environment. If you have a MATLAB license, you can export a simulation to MATLAB or save it as a Model M-file—a script file that runs in MATLAB. This makes it possible to incorporate models with other products in the MATLAB family such as Simulink and the Control System Toolbox.

## What Problems Can You Solve?

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

- Propagation of electromagnetic waves
  - Harmonic analysis
  - Transient analysis
  - Eigenfrequency/eigenmode analysis
- Propagation of light
  - Harmonic analysis
  - Transient analysis
  - Eigenfrequency/eigenmode analysis

All categories are available in both 2D and 3D. In 2D the package offers in-plane application modes for problems with a planar symmetry as well as axisymmetric application modes for problems with a cylindrical symmetry. It further provides four application modes for 3D harmonic wave propagation in one direction, which can be formulated as 2D equations.

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 the currents and charges generating the electromagnetic fields vary so slowly in time that the electromagnetic fields are practically the same at every instant as if they had been generated by stationary sources. For electrostatic, magnetostatic, and quasi-static electromagnetics, use the AC/DC Module, a COMSOL Multiphysics add-on module for low-frequency electromagnetics.

When the variations in time of the sources of the electromagnetic fields are more rapid, it is necessary to use the full Maxwell application modes 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 application modes.

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. For a more detailed

description of some of these applications, refer to the matching book that comes with this product, the *RF Module Model Library.*

## *New Features in the RF Module 3.5a*

RF Module 3.5a adds a number of valuable new capabilities including these new features:

- ECAD interface, ODB++ files: Import of PCB designs made in Cadence Allegro and Mentor Graphics Board Station and other software that supports the XML version of Valor's ODB++ format. See "ECAD Import" on page 71 for more information.

- ECAD interface, Artwork/NETEX-G software and Gerber/drill files: Import of PCB designs from any ECAD software package through the use of the software NETEX-G from Artwork and generic Gerber and drill file formats. See "ECAD Import" on page 71 for more information.

- SPICE circuit support. See "SPICE Circuit Import" on page 85 for more information.

- Circuit ports for wave applications, modeling the connection of a transmission line or an antenna to an external circuit. See "Circuit Ports" on page 88 for more information.

- Improved axisymmetric formulation

- New models:
  - PCB microwave filter with stress
  - Conical antenna with circuit
  - Shape optimization of a dipole antenna (requires Optimization Lab)
  - Schumann resonance
  - Wave guide optimization (requires Optimization Lab)

# Application Mode Summary

An application mode in COMSOL Multiphysics is a specification of the equations and the set of dependent variables you want to solve for and an interface adapted to the application (do not confuse the COMSOL Multiphysics application mode with the mode analysis, which is an analysis type for confined waves). When you have selected the application mode, you can also choose an analysis type. However, you can also change this later in the COMSOL Multiphysics user interface. The available analysis types are time-harmonic, time-dependent (transient), eigenfrequency, and mode analysis. Below you first find a short introduction to the field variables (dependent variables) in some of the 2D application modes. Following that is a section with some general details about the two analysis types in time-dependent problems. Finally, there is a summary with a short description of all the application modes in the RF Module.

## *Field Variables in 2D*

When you want to solve for a vector field in 2D you usually get two different cases. For electromagnetic waves the cases are in-plane waves and perpendicular waves (only in-plane waves exists for axial symmetry). "In-plane" means that the current flows or the wave travels parallel to the cross section.

In wave propagation there are application modes for transverse electric (TE), transverse magnetic (TM), and hybrid-mode waves. For in-plane propagation, the wave propagates parallel to the 2D cross section. The plane with a constant phase is then perpendicular to the cross section. The analysis types are eigenfrequency analysis, harmonic propagation (time-harmonic analysis), and time-dependent analysis. Models using axial symmetry have the same application modes and analysis types. A typical example of axisymmetric wave propagation is the coaxial cable, where the wave propagates along the cable in the $z$ direction. In the perpendicular application mode, the waves propagate perpendicular to the cross section. So viewing the 2D cross section, you look in the direction of the propagating wave. The analysis types are mode analysis and eigenfrequency analysis. In mode analysis you fix the frequency and calculate the possible propagation constants. In eigenfrequency analysis, it is the other way around. There is no azimuthal wave case because waves cannot travel around an axis.

## Time-Dependent and Time-Harmonic Analysis

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. 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 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 predefined scalar variable. This approach is useful because, combined with Fourier analysis, it applies to all periodic signals with the exception of nonlinear problems. Examples of typical harmonic simulations are wave-propagation problems like waveguides and antennas (see "H-Bend Waveguide with S-parameters" on page 111 in the *RF Module Model Library*).

For nonlinear problems you can use a time-harmonic analysis after a *linearization* of the problem, which assumes that the distortion of the sinusoidal signal is small. See "Distributed SPICE Model of an Integrated Bipolar Transistor" on page 456 in the *COMSOL Multiphysics Model Library*.

You need to specify a time-dependent analysis when you think that 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 analysis if you have a periodic input with many harmonics, like a square-shaped signal.

## Application Modes

Each of the mode descriptions below has a reference to the page where you can find a more detailed description. You can also look at Table 4-1 on page 131, which provides a summary of all application modes with dependent variables and references to more detailed information.

**ELECTROMAGNETIC WAVES**

For wave problems in 3D, there are two application modes:

• Electromagnetic Waves

This application mode supports harmonic propagation (time-harmonic analysis), time-dependent analysis and eigenfrequency analysis. See "3D Electromagnetic Waves Application Mode" on page 137.

• Boundary Mode Analysis

The simulation is a 2D problem defined on a planar face in the 3D geometry. The mode analysis is the only available analysis type. See "Boundary Mode Analysis Application Mode in 3D" on page 169.

The 2D application modes were discussed in the section "Field Variables in 2D". Only the references to the detailed descriptions for each application mode are given here.

• Perpendicular Waves

See "Perpendicular Waves Application Mode" on page 160.

• In-Plane Waves

See "In-Plane Waves Application Mode" on page 148 or "Axisymmetric Waves Application Mode" on page 158.

• Boundary Mode Analysis

The simulation is a 1D problem defined on a straight line in the 2D geometry. The mode analysis is the only available analysis type. See "Boundary Mode Analysis Application Mode in 2D" on page 172.

# 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 chapter 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.

# Model Descriptions

*Formats for the Model Descriptions*

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

### MODEL NAVIGATOR

The **Model Navigator** appears when you start COMSOL Multiphysics or when you restart completely within COMSOL Multiphysics by selecting **New** from the **File** menu or by clicking the **New** toolbar button. On the **New** page in the **Model Navigator** you specify the application mode, names of dependent variables, and the analysis type: static, time-harmonic, transient, mode analysis, or eigenfrequency. You can also set up a combination of application modes from the RF Module, COMSOL Multiphysics, or any other available module. See the section "Creating and Opening Models" on page 22 in the *COMSOL Multiphysics Quick Start* for more information about the Model Navigator.

### OPTIONS AND SETTINGS

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

### GEOMETRY MODELING

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

### BOUNDARY CONDITIONS

You specify the boundary conditions for a model in the **Boundary Settings** dialog box. For details, see "Specifying Boundary Conditions" on page 246 in the *COMSOL Multiphysics User's Guide*. Valid boundary conditions for each electromagnetics mode are summarized in "The Application Mode Formulations" on page 130 of this manual.

### SUBDOMAIN SETTINGS

You specify equation parameters in the **Subdomain Settings** dialog box. For details see "Specifying Subdomain Settings and PDE Coefficients" on page 209 in the *COMSOL Multiphysics User's Guide*. The physical parameters of specific interest for electromagnetics modeling are summarized in "The Application Mode Formulations" on page 130 of this manual, where you can also learn about the derivation of the equations as well as the boundary conditions.

### SCALAR VARIABLES

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

### MESH GENERATION

The program must mesh the geometry before it can solve the problem. Sometimes it is sufficient to click the **Initialize Mesh** button on the Main toolbar. In other cases you need to adjust settings in the **Free Mesh Parameters** dialog box and the other mesh-generation tools on the **Mesh** menu. Read more about meshing in "Creating Meshes" on page 300 of the *COMSOL Multiphysics User's Guide*.

### COMPUTING THE SOLUTION

To solve a problem, for most cases simply click the **Solve** button on the Main toolbar. In other cases it might be necessary to adjust the solver properties, which you do in the **Solver Parameters** dialog box. For details see "Selecting a Solver" on page 378 of the *COMSOL Multiphysics User's Guide*.

### POSTPROCESSING AND VISUALIZATION

The powerful visualization of COMSOL Multiphysics tools are accessible in the program's Postprocessing mode, but to use them you must be familiar with the **Plot Parameters** dialog box and the other postprocessing tools on the **Postprocessing** menu. See "Postprocessing Results" on page 456 in the *COMSOL Multiphysics User's Guide* for details.

**ADDITIONAL POSTPROCESSING**

For further postprocessing calculations, you can export the solution to MATLAB. Details of modeling by programming are available in the section "The Programming Language" on page 28 of the *RF Module Reference Guide* and in the *COMSOL Multiphysics MATLAB Interface Guide*.

# Preparing for Modeling

This section is intended to guide you through the selection process among the application modes 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 will get help in answering questions like:

- Which spatial dimension should I use: 2D, 3D, or 2D axial symmetry?
- Is my problem suited for time-dependent or time-harmonic 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 application mode 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 application modes, the analysis 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 PROBLEMS

The text below guides you through some of the common approximations made for 2D problems. 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 with S-parameters" on page 111 in the *RF Module Model Library*, where the **E** 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.
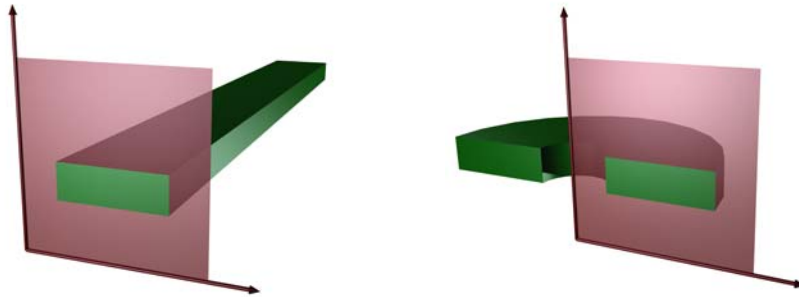


*Figure 2-1: The cross sections and their real geometry for Cartesian coordinates and cylindrical coordinates (axial symmetry).*

### Axial Symmetry (Cylindrical Coordinates)

If you can construct the 3D geometry by revolving a cross section around an axis, and no variations in any variable occur when going around the axis of revolution, you can use an axisymmetric application mode. 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 $\alpha r$, where $\alpha$ is the revolution angle (for example, $2\pi$ for a full turn). See the model "Monoconical RF Antenna" on page 65 in the *RF Module Model Library* for an example.

### 3D PROBLEMS

Although COMSOL Multiphysics fully supports arbitrary 3D geometries, it is important to simplify the problem. This is because 3D problems easily get large and require more computer power, memory, and time to solve. The extra time you spend on simplifying your problem 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 will be more accurate in 2D, because you can use a much denser mesh. If you find this applicable, take a look at the section "2D Problems".

- 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 application modes.

• 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 "Monoconical RF Antenna" on page 65 in 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 "Dielectric Scattering PML" on page 8 in the *RF Module 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, 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 ("Creating Boundary Layer Meshes Interactively" on page 359 in the *COMSOL Multiphysics User's Guide*).

- 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 may be shorter in a dielectric medium.

**SOLVERS**

You can, in most cases, use the solver that COMSOL Multiphysics suggests. The choice of solver is optimized for the typical case for each application mode and analysis 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 use a large amount of memory. For extremely large 3D problems, you might need a 64-bit platform. You can find a more detailed description on the solver settings in "Solving the Model" on page 377 in the *COMSOL Multiphysics User's Guide*. See also "Solving Large 3D Problems" on page 92.

*Hermitian Transpose*

The **Use Hermitian transpose of constraint matrix and in symmetry detection** setting on the **Advanced** page controls the handling of complex numbers in constraints to the equations. One case in the RF Module when complex numbers appear in a constraint is when using a periodic boundary condition that includes a complex phase factor, also known as a Floquet periodicity. In this case, select the **Use Hermitian transpose of constraint matrix and in symmetry detection** check box.

# An Example—H-Bend Waveguide

## *Introduction*

This examples shows how to model a rectangular waveguide for microwaves. A single hollow waveguide can conduct two kinds of electromagnetic waves: transversal magnetic (TM) or transversal electric (TE) waves. This model examines a TE wave, one that has no electric field component in the direction of propagation. More specifically, for this model you select the frequency and waveguide dimension so that $TE_{10}$ is the single propagating mode. In that mode the electric field has only one nonzero component—a sinusoidal with two nodes, one at each of the walls of the waveguide.

One important design aspect is how to shape a waveguide to go around a corner without incurring unnecessary losses in signal power. Unlike in wires, these losses usually do not result from ohmic resistance but instead arise from unwanted reflections. You can minimize these reflections by keeping the bend smooth with a large enough radius. In the range of operation the transmission characteristics (the ability of the waveguide to transmit the signal) must be reasonably uniform for avoiding signal distortions.

## *Model Definition*

This example illustrates how to create a model that computes the electromagnetic fields and transmission characteristics of a 90° bend for a given radius. This type of waveguide bends changes the direction of the **H** field components and leaves the direction of the **E** field unchanged. The waveguide is therefore called an *H-bend*. The H-bend design used in this example is well-proven in real-world applications and you can buy similar waveguide bends online from a number of manufacturers. This particular bend performs optimally in the ideal case of perfectly conducting walls as is shown later on in this model by computing the (in this case optimal) transmission characteristics.

The waveguide walls are typically plated with a very good conductor, such as silver. In this example the walls are considered to be made of a perfect conductor, implying that $\mathbf{n} \times \mathbf{E} = 0$ on the boundaries. This boundary condition is referred to as a *perfect electric conductor* (PEC) boundary condition.

The geometry is as follows, as viewed in the *xy*-plane.
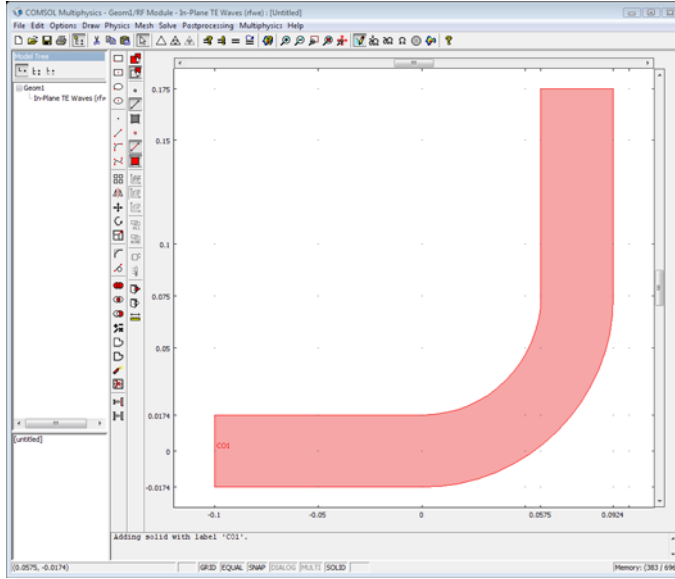


**DOMAIN EQUATIONS**

The waveguide is considered to continue indefinitely before and after the bend. This means that the input wave needs to have the form of a wave that has been traveling through a straight waveguide. The shape of such a wave is determined by the boundary conditions of Maxwell's equations on the sides of the metallic boundaries, that is, the PEC boundary condition. If polarized according to a $TE_{10}$ mode, the shape is known analytically to be $\mathbf{E} = (0, 0, \sin(\pi(a - y)/(2a)))\cos(\omega t)$ given that the entrance boundary is centered around the $y = 0$ axis, and that the width of the waveguide, in the $y$ direction, is $2a$.

In the RF Module you can model this waveguide in the 2D In-Plane TE Waves application mode or in the 3D Electromagnetic Waves application mode as a time-harmonic wave propagation model. This means that only the phasor component of the field is modeled. The incident field then has the form $\mathbf{E} = (0, 0, E_{0z}) = (0, 0, \sin(\pi(a - y)/(2a)))$, and is considered as part of the expression

$\mathbf{E}(t) = \text{Re}\{(0, 0, \sin(\pi(a-y)/(2a))e^{j\omega t})\} = \text{Re}\{\mathbf{E}e^{j\omega t}\}$, where complex-valued arithmetic has been used (also referred to as the $j\omega$ method).



$E_z = \cos((\pi/2)(y/a))$
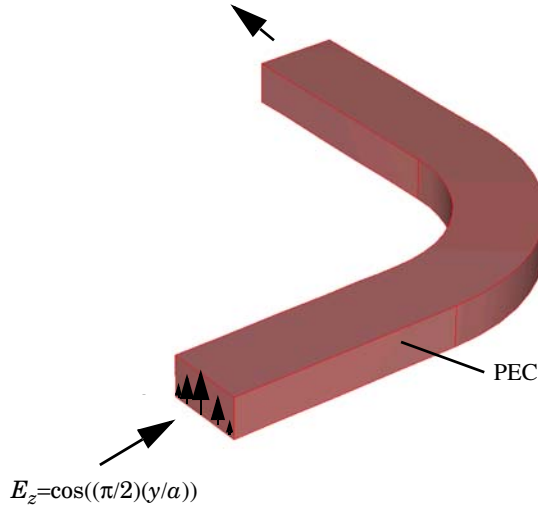
The width of the waveguide is chosen so that it has a cutoff frequency of 4.3 GHz. This makes the waveguide operational between 5.4 GHz up to 8.1 GHz. At higher frequencies other modes than the $TE_{10}$ appear, causing a "dirty" signal. The input wave then splits into several modes that are hard to control without having large power losses. Below the cutoff frequency, no waves can propagate through the waveguide. This is an intrinsic property of microwave waveguides.

The cutoff frequency of different modes in a straight waveguide is given by the relation

$$(\nu_c)_{mn} = \frac{c}{2}\sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where $m$ and $n$ are the mode numbers ($m = 1, n = 0$ for the $TE_{10}$ mode), $a$ and $b$ are the lengths of the sides of the waveguide cross-section, and $c$ is the speed of light.

For this waveguide, $a = 2b$ and $b = 0.0174245$.

The first few cutoff frequencies are $(\nu_c)_{10} = 4.3$ GHz, $(\nu_c)_{01} = 8.6$ GHz, $(\nu_c)_{11} = 9.6$ GHz and the operational range is chosen to be $1.25(\nu_c)_{10} = 5.4$ GHz to $0.95(\nu_c)_{10} = 8.1$ GHz. This is to have reasonable margins for manufacturing errors and to avoid the large reflections that occur at lower frequencies.

## BOUNDARY CONDITIONS

This model makes use of the predefined port boundary condition. It is an automated version of the matched boundary condition described later in this section. An additional advantage is that the port boundary condition automatically creates postprocessing variables for the S-parameters.

The input matched boundary condition consists of two parts: an incident planar wave and an absorbing boundary condition. The matched boundary condition is also used at the output boundaries to eliminate any reflections there. At the output boundaries there is no excitation. The walls of the waveguide are considered to be good conductors, so you can use the perfectly electric conductive (PEC) boundary condition.

For specifying the absorbing boundary condition you must know the propagation constant, $\beta$, of the wave. You can find the propagation constant from an eigenmode analysis of the waveguide cross section.

In this simple case, however, you can also compute the propagation constant by hand using the relation

$$k_0^2 = k_x^2 + k_y^2 + k_z^2$$

for the wavenumbers in the $x$, $y$, and $z$ directions, respectively, at the waveguide entrance port. Here $x$ is the direction of propagation, and $y$ and $z$ are the transversal directions, with $z$ as the out-of-plane direction. In an infinitely extended straight waveguide, the following equations define the free-space and the $x$, $y$, and $z$ direction wavelengths:

$$k_0 = \frac{2\pi}{\lambda_0}$$

$$k_x = \frac{2\pi}{\lambda_x} = \beta$$

$$k_y = \frac{2\pi}{\lambda_y}$$

$$k_z = \frac{2\pi}{\lambda_z}$$

In this case, the $y$ direction wavelength is known to be $2a$ because the model describes the lowest propagating mode, which is a half wavelength across the transversal direction. This means for the wave number in the $y$ direction

$$k_y = \frac{2\pi}{2a}$$

Also, for the $TE_{10}$ mode,

$$k_z = 0$$

The free-space wave number is

$$k_0 = \frac{2\pi}{\left(\frac{c_0}{\nu}\right)} = \frac{2\pi}{\left(\frac{3 \cdot 10^8}{\nu}\right)}$$

The propagation constant evaluates to

$$\beta = k_x = \sqrt{k_0^2 - k_y^2} = \sqrt{\left(\frac{2\pi}{\left(\frac{3 \cdot 10^8}{\nu}\right)}\right)^2 - \left(\frac{2\pi}{2a}\right)^2},$$

which is a frequency-dependent expression.

Now, for the matched boundary condition, entering the correct propagation constant eliminates all waves with that wavenumber in the propagating direction. To make the boundary condition perfectly absorbing, the propagating direction needs to be the same as the normal direction of the output boundary.

---

**Note:** The input amplitude can be replaced by any analytical expression. You can, for instance, create a wave corresponding to a truncated Gaussian beam or simply a planar wave, for which the expression would be 1.

---

The first part of the analysis is made for a frequency that is 20% above the cutoff frequency. This is to show a generic propagating wave within the frequency range of operation.

## Results and Discussion

The wave is found to propagate through the bend with very small reflection.



Slice: Electric field, z component [V/m]
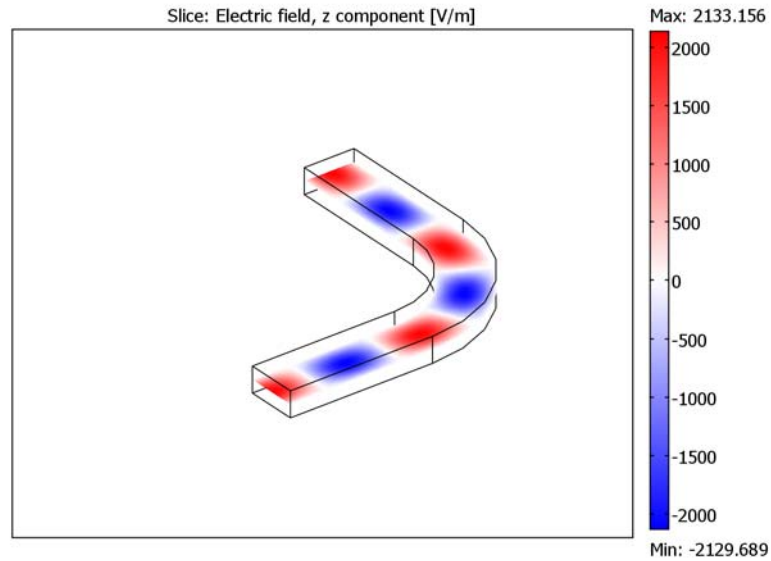
To verify that there is almost no reflection you can compare the power flow of the outgoing wave to that of the incoming wave. The entrance port is excited using the port boundary condition, which automatically normalizes the excitation to unit power (1 W). Comparing this to the power flow over the output port, 1.006 W, shows that the power loss is less than 1%.

For the 2D version of the model the power flow is 0.99993 W, indicating that any reflected power is too small to be detected. The 2D model is more accurate than the 3D model because it uses a finer mesh that better resolves the waves.

**Model Library path:** RF_Module/RF_and_Microwave_Engineering/
waveguide_hbend_3d

**MODEL NAVIGATOR**

**1** In the **Model Navigator**, select **3D** in the **Space dimension** list.

**2** In the list of application modes, select **RF Module>Electromagnetic Waves>Harmonic propagation**.

**3** Click **OK**.

**OPTIONS AND SETTINGS**

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

**2** Define the following constants in the **Constants** dialog box (the descriptions are optional); when done, click **OK**.
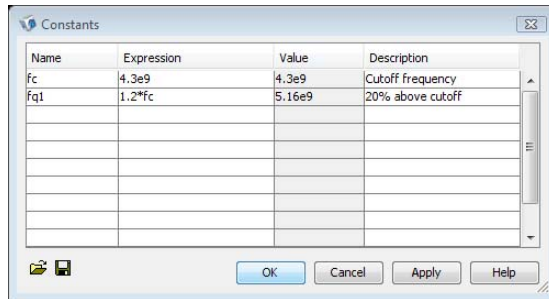


| NAME | EXPRESSION | DESCRIPTION |
|------|-----------|-------------|
| fc   | 4.3e9     | Cutoff frequency |
| fq1  | 1.2*fc    | 20% above cutoff |

**GEOMETRY MODELING**

**1** From the **Draw** menu, open the **Work-Plane Settings** dialog box. Click **OK** to obtain the default work plane in the *xy*-plane.

**2** From the **Options** menu, choose **Axes/Grid Settings**.

**3** In the **Axes/Grid Settings** dialog box, specify the following settings; when done, click
   **OK**.

| AXIS | | GRID | |
|---|---|---|---|
| x min | -0.175 | x spacing | 0.05 |
| x max | 0.175 | Extra x | 0.075-0.0174245 0.075+0.0174245 |
| y min | -0.04 | y spacing | 0.05 |
| y max | 0.2 | Extra y | -0.0174245 0.0174245 0.075 0.175 |

   To define the grid spacings, first click the **Grid** tab and clear the **Auto** check box.

   The waveguide width 2*0.0174245 corresponds to a cutoff frequency of 4.3 GHz.

**4** Start by drawing three lines. After selecting the **Line** button on the Draw toolbar,
   click at (0, –0.0174), (–0.1, –0.0174), (–0.1, +0.0174), and (0, +0.0174).

**5** Click the **2nd Degree Bézier Curve** button, and click at (0.0576, 0.0174) and
   (0.0576, 0.075).

**6** Click the **Line** button, and click at (0.0576, 0.175), (0.0924, 0.175), and
   (0.0924, 0.075).

**7** Click the **2nd Degree Bézier Curve** button, and click at (0.0924, –0.0174).

**8** Finish by clicking the right mouse button to close the boundary curve and create a
   solid object.
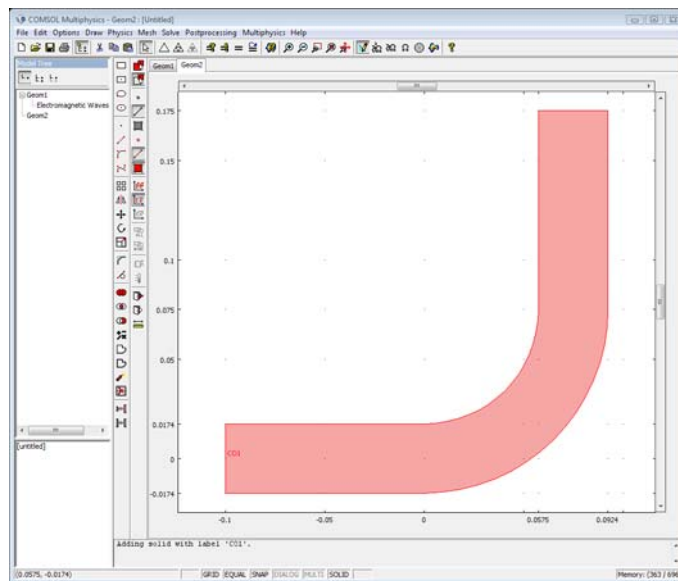
**9** Select **Extrude** from the **Draw** menu. Extrude the object using a distance of 0.0174.

**10** Click the **Zoom Extents** button on the Main toolbar.
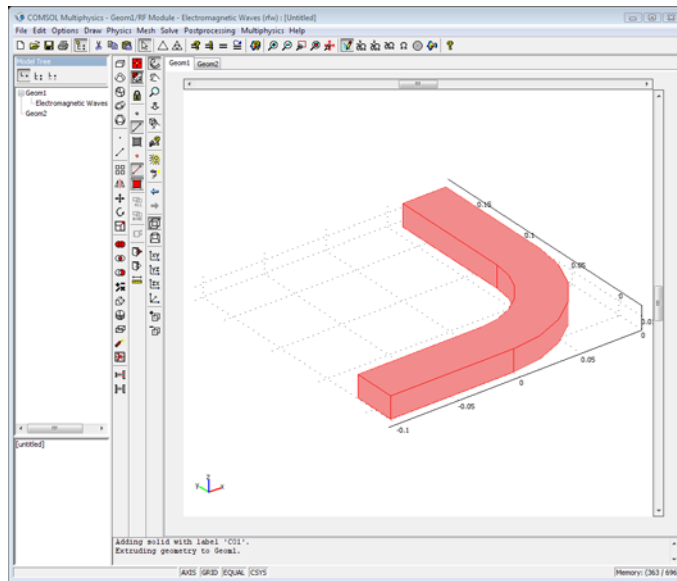


### PHYSICS SETTINGS

*Scalar Variables*

**1** From the **Physics** menu, choose **Scalar Variables**.

**2** In the **Application Scalar Variables** dialog box, set the frequency nu_rfw to fq1, and then click **OK**.

*Boundary Conditions*

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

**2** Select Boundaries 2–8 and 10.

**3** In the **Boundary condition** list, select **Perfect electric conductor** as the boundary condition. These boundaries represent the inside of the walls of the waveguide which is plated with a metal, such as silver, and considered to be a perfect conductor.

**4** On Boundaries 1 and 9, specify the **Port** boundary condition. On the **Port** page, set the values according to the following table; when done, click **OK**.

| SETTINGS | BOUNDARY I | BOUNDARY 9 |
|---|---|---|
| Port number | 1 | 2 |
| Wave excitation at this port | Selected | Cleared |
| Mode specification | Rectangular | Rectangular |
| Mode type | Transverse electric (TE) | Transverse electric (TE) |
| Mode number | 10 | 10 |

*Subdomain Settings*
Use the default values for $\varepsilon_r$, $\mu_r$, and $\sigma$, because the waveguide is filled with air.

**MESH GENERATION**

**1** In the **Free Mesh Parameters** dialog box, click the **Custom mesh size** button, and then type 0.006 in the **Maximum element size** edit field.
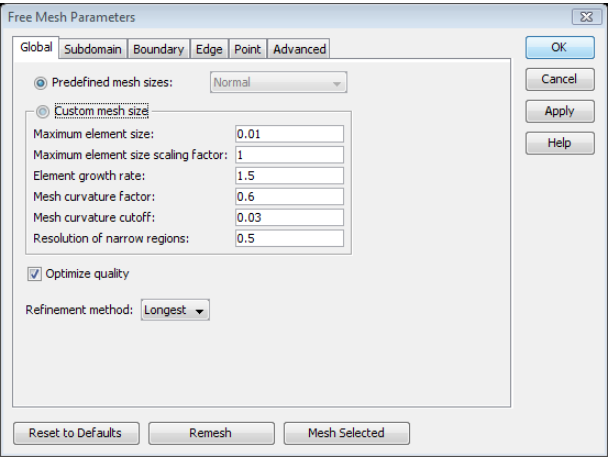
**2** Click **Remesh** to generate the mesh; then click **OK**.
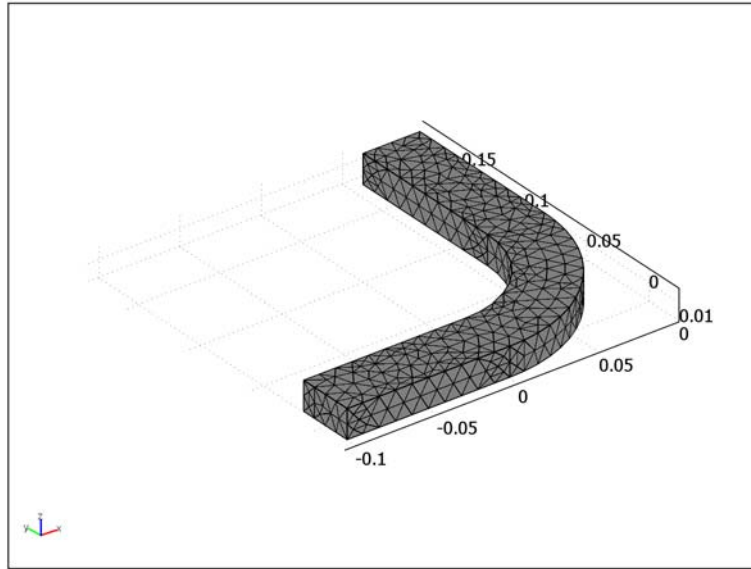
Click the **Solve** button on the Main toolbar to solve the problem.

The default plot is a slice plot of the total energy density. This is a convenient way to visualize the good transmission, because reflections give rise to a wave pattern in the energy distribution. To better see the propagating wave change the position of the slices.

**1** From the **Postprocessing** menu, choose **Plot Parameters**.

**2** On the **Slice** page in the **Plot Parameters** dialog box, set the **Slice positioning x levels** to 0, **y levels** to 0, and **z levels** to 1.

**3** Click **Apply** to see the following plot.



Slice: Total energy density, time average [J/m$^3$]

Max: 1.373e-5

x10$^{-5}$

1.3

1.2

1.1

1

0.9

0.8

0.7

Min: 6.105e-6
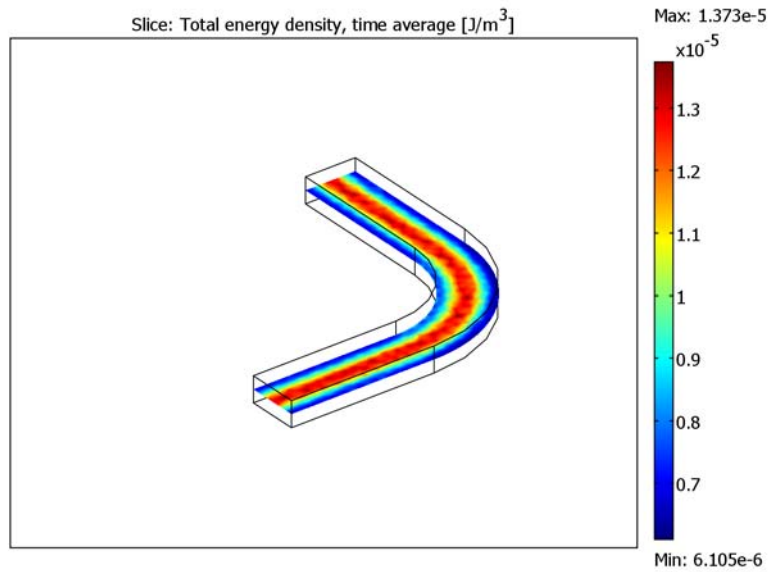
To better see the propagating wave, plot the electric field's $z$-component.

**4** From the **Predefined quantities** list, select **Electric field, z-component**.

**5** From the **Color table** list choose **WaveLight**, then click **OK**.



Slice: Electric field, z component [V/m]
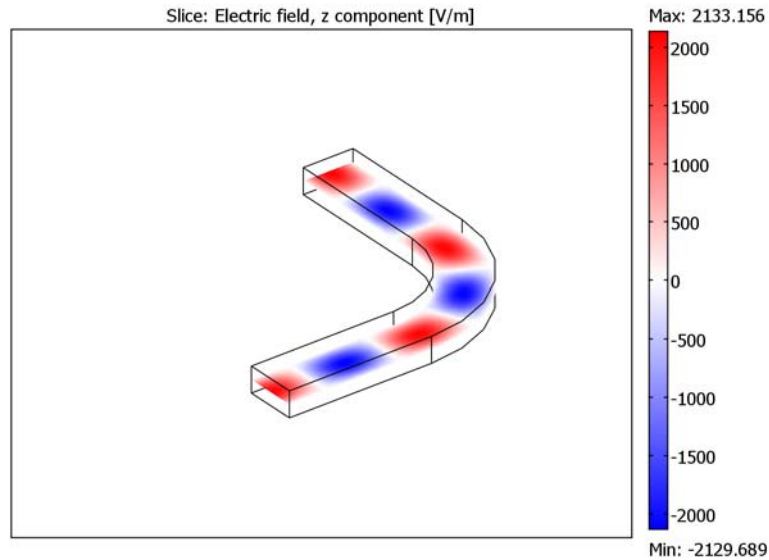
**6** To compare the power flow of the incident wave to the power flow of the outgoing wave, perform a boundary integration. The entrance port is excited using the port boundary condition, which provides an excitation with a power level of 1 W. To obtain the power outflow from the exit port, open the **Boundary Integration** dialog box from the **Postprocessing** menu, and integrate **Power outflow, time average** over Boundary 9. The result is about 1.006 W and appears in the message log at the bottom of the main window.

---

**Model Library path:** RF_Module/RF_and_Microwave_Engineering/
waveguide_hbend_2d

---

*2D Modeling Using the Graphical User Interface*

The results obtained in the 3D calculation are independent of the height $b$ of the waveguide as the $TE_{10}$ wave does not vary in the $z$ direction. This means that the model can just as well be made in 2D.

You find an extended version of this model with an S-parameter study in the section "H-Bend Waveguide with S-parameters" on page 111 in the *RF Module Model Library.* See also the section "S-Parameters and Ports" on page 57 for more information about S-parameter calculations.

**MODEL NAVIGATOR**

**1** Select **2D** from the **Space dimension** list.

**2** Select the **RF Module>In-Plane Waves>TE Waves>Harmonic propagation** application mode.

**3** Click **OK**.

**OPTIONS AND SETTINGS**

Define the same constants as in the 3D model on page 27.

**GEOMETRY MODELING**

Use the same axes/grid settings and draw the same geometry as in the 2D work plane in the 3D model.

**PHYSICS SETTINGS**

*Scalar Variables*

**1** From the **Physics** menu, choose **Scalar Variables**.

**2** In the **Application Scalar Variables** dialog box, set the frequency `nu_rfwe` to `fq1`, and then click **OK**.

*Boundary Conditions*

Use the same boundary conditions as in the 3D model:

**1** On Boundaries 2–4 and 6–8, select the **Perfect electric conductor** boundary condition.

**2** On Boundaries 1 and 5, specify the **Port** boundary condition. On the **Port** page set the values according to the table below:

| SETTINGS | BOUNDARY I | BOUNDARY 5 |
|---|---|---|
| Port number | 1 | 2 |
| Wave excitation at this port | Selected | Cleared |
| Mode specification | Analytic | Analytic |
| Mode number | I | I |

## MESH GENERATION

**1** Initialize the mesh.

**2** Refine the mesh twice.

## COMPUTING THE SOLUTION

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

## POSTPROCESSING AND VISUALIZATION

The default plot shows the electric field's *z* component. Qualitatively, the result coincides with the result in the 3D model. The difference in amplitude is because the input power in the 2D port boundary condition is $1$ W per unit depth ($1$ m).
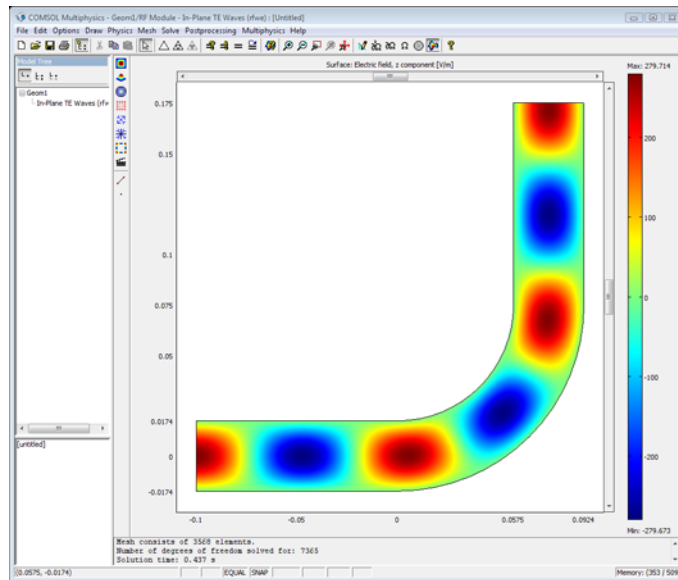


**1** To plot the energy density, open the **Plot Parameters** dialog box.

**2** Click the **Surface** tab, and select **Total energy density, time average** from the **Predefined quantities** list on the **Surface Data** page. Click **OK**.
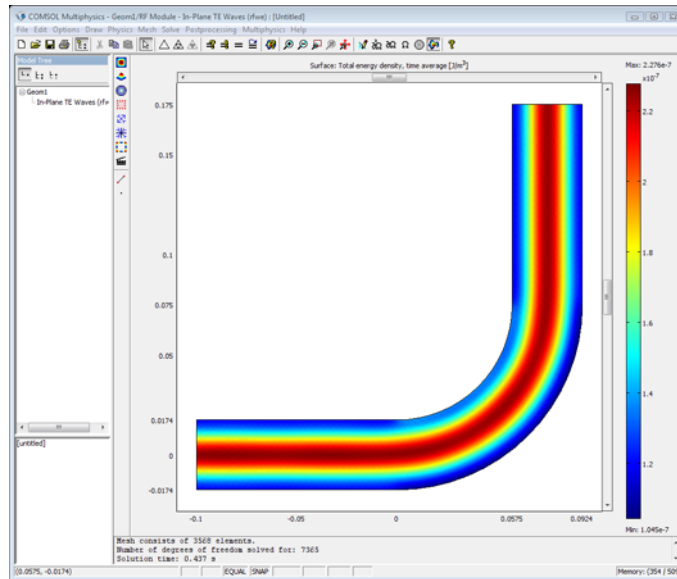


This again shows that the reflections are very small. To further verify this compare the power flow of the incoming wave to the outgoing wave. To compare the power flow of the incident wave to the power flow of the outgoing wave, perform a boundary integration. The entrance port is excited using the port boundary condition, which provides an excitation with a power level of 1 W (per unit depth). To obtain the power outflow from the exit port, open the **Boundary Integration** dialog box from the **Postprocessing** menu, and integrate **Power outflow, time average** over Boundary 5. The result, 0.99993 W, which appears in the message log, indicates that any reflected power is too small to be detected. Note that the 2D model is more accurate than the 3D model because it uses a finer mesh that better resolves the waves.

# Periodic Boundary Conditions

The section "Using Periodic Boundary Conditions" on page 257 in the *COMSOL Multiphysics User's Guide* presents a general description on how to define general periodic boundary conditions. The RF Module has an automatic **Periodic condition** accessible from the **Boundary Settings** dialog box, so it is not necessary to use the **Periodic Boundary Conditions** dialog box. Use the latter dialog box for special cases when you need full control of the periodic condition. The automatic 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.

The 3D Electromagnetic Waves application modes and some transient in-plane application modes use vector elements for the electric and magnetic fields. In these application modes, there is a variable $\Psi$ that implements an extra equation to explicitly set the divergence of the $\mathbf{D}$ or $\mathbf{B}$ field to zero. Similar to using assemblies with vector elements, periodic conditions must use this extra equation when the source and destination of the periodic condition have incompatible meshes; see "Using Assemblies in Electromagnetic Problems" on page 107 for more details. This variable $\Psi$ must also be made periodic, something the automatic periodic condition takes care of if the divergence condition is active. The following example model uses the copy mesh feature to ensure that the mesh on the destination boundary is identical to that on the source boundary. As a result, no extra divergence condition is required.

## *Model Definition*

This example model describes how to define periodic boundary conditions in the 3D Electromagnetic Waves application mode. Consider the cube in Figure 2-2. In this model you let an incident tilted plane wave enter the cube at the top. In order to fulfill the symmetry, the electric field on the two boundaries parallel to the *yz*-plane differ

with a phase shift that depends on the wave number and the incident angle of the plane wave

$$\Phi = k_0 \left( \begin{bmatrix} \sin\Theta \\ 0 \\ -\cos\Theta \end{bmatrix} \cdot \mathbf{r}_{per} \right)$$

where $k_0$ is the wave number for free space, $\Theta$ is the incident angle, and $\mathbf{r}_{per}$ is a vector perpendicular to the symmetry boundaries with a magnitude equal to the distance between them. You solve for the incident angle $\pi/10$ or 36°.
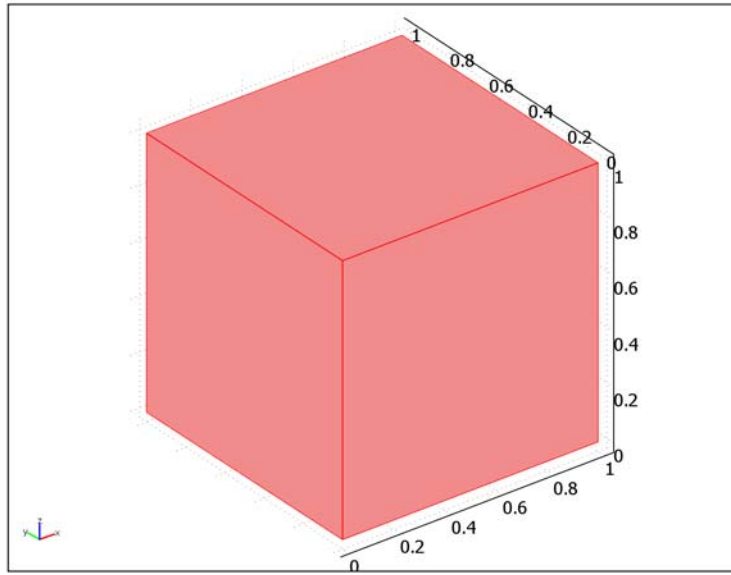


*Figure 2-2: The cubic model geometry.*

## Results and Discussion

The plot in Figure 2-3 shows the *y* component of the electric field for an angle of π/10.



theta(2)=0.314159    Slice: Electric field, y component [V/m]
Arrow: Magnetic field                    Max: 1.125

Min: -1.165
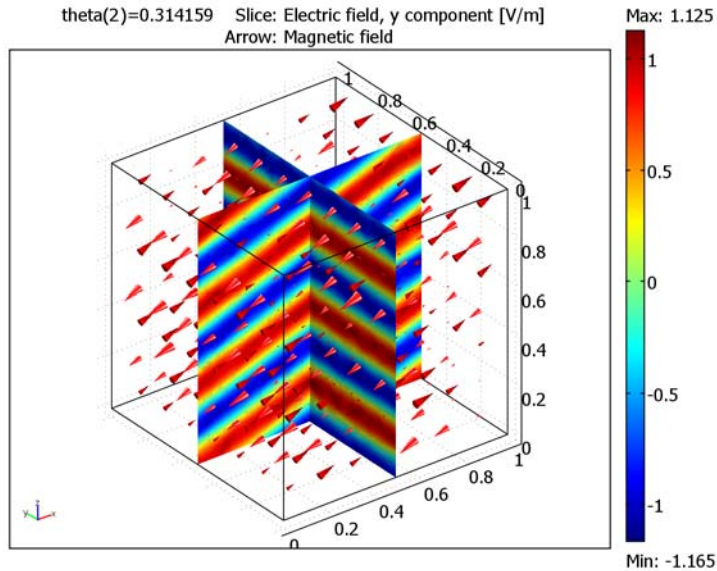
*Figure 2-3: The electric field's y component in a slice plot together with the magnetic field as arrows.*

**Model Library path:** `RF_Module/Tutorial_Models/`
`periodic_boundary_condition`

## Modeling Using the Graphical User Interface

### MODEL NAVIGATOR

**1** In the **Model Navigator** select **3D** from the **Space dimension** list. Open the **RF Module** folder, then select **Electromagnetic Waves>Harmonic propagation**.

**2** Click **OK**.

## GEOMETRY MODELING

**1** Use the **Block** tool to make a cube with sides of length 1.

**2** Click the **Zoom Extents** button on the Main toolbar.

## OPTIONS AND SETTINGS

Open the **Constants** dialog box from the **Options** menu. Define a constant named `theta` with a value equal to `pi/10`. You can also type `36[deg]` if you prefer to use degrees.

## PHYSICS SETTINGS

*Boundary Conditions*

You specify the periodic condition in the **Boundary Settings** dialog box. Select the boundaries that define one periodic condition, which consist of one or more source boundaries plus one or more destination boundaries. If you want several periodic conditions with different orientations, separate them with a periodic pair index entered in the **Periodic pair index** edit field.

**1** Go to the **Physics** menu and open the **Boundary Settings** dialog box.

**2** Select Boundaries 1 and 6, and choose **Periodic condition** from the **Boundary condition** list. Then choose the **Type of periodicity** to be **Floquet periodicity**, and type `k0_rfw*sin(theta)`, `0` and `-k0_rfw*cos(theta)` in the three edit fields for

**k-vector for Floquet periodicity**. The **k**-vector here has the same magnitude and direction as the **k**-vector for the plane wave incident to the cube.
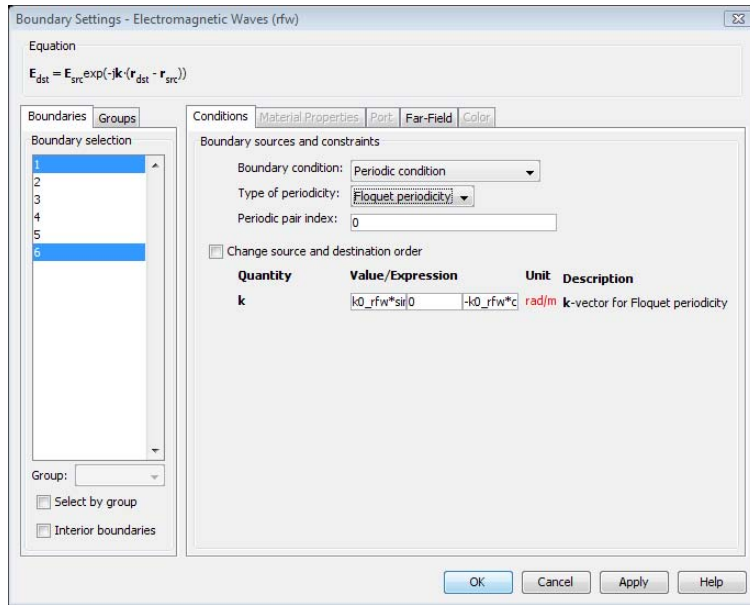


**Note:** The group that contains the boundary with the lowest number becomes the source by default. It is possible to change the order of source and destination by selecting the **Change source and destination order** check box. This can be necessary when the mesh is not identical on the source and destination to make sure that the boundaries with the finest mesh become the destination.

**3** Proceed to set up the incident field. Select boundary 4 and choose the **Scattering boundary condition** from the **Boundary condition** list. In the three edit fields for **Electric field**, type 1 in the middle one. In the three edit fields for the **Wave direction**, type a vector with the same direction as the Floquet periodic condition: `sin(theta)`, `0` and `-cos(theta)`. This vector only defines a direction, so the magnitude does not matter.

**4** For Boundary 3, where the wave exits, choose the **Matched boundary** condition. Then type `k0_rfw*cos(theta)` in the **Propagation constant** edit field.

**5** Click **OK**.

**1** Open the **Free Mesh Parameters** dialog box from the **Mesh** menu and select **Coarse** from the **Predefined mesh sizes** list.

**2** On the **Boundary** page, select Boundary 1 and click the **Mesh Selected** button.

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

**4** Choose **Mesh>Copy Boundary Mesh Parameters**.

**5** Select the check box for Boundary 1 in the **Source boundaries** list.

**6** Select the check box for Boundary 6 in the **Target boundary** list.

**7** Click the **Copy Mesh** button, then **OK**.

**8** Click the **Mesh Remaining (Free)** button on the Mesh toolbar.

**COMPUTING THE SOLUTION**

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

**2** In the **Solver Parameters** dialog box, choose **Direct (PARDISO)** from the **Linear system solver** list.

**3** Click **OK**, then **Solve**.

**POSTPROCESSING AND VISUALIZATION**

**1** Open the **Plot Parameters** dialog box.

**2** Click the **Slice** tab.

**3** Select **Electric field, y-component** from the **Predefined quantities** list.

**4** Set the **Number of levels** in the $x$ and $y$ directions to 1.

**5** On the **Arrow** page, select the **Arrow plot** check box, choose the **Magnetic field** from the **Predefined quantities** list, and click **OK**.

**6** Click the **Scene Light** button.

This will get you an arrow plot of the magnetic field and a slice plot of the y-component of the electric field, for a tilted plane wave. This plot is the one that you see in Figure 2-3 on page 39.

**Note:** If the periodic boundary condition includes a complex phase factor you must select the **Use Hermitian transpose of constraint matrix and in symmetry detection** check box on the **Advanced** page in the **Solver Parameters** dialog box. This is usually the case for a periodic condition of type **Floquet periodicity**, so the setting should be turned on automatically when you use this boundary condition. It might be necessary to select this check box manually, especially when other application modes are present.
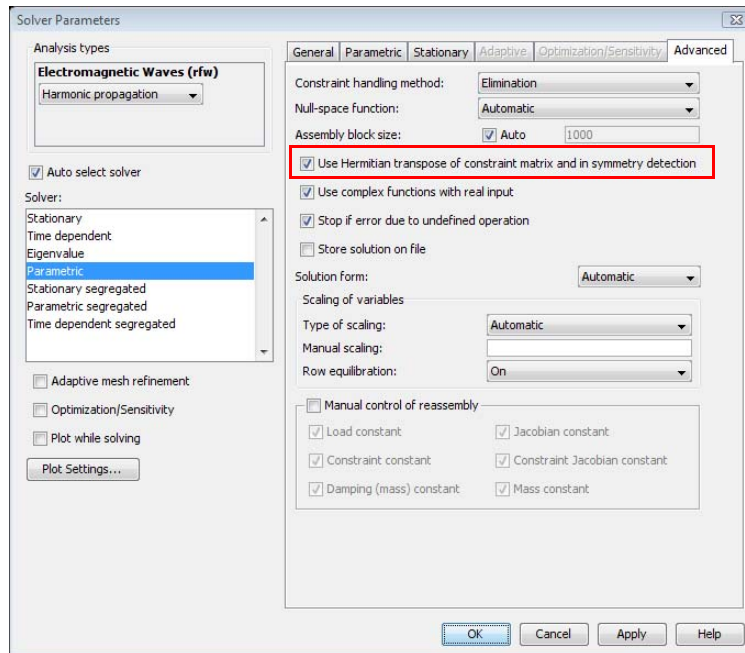
# Perfectly Matched Layers (PMLs)

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 *matched boundary condition*. The former is perfectly absorbing for a plane wave, whereas the latter is perfectly absorbing for guided modes, provided that the correct value of the propagation constant is supplied. However, in many scattering and antenna-modeling problems, you cannot describe the incident radiation as a plane wave with a well-known direction of propagation. In such situations, consider using *perfectly matched layers* (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 as in Ref. 1. The following section describes how to use the semiautomatic PMLs in the RF Module to create planar, cylindrical, and spherical PMLs.

## *PML Implementation*

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

$$t' = t\frac{\lambda}{\delta t}(1-i) \tag{2-1}$$

The coordinate, $t$, and the width of the infinite element region, $\delta t$, are input parameters for each region. These parameters get a default setting according to the drawn geometry, but it does not work for complex configurations. Typical examples of PML regions that work nicely are shown in the following figures for each of the PML types. These types are:

- **Cartesian**—PMLs absorbing in Cartesian coordinate directions.
- **Cylindrical**—PMLs absorbing in cylindrical coordinate directions from a specified axis. For axisymmetric geometries the cylinder axis is the $z$-axis.

- **Spherical**—PMLs absorbing in the radial direction from a specified center point.

  For each of the above PML types, you can choose the coordinate directions in which the PML should absorb waves, that is, for which directions a coordinate transformation of the type Equation 2-1 should apply. To allow complete flexibility in defining a PML there is, in addition, a fourth option:

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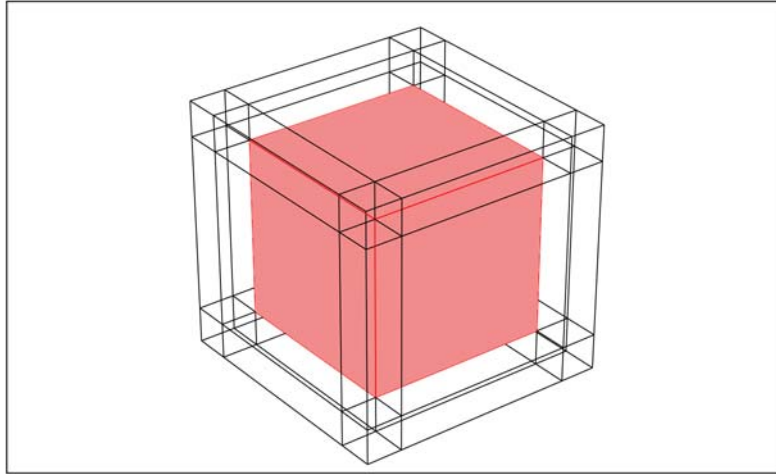


*Figure 2-4: A cube surrounded by typical PML regions of the type "Cartesian."*
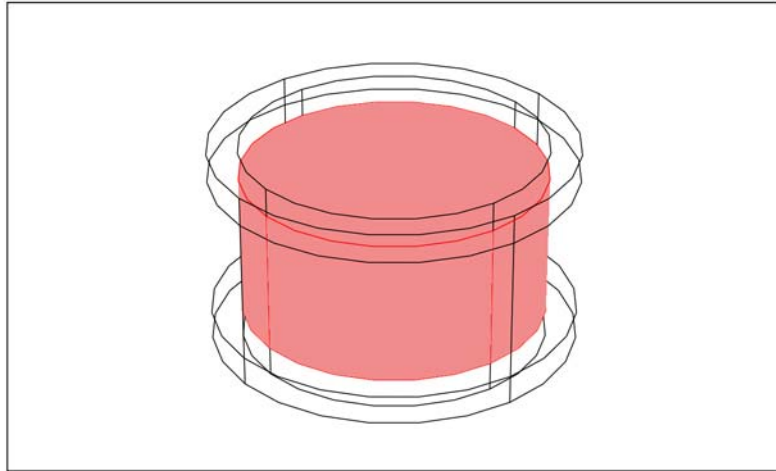
*Figure 2-5: A cylinder surrounded by typical cylindrical PML regions.*
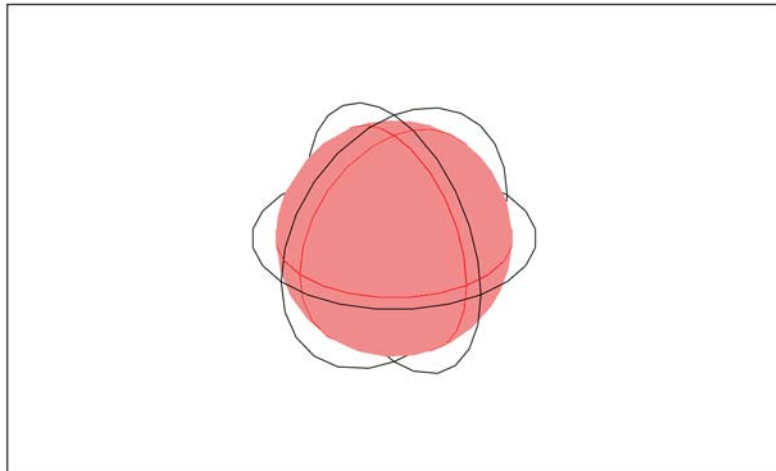


*Figure 2-6: A sphere surrounded by a typical spherical PML region.*

If you use other shapes for the PML regions not similar to the shapes shown in the previous figures, it might be necessary to define the PML parameters manually. The

software store the default parameters in variables with the naming convention `<param>_guess_<suffix>`, where `<param>` is the name of the parameter and `<suffix>` is the application mode suffix.
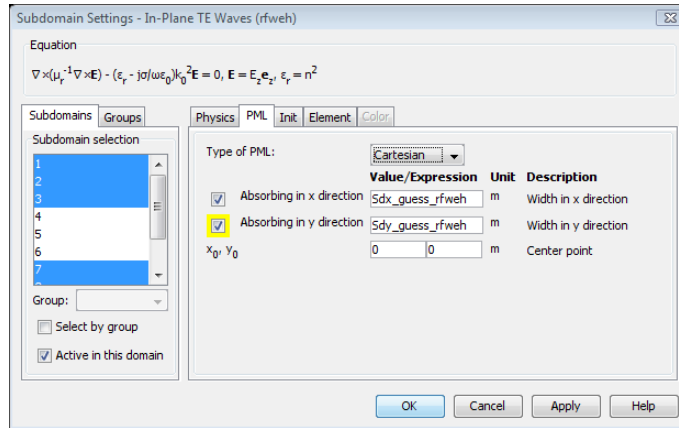


When the width is equal to the actual PML region width in the geometry, the coordinate scaling alters the wavelength so that it is equal to the PML region width. Especially for small geometries, where the size of the geometry is smaller than the wavelength, it might be better to keep the wavelength unchanged in the PML region. Enter `lambdaS_<suffix>` as PML region width to get this effect. The damping is not as good, but it can still be enough to avoid reflections. An absorbing boundary condition outside the PML region often reduce the reflections further; for example, it is often efficient to use the scattering boundary condition for spherical waves together with a spherical PML region.

You can check the values of the default guess variables for the PML parameters in **Equation System>Subdomain Settings** under the **Physics** menu. Click the **Variables** tab and look for variables with _guess_ in the name.
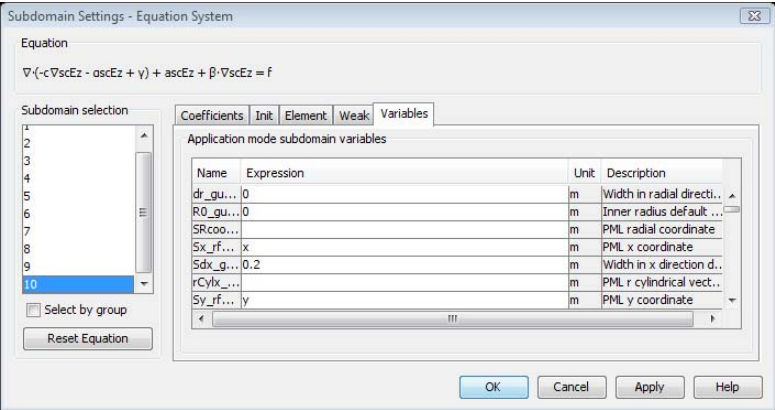


## Reference

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*.

• Dielectric Scattering PML on page 8 (2D Cartesian PML).

• Absorbed Radiation (SAR) in the Human Brain on page 142 (3D Cartesian PML).

• Balanced Patch Antenna for 6 GHz on page 182 (spherical PML).

The model Dielectric Scattering PML is a 2D tutorial model, so it is recommended to start with that one.

# Scattered-Field Formulation

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, the excitation of the plane wave 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.

## *Application Modes with Scattered Fields*

The scattered-field formulation is available for the 3D Electromagnetic Waves, In-Plane Waves, and Meridional Waves application modes. It is accessible as the analysis type **Scattered harmonic propagation** that you choose directly in the **Model Navigator** under one of the abovementioned application modes. Alternatively, you can choose it from the **Field type** list in the **Application Mode Properties** dialog box. You open this dialog box by choosing **Physics>Properties**.

The scattered field is always available as a variable with the letters `sc` in front of the field name, for example, `scE` for the scattered electric field. If you solve for the total field, this variable is the total field subtracted with the incident field. Similarly, 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. You specify the incident field in the **Application Scalar Variables** dialog box, which you open by choosing **Physics>Scalar Variables**.

## *Example Model—Dielectric Scattering PML*

For an example on the scattered-field formulation, see the example model for PMLs in the section "Dielectric Scattering PML" on page 8 in the *RF Module 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 $(\theta, \varphi)$. 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" on page 123.

## *Far-Field Support in the Application Modes*

The 3D Electromagnetic Waves, the Axisymmetric Waves, and the In-Plane Waves application modes support far-field postprocessing. To define the far-field variables use the **Boundary Settings** dialog box. On the **Far-Field** page, select the boundaries where the algorithm then integrates the near field, and enter a name for the far electric field. Make sure the tangential electric field and the tangential curl of the electric field appear in the next two columns. The tabular structure of the dialog box makes it possible to define multiple far-field variables and change the fields used in the integral.
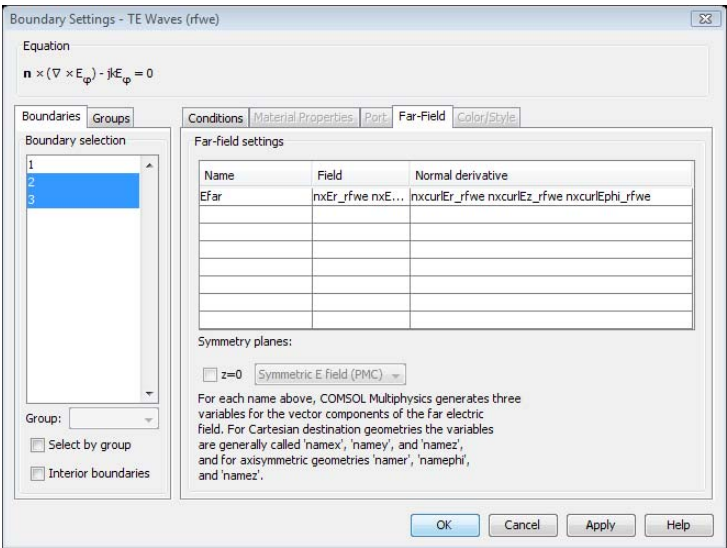
In this dialog box, you can also specify if you want to use symmetry planes in your model when calculating the far-field variable. The symmetry planes have to coincide with one of the Cartesian coordinate planes (the check boxes **x=0**, **y=0**, and **z=0**). For each of these planes it is possible to select the type of symmetry you use, which can be of either **Symmetric E field (PMC)** or **Symmetric H field (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 dialog box. 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 in the direction where the electric field is nonzero. For example, in the In-Plane TE Waves application mode only the *z* component of the far field is available.

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 $(\theta, \varphi)$. 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 Pattern from a Dipole Antenna*

To illustrate far-field postprocessing, this example shows how to calculate the far-field pattern from a simple dipole antenna. The antenna is a current loop with the radius 0.2 m and the frequency 1 GHz. Figure 2-7 shows the resulting near-field radiation

pattern at a distance 0.4 m from the center of the loop together with the far-field radiation pattern.

For another example of far-field postprocessing, see the model "Monoconical RF Antenna" on page 65 of the *RF Module Model Library*.
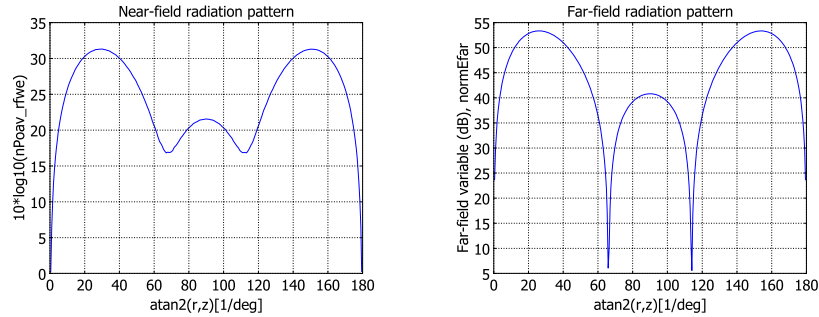


*Figure 2-7: The near-field radiation pattern from the dipole (left), and the far-field radiation pattern (right).*

**Model Library path:** RF_Module/Tutorial_Models/far_field

*Modeling Using the Graphical User Interface*

**MODEL NAVIGATOR**

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

**2** Open the **RF Module** folder, then select **Electromagnetic Waves>TE Waves**.

**3** Click **OK**.

**GEOMETRY MODELING**

**1** Select the **Ellipse/Circle (Centered)** tool and draw a circle centered at the origin with the radius 0.4 using the right mouse button.

**2** Select the **Rectangle/Square** tool and draw a rectangle with corners at (−0.4, −0.4) and (0, 0.4).

**3** Select both the rectangle and the circle and click the **Difference** button. This subtracts the rectangle from the circle.

**4** Finally add a point at $(0.2, 0)$.
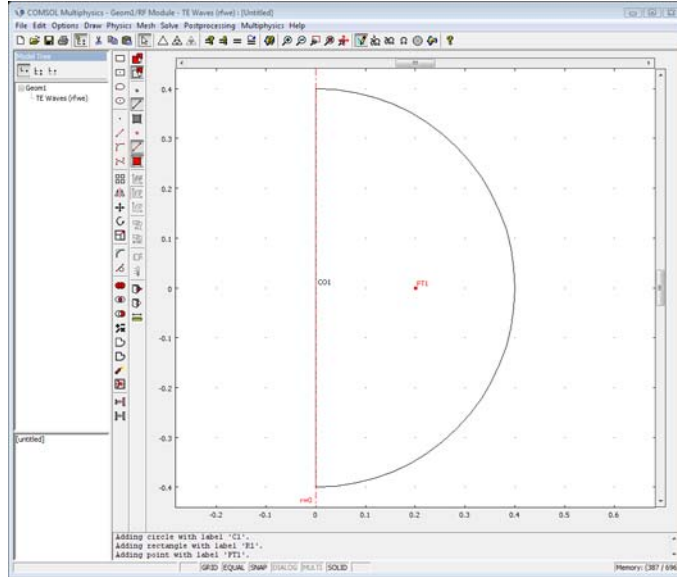
### PHYSICS SETTINGS

*Boundary Conditions*

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

**2** The vertical boundary at $r = 0$ (boundary 1) represents the symmetry axis. Therefore select the **Axial symmetry** boundary condition at this boundary in the **Boundary Settings** dialog box.

**3** The other two boundaries represent an open boundary where there should be no reflections. To achieve as small reflections as possible, select the **Scattering boundary condition** at these boundaries.

**4** Click **OK**.

*Point Settings*

The point at $(0.2, 0)$ (Point 3) represents a circular loop with a current. To specify this current set $I_0$ to 1 at this point in the **Point Settings** dialog box. When done, click **OK**.

### MESH GENERATION

**1** Click the **Initialize Mesh** button on the Main toolbar.

**2** Click the **Refine Mesh** button on the Main toolbar to refine the mesh.

## COMPUTING THE SOLUTION

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

## POSTPROCESSING AND VISUALIZATION

The plot of the solution shows the electric field $E_\varphi$ emitted from the circular loop.
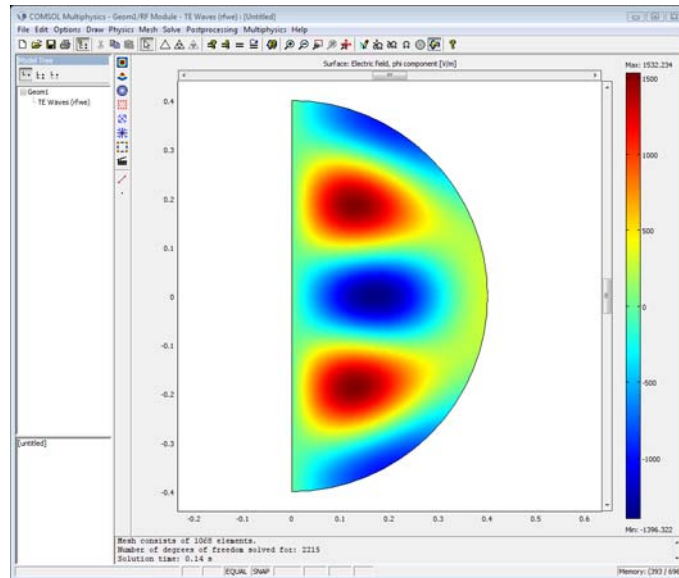


*Far-Field Postprocessing*

To calculate the far field first define a variable for the far electric field.

**1** Open the **Boundary Settings** dialog box from the **Physics** menu.

**2** On the **Far-Field** page select Boundaries 2 and 3 and enter the variable name `Efar`. The tangential electric field and the tangential curl of the electric field then appear in the other two columns.
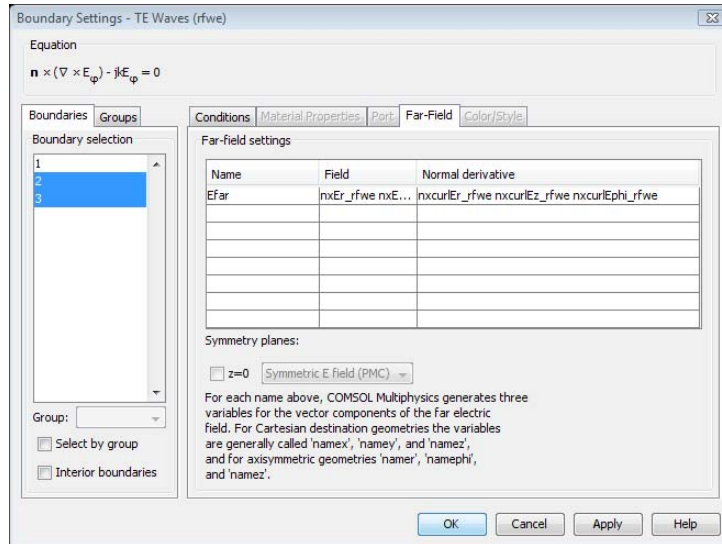


**3** Click **OK**.

**4** The far-field variables are not immediately available. They first have to be evaluated using the previously calculated solution. To do this, choose **Update Model** from the **Solve** menu.

The definition of the variable `Efar` generates a function, `Efarphi(r,z)`, which is the $\varphi$ component of the far electric field in the point $(r, z)$. It also generates a variable, `Efarphi`, which is equal to the function value in every point. Because the near electric field only has a $\varphi$ component, so does the far field. The radiation pattern is proportional to $|\mathbf{E}_{far}|^2$, which you can access with the expression `normEfar^2` and its dB value with `normEfardB`. To visualize the radiation pattern follow these steps:

**1** Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.

**2** On the **Line/Extrusion** page, select Boundaries 2 and 3.

**3** In the **y-axis data** area, enter the **Expression** `normEfardB`.

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

**5** In the **X-Axis Data** dialog box, enter the **Expression** `atan2(r,z)[1/deg]` to display the polar angle in degrees on the horizontal axis. Click **OK** to close the dialog box.

**6** Click **OK** to plot the far-field radiation pattern on a dB scale.



Far-field radiation pattern
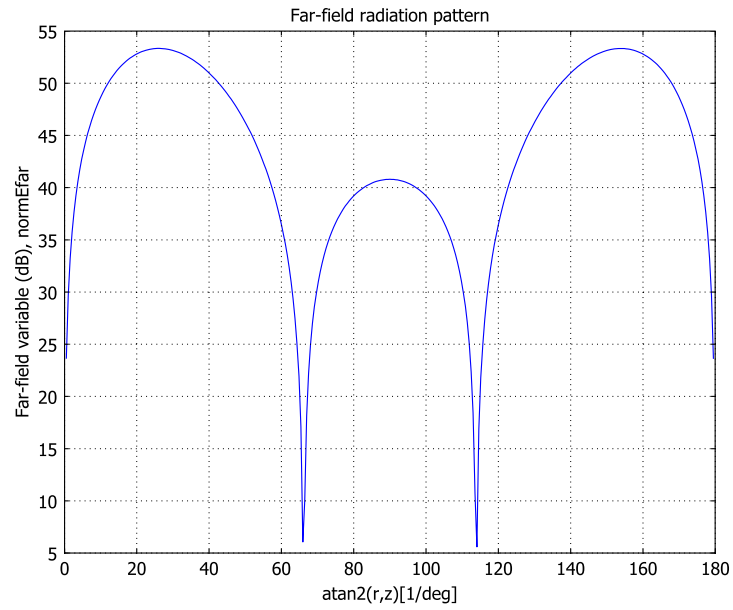
# S-Parameters and Ports

## *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" on page 119.

## *S-Parameter Calculations in COMSOL Multiphysics*

The In-Plane Waves, Axisymmetric Waves, and the 3D Electromagnetic Waves application modes have built-in support for S-parameter calculations. To set up an S-parameter study use the port boundary condition for each port in the model. Specify the eigenmode of interest at each port. This is done on the **Port** page in the **Boundary Settings** dialog box. A port boundary condition results in no reflections for the particular mode and gives you access to the S-parameters.

### THE COEFFICIENT PAGE

Under the coefficient tab you set a boundary as a port when you select **Port** from the **Boundary condition** list. This activates controls for specifying port number, port as input, and port power level and phase.

Each port must have a unique port number. One of the ports should be an inport with an incident field. To specify the inport, select the **Wave excitation at this port** check box for that port.

---

**Note:** It is only possible to excite one port at a time if the purpose is to compute S-parameters. If you excite several ports, the S-parameter calculation is turned off, and you can only use the ports to excite waves. The script `sparametermatrix.m`, available when you run COMSOL Multiphysics with MATLAB, allows you to cycle through the ports and compute the entire S-matrix. In other cases, for example, when studying microwave heating, more than one inport might be wanted, but then the S-parameter variables cannot be correctly computed.

---

To adjust the power level of the inport, use the **Power level** edit field next to the check box. It is also possible to enter a phase shift for the input signal in the **Port phase** edit field. The port phase is intended to use when several ports are used as inports because it does not affect the S-parameter calculation.

### THE PORT PAGE

The **Port** page gets enabled when you select a port boundary condition. This tab has controls for specifying the mode of the port. You can specify three different types of port modes:

- Modes for ports of certain shapes: rectangular, circular, and coaxial. They all apply to port of a homogeneous medium with metallic boundaries.
- Numeric modes with the data from a mode analysis made in the Boundary Mode Analysis application mode.
- User-defined modes, which lets you enter the expressions for the fields manually. The fields can be complex-valued if you like. The **Port phase** edit field has no impact for this mode type because the phase is determined by the entered fields.

*S-Parameter normalization*

You control the type of S-parameter normalization that is used via the **S-parameter output** field at the bottom of the **Port** page. Here you can choose between **Magnitude and phase** or **Magnitude only**.

*Rectangular and Circular 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. In 2D the mode number is a single number. If you select the node number $n$ for an in-plane wave this gives you the $TE_{n0}$ or $TM_{n0}$ wave. For axisymmetric waves the mode number is the mode number for the radial variation of the mode.
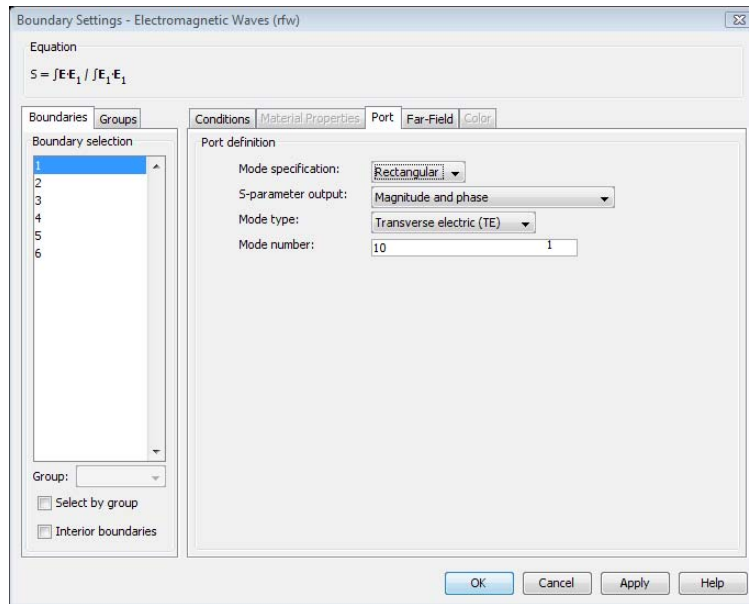


*Coaxial Modes*

Coaxial modes are available in 3D and for axisymmetric TM waves. Only the fundamental mode is available as a predefined mode.

*Numeric Modes*

The numeric option is available only if there is a Boundary Mode Analysis application mode in the model. You can take the data for the eigenmode from such a mode analysis

and use it in the S-parameter analysis. Specify the mode type of the solution to the boundary mode analysis by selecting from the **Mode type** list:

• **Automatic** means that the mode type is determined from the settings in the Boundary Mode Analysis application mode. This works if you have used the Boundary Mode Analysis application mode to calculate TE or TM waves. If hybrid-mode waves have been calculated, the mode type cannot be determined automatically.

• Use the other options to explicitly specify the mode type: transverse electric (TE), transverse magnetic (TM), or transverse electromagnetic (TEM).

If there are more than one Boundary Mode Analysis application mode in the model, select which one to use from the **Use numeric data from** list.
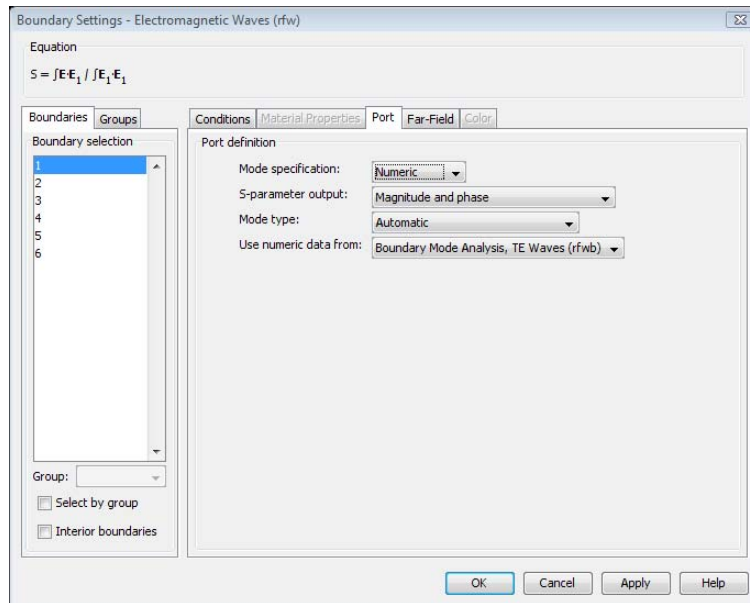


When using a numeric mode the mode analysis and the S-parameter analysis need to be done in sequence:

**1** Open the **Solver Manager** and make sure that **Current solution** option button is active for solutions not solved for.

**2** Select only the Boundary Mode Analysis on the **Solve For** page in the **Solver Manager** and solve that problem. If you have several Boundary Mode Analysis application modes, you only select one of them.
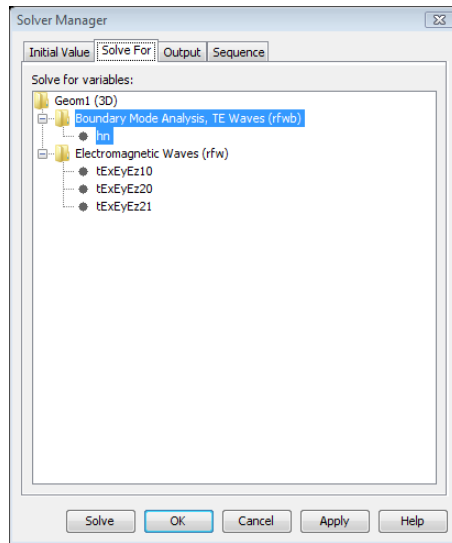


**3** From the **Physics** menu, choose **Properties** to open the **Application Mode Properties** dialog box. From the **Specify eigenvalues using** list, choose **Manual eigenvalue** and click **OK**. When you perform this operation, you copy the propagation constant for solution currently plotted into an application mode scalar variable for the propagation constant. You can also change this value manually by opening the **Application Scalar Variables** dialog box from the **Physics** menu. All the postprocessing variables now use this propagation constant instead of calculating them from the eigenvalues. The application mode is now in a frozen state where it makes no sense

to solve for eigenvalues. So before re-solving this application mode again, switch back to the original setting in the **Properties** dialog box.
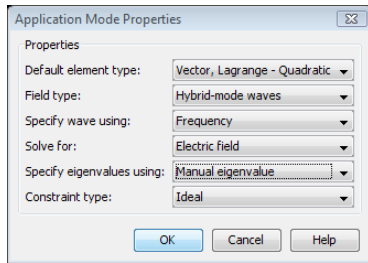


**4** If you have several numeric ports, repeat steps 2and 3 for the other Boundary Mode Analysis application modes.

**5** Then switch from the eigenvalue solver to the stationary solver in the **Solver Parameters** dialog box.

**6** Select only the Electromagnetic Waves application mode in the **Solver Manager**. On the **Initial Value** page, click the **Store Solution** button to store the result from the mode analysis.

**7** In the **Initial value** area, click the **Stored solution** button and select which solution to use. It is important that you select a solution for the eigenvalue that match the propagation constant in the scalar variables. When solving the wave propagation problem, the port boundary condition uses the stored solution.

When making a parametric study of the S-parameters as a function of frequency it is important that you keep the frequency variable for the Boundary Mode Analysis application mode constant. Only the frequency variable for the 3D Electromagnetic Waves application mode is allowed to vary. When calculating the data in the port boundary condition as a function of frequency it is necessary to know for which frequency the mode analysis were made. The two frequency variables are found in the **Application Scalar Variables** dialog box, as you can see in Figure 2-8. The Boundary Mode Analysis application mode uses the variable nu_rfwb, and the 3D Electromagnetic Waves application mode uses the variable nu_rfw. Clear the **Synchronize equivalent variables** check box to be able to edit them individually. The suffixes of the variable names, _rfwb and _rfw, determine which variables is which.

The suffix is given by the application mode name. The names used here are the default names.
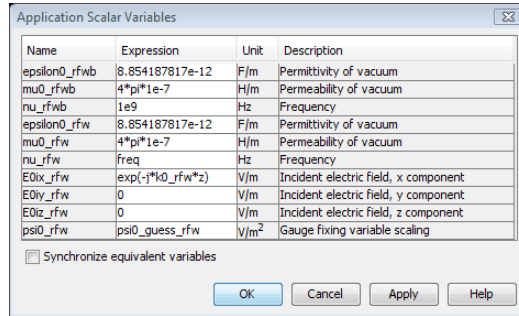


*Figure 2-8: The Application Scalar Variables dialog box with the variables for the Boundary Mode Analysis and 3D Electromagnetic Waves application mode.*

### 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:

- Select the mode type. The available options are transverse electric (TE), transverse magnetic (TM), and transverse electromagnetic (TEM).

- Enter the amplitude of the electromagnetic field in terms of either the electric or the magnetic field.

- The propagation constant can be given in three different ways. One method is to only specify the propagation constant $\beta$ for the current frequency. This identical to the way the matched boundary condition is specified. The only difference is that the port boundary condition uses a normalization to a certain power.

  The next method is to specify a reference frequency $\nu_0$ and the propagation constant $\beta_0$ for that frequency. The propagation constant as a function of frequency is then calculated as

$$\beta = \sqrt{\beta_0^2 - 4\pi^2 \varepsilon\mu(\nu_0^2 - \nu^2)}$$

  The third option is to specify the cutoff frequency, $\nu_{\text{cutoff}}$. The propagation constant as a function of frequency is in this case calculated as

$$\beta = \sqrt{4\pi^2 \varepsilon\mu(\nu^2 - \nu_{\text{cutoff}}^2)}$$

See Figure 2-9 on page 64, which shows where to enter these settings in the **Subdomain Settings** dialog box.
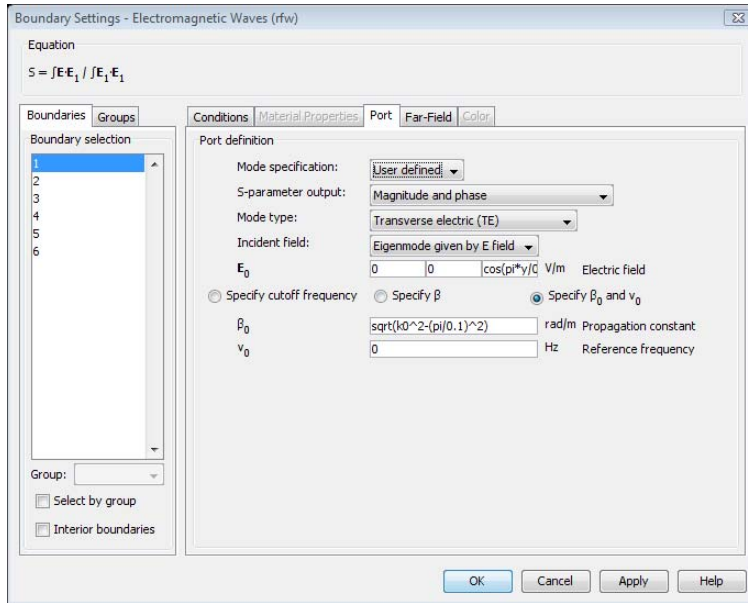
*Figure 2-9: Specification of the mode type, electric field components, and the propagation constant.*

### S-Parameter Variables

The RF Module automatically generates variables for the S-parameters. The port numbers 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_rfw and S21_rfw. S11_rfw is the S-parameter for the reflected wave and S21_rfw is the S-parameter for the transmitted wave. For convenience, two variables for the S-parameters on a dB scale, S11dB_rfw and S21dB_rfw, are also defined using the following relation:

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

The application mode name determines the suffix in the variable names (in this case _rfw), so it may vary.

The S-parameter variables are added to the **Predefined quantities** plot lists of the following postprocessing dialog boxes:

- **Global Plot Variables**. Open with **Postprocessing>Global Plot Variables**.
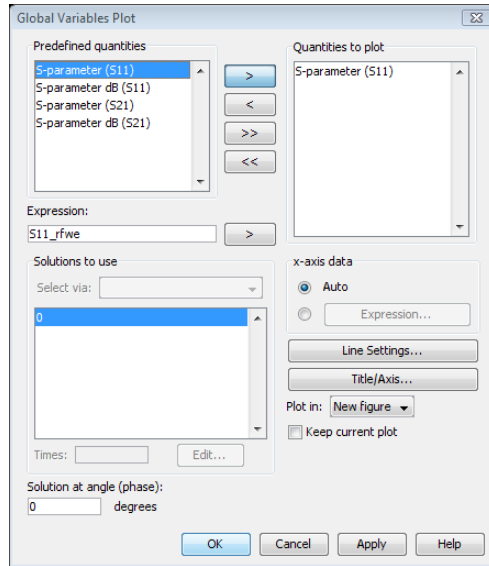


- **Global Data Display**. Open with **Postprocessing>Data Display>Global**.
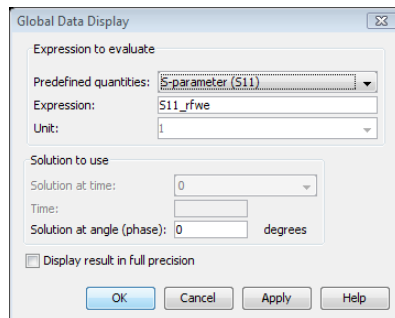


- **Domain Plot Parameters**. Open with **Postprocessing>Domain Plot Parameters**, and under the **Point** tab choose an arbitrary point.
- **Point Evaluation**. Open with **Postprocessing>Point Evaluation**. Choose an arbitrary point.

**Note:** When displaying S-Parameter variables, the phase of the solution must be set to zero. Otherwise, incorrect values will be displayed. In the various dialog boxes for displaying and plotting data that are available from the **Postprocessing** menu, you can set the phase of the solution via the **Solution at angle (phase)** edit field. If the dialog box contains several tabs, it usually appears on the **General** page.

*Parametric Studies*

Once the boundary condition is set up it is easy to make a parametric study using the parametric solver. Use the parametric solver and introduce a parameter representing the frequency. Set the frequency variable of the 3D Electromagnetic Waves application mode to this parameter in the **Application Scalar Variables** dialog box.

To plot the S-parameters as a function of frequency use the **Global Plot Variables** or **Domain Plot Parameters** dialog box. The S-parameter variables are available in the list of **Predefined quantities**. The S-parameters are global variables, so they are available at all levels in a geometry.

*Calculations Below the Cutoff Frequency*

When doing S-parameter calculations below the cutoff frequency you might get an error message saying **Attempt to evaluate real square root of negative number**. This is because the propagation constant is calculated as the square root of an expression that becomes negative. The default solver settings do not allow you to take the square root of real numbers. To overcome this behavior select the **Use complex functions with real input** check box on the **Advanced** page in the **Solver Parameters** dialog box.

*Model with S-Parameter Calculations*

For an example of how to use the boundary condition described above to make an S-parameter analysis, see the model "Waveguide Adapter" on page 116 of the *RF Module Model Library*. The model "Microstrip on a Printed Circuit Board" on page 20 of the *RF Module Model Library* has a detailed description how to model numerical ports with a boundary mode analysis application mode.

## S-Parameter Matrix Extraction and Export

In many situations it is of interest to calculate the entire S-parameter matrix, where each port in the model is used as input port. This is possible to do at the MATLAB command line using the function `sparametermatrix`. In addition, if the S-parameter calculation is made as a frequency sweep, it is possible to export the frequency-dependent S-parameter matrix to a Touchstone file with the command `touchwrite`. See Chapter 4, "Function Reference," of the *RF Module Reference Guide* for more information about these commands. For an example of how to use them, see the model "Three-Port Ferrite Circulator" on page 50 of the *RF Module Model Library*.

## Ports and Assemblies

You can declare ports for an assembly pair, and such ports behaves a bit different than ordinary ports. An assembly pair condition can allow a discontinuity in the solution field variables across the pair interface. This is used to remove the incident field on one side of the interface. The incident field can then travel into a waveguide, but the reflected waves just pass the interface and into the other side unaltered. The other side can be a PML region that effectively damps all types of reflected waves. This is useful for multimode simulations where the port boundary gives reflections on higher modes than it is designed for.

The incident wave travels into the master side of a pair. The slave side gets the wave without the incident field. It is therefore important to verify that the master-slave selection is correct in the **Identity Boundary Pair** dialog box. You open that dialog box from the **Physics** menu by choosing **Identify Pairs>Identity Boundary Pairs**.

For an example how to set up these assembly ports, see the model "Coaxial to Waveguide Coupling" on page 203 of the *RF Module Model Library*.

---

**Note:** Using analytical ports on assembly interfaces require that imprints are used or that the master and slave side are identical. Otherwise the analytical mode is calculated on the entire master boundary and the mode does not represent a true mode of the interface.

---

# 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. 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.

A lumped port specified as an input port calculates the impedance, $Z_{\text{port}}$, 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 $V_{\text{port}}$ is the extracted voltage for the port given by the line integral between the terminals averaged over the entire port. The current $I_{\text{port}}$ 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. You can find more details on how to use voltages and currents in the boundary condition in "Lumped Port Parameters" on page 122.

## *Lumped Ports in the RF Module*

Not all application modes can use lumped ports due to the polarization of the fields and how sources are specified. In 2D and axial symmetry only the transient analysis type for TM and hybrid-mode waves can use lumped ports. In 3D you can use them if you solve for the electric field in a time-harmonic or transient analysis. Neither eigenfrequency nor scattered field analyses can use lumped ports. For the application modes and analysis types that supports the lumped port, you find the **Lumped port** boundary condition in the **Boundary condition** list.

When you select **Lumped port** from the **Boundary condition** list, the dialog box provides inputs for specifying the port number and the port's voltage and phase.

Each port must have a unique port number. To connect a voltage generator at a port, select the **Voltage generator at this port** check box for that port.

To adjust the voltage of a port with a voltage generator, use the **V$_{in}$** edit field. For time-harmonic simulations it is also possible to enter a phase shift for the input signal in the **Port phase** edit field.

## THE PORT PAGE

Just as for the port boundary condition, the **Port** page gets enabled when you select a lumped port boundary condition. For lumped ports, this page provides an interface for specifying the impedance of the external cable attached to the port and the shape of the port area. Depending on the dimension, you can specify three different types of port shapes:

- Uniform shapes where the are two parallel metallic boundaries on both sides of the port area. This selection is not available for axial symmetry.

- Coaxial shape where the port area terminates two concentric cylinders of metallic boundaries. This selection is available for 3D and axial symmetry.

- User-defined shapes where you manually specify all the parameters necessary for the lumped port: the height of the port, the width of the port, and the voltage reference direction. The height of the port is the distance between the metallic boundaries. The voltage reference direction defines the line integral to get the voltage from the electric field. The width of the port is an effective width to convert between wave impedance and characteristic impedance of the cable.

## VARIABLES

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

| NAME | DESCRIPTION |
|---|---|
| Vport_*<port nr>*_g*<geometry nr>_<suffix>* | Extracted port voltage |
| Iport_<port nr>_g<geometry nr>_<suffix> | Port current |
| Zport_*<port nr>_<suffix>* | Port impedance |

For example, a lumped port with port number 2, defined in the first geometry, for the Electromagnetic Waves application mode with suffix `rfw`, defines the port voltage variable `Vport_2_g1_rfw`.

## *Example Model—Microstrip on a Printed Circuit Board*

For an example of how to use the lumped port boundary condition, see the model "Microstrip on a Printed Circuit Board" on page 20 of the *RF Module 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. The ECAD Import feature is available in the AC/DC Module, the MEMS Module, and the RF Module.

### PHYSICS IMPORT VS. GEOMETRY IMPORT

There are two main steps when importing a PCB layout file. First it imports the geometry objects and if importing into 3D, creates a 3D model of the layout. The next step is the *Physics import*. Physics import means that the information in the input file is used to set up named groups for the physics settings in COMSOL Multiphysics.

### 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 ordinary 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 may therefore take a very long time to go from Draw mode to one of the Physics modes (Subdomain mode, Boundary mode, etc.). In some situations this operation may fail.

As a result of these performance issues, the ECAD Import has its own extrude operation that automatically connects non matching 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 may fail. You can then switch to one of the other grouping options to get 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 under 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.

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 will still position 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" on page 76).

- Use the 3D GDS-II import with the ECAD import. The standard CAD import of COMSOL Multiphysics does not support prereading 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 minus sign in geometry expressions).

## 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, since it is 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" on page 72 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, it has to be changed before importing it. Throughout the rest of this chapter, files of this type are referred to as a Netex file.

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 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 will ignore 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, choose **Import>ECAD Import** from the **File** menu. This opens up the **Import ECAD Data From File** dialog box, where you only select the file you want to import. When you have selected the file and clicked the **Import** button, the program reads the file and the **ECAD Import Options** dialog box appears. The appearance of this dialog box is different depending on what file type you selected.

### THE ECAD IMPORT OPTIONS DIALOG BOX

*The ECAD Page*
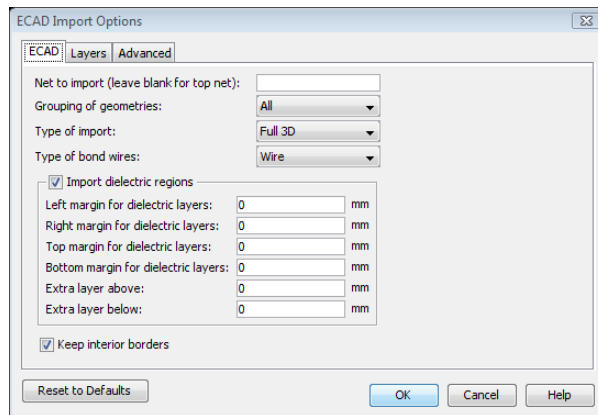


On the **ECAD** page there are a number of settings that control how to treat the information in the layout file. The content of this tab depends on the file type and if you import to a 3D geometry or a 2D geometry. The dialog box example above is for a Netex file.

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 term *cell* is used instead of net, but structurally they mean the same thing. Therefore, the edit field is labeled **Cell to import** when importing GDS files. This option is not present when importing ODB++ files.

The list **Grouping of geometries** 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 can switch to one of the physics modes without the need for 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 list **Type of import** specifies how to treat metal layers. The **Full 3D** option imports all metal layers with a thickness. Select the **Metal shells** options if you want to import all metal layers as an embedded boundary between dielectric regions.

For Netex files, bond wires or wirebonds can be imported in three different complexity levels. You choose the level from the **Type of bond wires** list and the choices are:

- **Wire**. 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.

- **Path of blocks**. The bond wire is modeled as a solid with a square cross section.

- **Path of cylinders**. Same as above but with a circular cross section.

- **Skip**. Import everything else and skip all bond wires.

In most electromagnetic simulations the material between the metal layers is important for the simulation result. If you select the **Import dielectric regions** check box, 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 file, a GDS file, or if there is no outline information in the ODB++ 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. A third option is to tag a layer that defines the outline; see the "The Layers Page" on page 79 for more details. As an extra option, you can also add an extra layer above and below the PCB. The unit for the margin specification is determined by the information stored in the file and is read before the **ECAD Import Options** dialog box opens.

With the **Keep interior borders** 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.
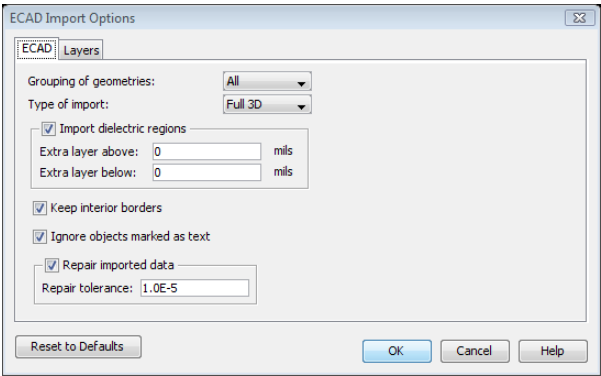


Clearing the **Ignore objects marked as text** check box tells the importer to skip all objects in an ODB++ file that have the TEXT tag set. It is common that PCB layouts have text written in copper. Such objects increase the problem size and are usually of no interest in a physical simulation.

*The Layers Page*
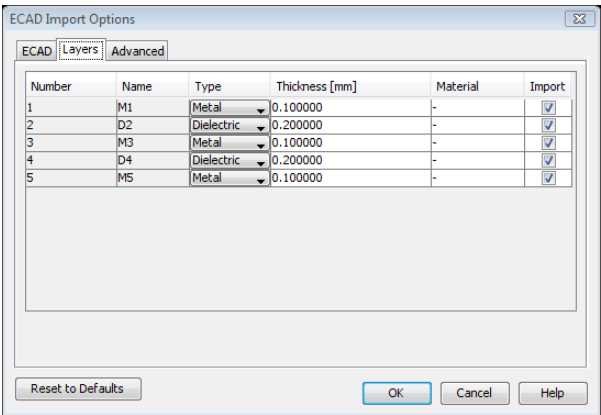


The layer information from the file is shown in a table, and you can change some of the table information here. The following columns are possible to change:

• 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 shells**. The **Outline** type uses a union of the objects in the selected layer as a PCB outline. For ODB++ files, you use the **Drill** type to define that the objects in the layer define drilled via holes through the PCB. For Netex files, the vias are defined within each metal and dielectric layer.

- The **Thickness** column. The thickness of the layer in the unit given in the column header (same as the margins). 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 **Material** column does not have any effect on the imported layer but has useful information about the expected material for the layer.

- 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 there is also another solid layer being imported. Then you must perform two imports. The only exception to this rule is when the import results in only face objects.

*The Advanced Page*
Only available for Netex and GDS files.
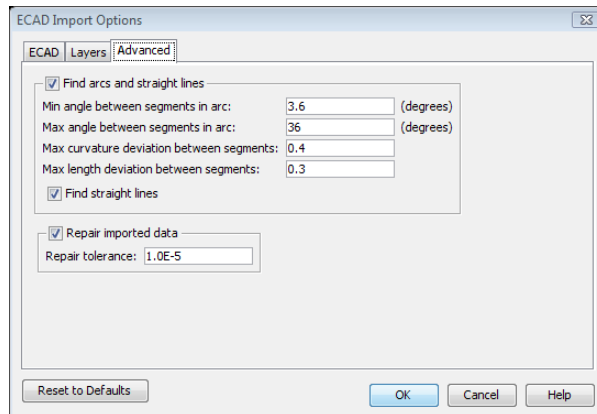


On this page you find the **Absolute import tolerance** setting. To activate the use of this tolerance during import, select the **Repair imported data** check box. For ODB++ files you find this option under the **ECAD** page.

Other options that can significantly reduce the complexity of imported layouts are the recognition of arcs and straight lines. With the **Find arcs and straight lines** check box selected, all polygon chains that represents arcs are identified and replaced with a more

efficient arc curve object. With the edit fields linked to this feature 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 **Min angle between segments in arc** edit field to determine if a group of segments lies on the same straight line.

*Importing Physics Settings*

If there is an existing application mode in the model when you launch the ECAD import, the final step in the import is to create selection groups for subdomain settings and boundary settings. These groups can be useful when you define the physics settings in the subdomains and on the boundaries. In the **Subdomain Settings** dialog box, for example, there is one group defined for each imported net. The dielectric layers also get separate group names. This makes it much easier when you want to use different material properties on metals and insulating layers.

The boundary groups get their names from the net on horizontal boundaries. This makes it easy to setup metal layers imported as boundary shells. The vertical boundaries get no special group name. Note that some groups might be split depending on the type of boundaries present in the group (internal, inactive, or external).

### GEOMETRY IMPORT

Although it is possible to use the standard CAD Import interface to import ECAD files, it is not recommended. With this interface it is not possible to change any of the options described in the previous section, so you always import the file using the default settings.

## Postimport Operations

### PHYSICS SETTINGS

The ECAD import creates subdomain and boundary groups as described in "Importing Physics Settings" on page 81. You can use these groups to easily select the regions that have the same setting, for example, a common material.

*Manual Selection of Multiple Subdomains or Boundaries*

You can also turn off the mouse control of the view with the **Orbit/Pan/Zoom** button on the Camera toolbar, and then select subdomains or boundaries using a rubber-band box selection (all objects within a rectangle). The **Go to XY View**, **Go to YZ View**, and **Go to ZX View** buttons on the Camera toolbar are also helpful. For high aspect ratios you

can deactivate the **EQUAL** button on the status bar by double-clicking it, which scales up the thickness of the imported geometry.

### 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.

**1** First switch to boundary mode by clicking the **Boundary Mode** button on the Main toolbar.

**2** Select all the boundaries in the bottom (or top) plane of the geometry.

**3** Click the **Mesh Selected (Free)** button on the Main toolbar.

**4** Click the **Subdomain Mode** button on the Mesh toolbar to prepare to select subdomains to mesh.

**5** Select all the subdomains in the layer adjacent to the boundaries you just meshed.

**6** Assuming that this is a metal layer, click the **Mesh Selected (Swept)** button on the Mesh toolbar.

**7** Select all subdomains of the next layer and click the **Mesh Selected (Free)** button on the Mesh toolbar because this is a dielectric layer. It is sometimes necessary to increase the scaling along the thickness of the dielectric layer.

**8** Repeat Steps 6 and 7 until all layers are meshed.

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.

**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 borders** check box on the **ECAD** page of the **ECAD Import Options** dialog box. 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 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 will be 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 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 an ordinary CAD import from the **File** menu, COMSOL Multiphysics is in the Draw mode. When you continue to one of the physics modes with multiple geometries 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 you can always continue to the physics modes.

Another possibility is to use assemblies, because then COMSOL Multiphysics does not have to combine the objects (parts) before going to the physics modes. 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.

### DISPLAY PERFORMANCE

The large number of faces that you get when importing a complex layout can cause performance problems for the Java 3D renderer. To minimize the risk of such problems you can adjust the visualization settings so that the software uses a coarse rendering mesh and in some cases also choose to render only edges.

1 From the **Options** menu, open **Visualization/Selection Settings** dialog box.

2 To use a coarse rendering mesh, choose **Coarse** from the **Visualization mesh** list.

3 If you wish to render only edges, make sure that all check boxes are cleared except the **Edge** check box in the **Render** frame.

4 Click **OK** to apply the new settings.

# SPICE Circuit Import

It is possible to add circuit elements as ODE variables to a COMSOL Multiphysics model. These variables can 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 circuit definition comes from a netlist entered in the SPICE format developed at University of California, Berkeley (Ref. 1). Most circuit simulators can export to this format or some dialect of it. The SPICE circuit import contains an additional syntax to support linking to COMSOL Multiphysics models.

## SPICE Import

If you choose **SPICE Circuit Editor** from the **Physics** menu a dialog box appears where you can enter a netlist. You can either enter the netlist commands directly in the **SPICE netlist** text area or use any of the toolbar buttons to generate the proper commands. It is also possible to load a netlist file saved on disk. The contents of the text area are saved with the COMSOL Multiphysics model, so the netlist is still there when you open the model again.
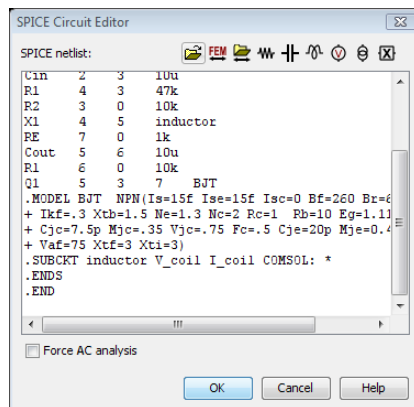


The two check boxes controls two options during the import. Selecting the **Force AC analysis** check box always produces a time-harmonic implementation of the circuit. If you couple the circuit to a time-harmonic application mode, the import automatically use a time-harmonic circuit implementation. If you select the **Include model parameters** check box all parameters to any device models are included to the global expressions

as individual variables. With the check box cleared the parameters are included as numbers to reduce the number of variables. The model parameters are necessary to include if you, for example, want to do a parameter extraction or if you just want to be able to change their values in a parameter ramp.

When you click any of the toolbar buttons, a dialog box appears where you enter the name of the device, the terminal names (node names) that it connects to as space-separated entries, and finally a device value.
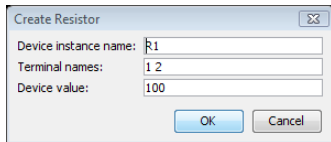


For subcircuit definitions of links to a COMSOL Multiphysics model, the dialog box is slightly different. Here you enter a subcircuit reference name that you use to refer to this subcircuit. The terminal names are the circuit terminal indexes or variables defined in the COMSOL Multiphysics model.
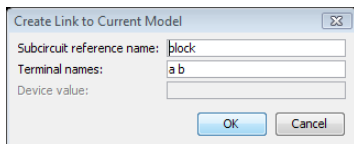


There are two toolbar buttons to create links to a COMSOL Multiphysics model. The **Create Link to Current Model** button connects the circuit to the model present in the main window. The **Create Link to Model File** button connects the circuit to a model file saved on disk.

These two buttons only generate a subcircuit definition of the link, exactly like a standard SPICE subcircuit definition. In order to use the link in a circuit you must also add a subcircuit instantiation command. You can do this with the **Create Subcircuit Instance** toolbar button. In the dialog box you enter the name of the instance, the nodes in the circuit that the subcircuit nodes are connected to, and finally the subcircuit reference name defining the link. If the subcircuit definition links to a model file, creating the link also appends new geometries to the current model.

## Supported SPICE Functionality

Currently the SPICE import supports the devices summarized in Table 2-1 with mentioned limitations.

TABLE 2-1:  SUMMARY OF SUPPORTED DEVICES

| SYMBOL | DESCRIPTION | LIMITATION |
|---|---|---|
| R | Resistor | No temperature dependence |
| C | Capacitor | No voltage and temperature dependence |
| L | Inductor | No current and temperature dependence |
| K | Mutual coupling | Only simple coupling between two inductors. No non-linear core models supported. |
| V | Independent voltage source | Supports constant sources, pulse sources, and sine sources. Variable names can be used to implement arbitrary expressions by adding them later as global expressions |
| I | Independent current source | See above |
| E | Voltage-controlled voltage source | Gain-controlled source |
| F | Current-controlled current source | See E device |
| G | Voltage-controlled current source | See E device |
| H | Current-controlled voltage source | See E device |
| D | Diode | No temperature dependence other than the diode equation |
| Q | Bipolar transistor | Implements parts of the Gummel-Poon transistor model of NPN type. No small-signal model available. |
| M | MOS transistor | Implements the MOS transistor model as defined by Shichman and Hodges of N-type. No small-signal model available. |
| X | Subcircuit | Instantiate a subcircuit |

The SPICE import also supports some commands in the netlist file, listed in Table 2-2.

TABLE 2-2: SUPPORTED COMMANDS IN THE NETLIST

| COMMAND | DESCRIPTION |
| --- | --- |
| .LIB "\<library>" | Loads the library within quotes |
| .INC "\<file>" | Includes the file within quotes |
| .MODEL | Creates a device model with user-defined parameter values |
| .SUBCKT | Creates a subcircuit |
| .TEMP | Sets the global temperature |

All other commands and unsupported model parameters are ignored.

### INSTANCES OF COMSOL MULTIPHYSICS MODELS

The `.SUBCKT` command also handles instances of COMSOL Multiphysics model files. The extra option `COMSOL:` defines the model filename or application mode to link to. An asterisk (`*`) here automatically searches in the current model for the first geometry or application mode with proper terminal definitions, which either can be circuit terminal boundary conditions or global variables. Note that any ground boundary condition in the COMSOL Multiphysics model file using the circuit terminal method gets directly connected to the ground node 0 of the circuit.

*Circuit Ports*

Application modes with the circuit port boundary condition can handle the connection automatically in the model. In the **Boundary Settings** dialog box, you select the circuit port boundary condition for those boundaries you want to connect the circuit to. The ports are separated by different port numbers that you enter in the **Port number** edit field present for all circuit port boundaries. The following netlist shows an example:

```
Vin 0 1 1V
R1 0 2 10k
R2 0 310k
X1 1 2 mph

.SUBCKT mph a COMSOL: *
.ENDS
```

The subcircuit command searches the current model for circuit port conditions with port number `a`. The boundaries with index `a` gets connected between node 1 and 2 in the circuit. If you wish to use several port connections, just add one subcircuit definition per port.

**Note:** The circuit port boundary condition is actually identical to the lumped port boundary condition, but without possibility to specify a voltage source. Because the lumped port models a connection between two terminals, it is necessary to use two nodes in the circuit netlist and one in the subcircuit definition.

*Terminal Variables*

Application modes without the circuit terminal boundary conditions need two variables: one for device voltage and one for device current. The names of these variables are passed to the `.SUBCKT` command after the name of the device. If there are more than two terminals in the model file, you must define one subcircuit for each voltage-current pair. The following netlist shows a simple example when using terminal variables:

```
Vin 0 2 1V
R1 2 1 10k
X1 1 0 Rfem
.SUBCKT Rfem V_res I_res COMSOL: Rmodel.mph
.ENDS
```

This circuit connects the finite element model in the file `Rmodel.mph` to a circuit with a voltage source `Vin` and a resistor `R1` through the instance `X1`. The model file `Rmodel.mph` has two variables, `V_res` and `I_res`, defined in the global scope. The variable `I_res` must be a variable defined in the **Global Variables** dialog box, because it is altered by the SPICE Import. The instance `X1` is connected to nodes 1 and 0, and the voltage between these nodes is made equal to `V_res`, by changing the current `I_res`. `I_res` controls the current from node 1 to node 0.

If the second variable begins with one of the letters `V`, `v`, `U`, or `u`, COMSOL Multiphysics defines the circuit so it controls the voltage and reads the current. It is always the second variable that has to be present in the **Global Variables** dialog box, which in this case is interpreted as a voltage. This approach can generate less ODE variables for models where it is easier to give the voltage and calculate the current.

**Note:** The number of instances of a model file that you can use in a circuit is limited to one for the terminal variable approach. Using additional instances causes variable-name conflicts. The circuit terminal approach can use several instances.

### LIMITATIONS OF THE SPICE IMPORT

Not all device models are implemented, and those implemented do not support the full range of parameters available. Table 2-1 lists the supported devices. You might therefore have to simplify the netlist prior to the import. In addition, support for model parameters specifying a temperature dependence is not implemented. However, the temperature affects the leakage current on all modeled p-n junctions according to the diode equation.

The transit-time capacitance is not supported for the semiconductor device models where it is used.

All statements for specifying input signals are not supported. Currently the supported statements are SIN and PULSE. The following table summarizes the syntax of these two signal specifications.

| ARGUMENT | SIN | PULSE |
|---|---|---|
| 1 | DC level | Initial value |
| 2 | Amplitude | Pulsed value |
| 3 | Frequency | Delay |
| 4 | Delay | Fall time |
| 5 | Decay factor | Rise time |
| 6 | Phase | Pulse width |
| 7 | N/A | Period |

The SPICE import does not support small-signal analysis of semiconductor device models. In a time-harmonic simulation you can only connect sources and passive devices.

### SCRIPT SUPPORT

You can also access the functionality described in the preceding section from MATLAB using the `spiceimport` command; see Chapter 4, "Function Reference." of the *RF Module Reference Guide*.

### ITERATIVE SOLVER SETTINGS

When coupling a 3D finite element model to an external circuit it can be quite difficult to find proper solver settings that do not consume too much memory. Most external circuits are coupled directly to an application mode that solves for the electrostatic potential. A simple approach can be to use the segregated solver and put the external circuit solution variables with the solution variable for the electrostatic potential in one

segregated group. The rest of the solution variables are added to one or more other groups.

If you want to try to solve a finite element problem with an external circuit using an iterative solver, you probably need to use the Vanka solver as preconditioner or smoother. Then put all the solution variables for the external circuit as Vanka variables. Because all circuit variables from the SPICE Import begins with `sim_`, you can use wild cards to add them all to the Vanka variable list. To do this, just type `sim_*` in the **Variables** edit field in the **Linear System Solver Settings** dialog box; see the section "Solving Large 3D Problems" on page 92 for more information about using the Vanka solver.

## *Example Model using SPICE Import*

The model "Monoconical RF Antenna" on page 65 of the *RF Module Model Library* contains a transient analysis involving an external circuit, which is added to the model using the SPICE circuit editor.

## *Reference*

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

# Solving Large 3D Problems

For large 3D electromagnetic problems, where the default direct linear solver requires too much memory, you can use the iterative linear solver GMRES together with the geometric multigrid (GMG) preconditioner for wave problems. This section describes how to use the set up the iterative solvers in the context of electromagnetic wave simulations. You can find general information about the geometric multigrid solver in the section "The Geometric Multigrid Solver/Preconditioner" on page 557 in the *COMSOL Multiphysics Reference Guide*.

## *Hierarchy Generation*

There are several types of hierarchy used by the geometric multigrid preconditioner. The default is to use the element order, where the coarsest mesh usually is the $1^{st}$ order element. The finer hierarchy can then be either quadratic or cubic vector elements. It can also use a hierarchy of meshes, where the finer meshes in the hierarchy have to be obtained by refining the coarsest mesh. It is also possible to manually make all the meshes using the **Mesh Cases** dialog box and tell the preconditioner to use these meshes; see the section "Hierarchy Generation Method" on page 93.

**Note:** The solver calculates the solution for the finest of the meshes in the hierarchy. Therefore, when you let the preconditioner generate the meshes automatically, you obtain a solution for a much finer mesh than the one you have made.

### CONSTRAINTS ON THE COARSEST MESH

The coarsest mesh cannot be arbitrarily coarse, because then the iterative solver does not converge. For wave problems, for instance, the minimum requirement is that there are at least two mesh elements per wavelength in the geometry. This is the *Nyquist criterion*. Keep in mind that the wavelength varies with the material properties and that the mesh has to resolve the waves in the whole geometry.

A direct solver is used to solve the equation on the coarsest mesh. Therefore you should try to make the mesh as coarse as possible while still fulfilling the Nyquist criterion of two mesh elements per wavelength.

## Solver Settings

Electromagnetic wave problems depend on the geometric multigrid solver (used as preconditioner), and this solver relies on a hierarchy of meshes. The next section describes the options you have when you create the hierarchy, because this is crucial for a converging solution in most cases.

### HIERARCHY GENERATION METHOD

There are several hierarchy generation methods that you can use with the geometric multigrid solver, and not all are suitable for electromagnetic problems. You select them from the **Hierarchy generation method** list in the **Linear System Solver Settings** dialog box. The default is **Lower element order first**. This option finds the lower-order elements and use them as coarser levels. Another option is **Refine mesh**, which automatically generates the finer meshes in the hierarchy. The **Manual** option lets you pick a set of meshes that you have created beforehand. If you solve with the **Refine mesh** option, the solve step stores the generated meshes and automatically switches to the **Manual** option.
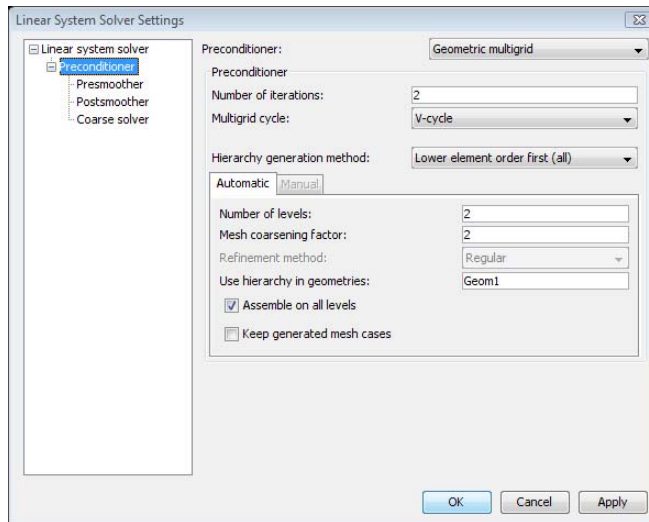


*Figure 2-10: The multigrid settings for the GMG preconditioner.*

### Lower Element Order

The use of higher-order vector elements opens up the possibility to use different element orders in the hierarchy. You have to use a vector element order of at least 2. Specify the element order either at model creation in the **Model Navigator** or in the

**Subdomain Settings** dialog box under the **Element** tab. The default vector element order is 2, so the default hierarchy is first order on the coarsest level and second order on the finest level. This is a robust setting that often produce convergence. Another efficient option is to use third order as the finest level and first order on the coarsest level. The main drawback with third-order vector elements is that they produce around 20 degrees of freedoms (DOF) per mesh element, which sometimes makes it hard to resolve the geometry and keep the number of DOF at a reasonable level. The same figure for second order is around 6.5 DOF per mesh element. Generally, it is best to use as high order as you can because the overall error is often lower for higher-order elements with the same number of DOF.

*Refine Mesh*
The refine mesh option automatically generates finer meshes. The settings on the **Automatic** page determines how this refinement is done.

- **Number of levels** determines how many meshes the mesh refinement generates, that is, how many times the original mesh is refined. This number is the total number of meshes including the original mesh. For example, the default number 2 gives you one refinement.

- The **Refinement method** specifies how the mesh is refined. See the section "Refinement Methods" on page 344 in the *COMSOL Multiphysics User's Guide* for general information about the refinement methods. **Regular** refinement creates finer meshes than the **Longest** method.

These two parameters often have to be tuned given how coarse the original mesh is. If the original mesh just fulfills the Nyquist criterion of two mesh elements per wavelength, then two regular refinements are needed to obtain an accurate solution. If, on the other hand, the original mesh is finer, for example, because the geometry makes it difficult to make it coarser, then two regular refinements might give a too fine mesh, making the solvers use more memory than necessary. If so, one regular refinement can be sufficient or the longest method might be a better choice.

*Manual*
Manual hierarchy generation lets you pick meshes that you have made manually. This gives you better control of which meshes the solver uses. To create a set of meshes use the **Mesh Cases** dialog box in the **Mesh** menu and create as many mesh cases as the number of meshes you need. Then switch between the mesh cases by selecting them in the **Mesh** menu and mesh the geometry for each mesh case. Make the mesh of each mesh case finer than the previous one. Alternatively, you can select the element order

manually in the **Subdomain Settings** for each mesh case, and use the same mesh for both mesh cases.
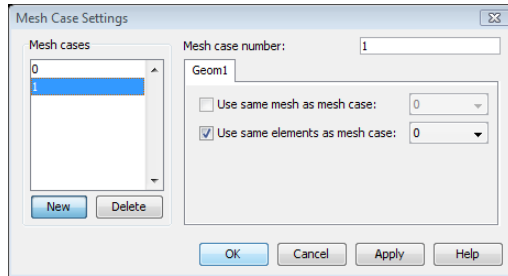


*Figure 2-11: Settings for using the same mesh for both mesh cases.*

---

**Note:** The meshes must be made by refining the coarsest one if you use the same element order.

---

On the **Manual** page of the **Linear System Solver Settings** dialog box, select the meshes you want to use. Make sure **Assemble** is selected for all meshes.

### SOLVER SETTINGS FOR ELECTROMAGNETIC WAVES

In most cases it is sufficient to use the default setting, which is **GMRES** for the **Linear system solver** list and **Geometric multigrid** for the **Preconditioner** list. In other cases it is necessary to adjust how the solver generates the mesh hierarchy or fine tune the solver settings.
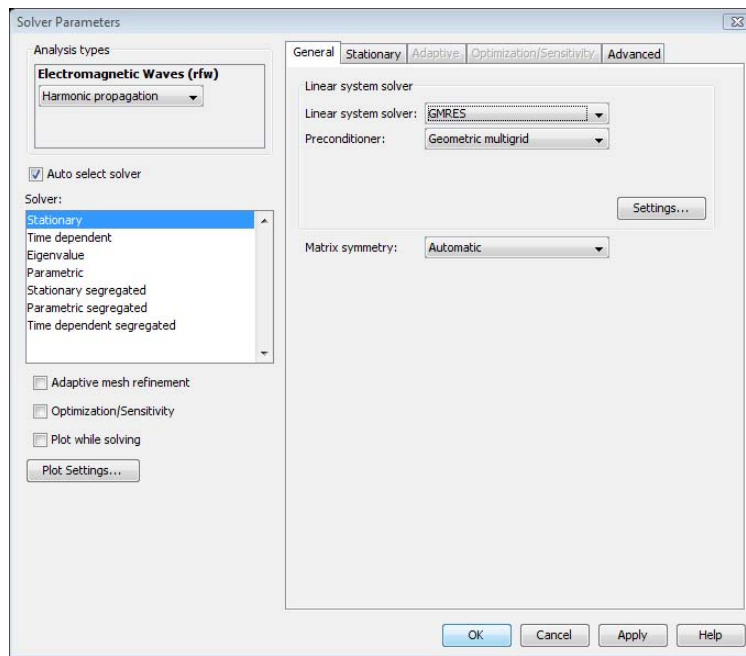
*Figure 2-12: The default settings for electromagnetic waves.*

### Presmoother

The default presmoother is **SOR vector**.
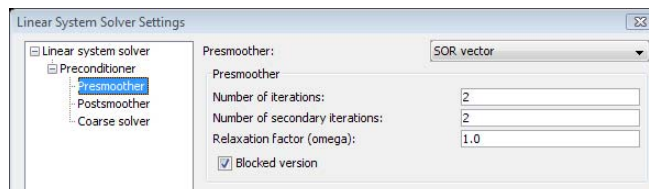


*Figure 2-13: The default presmoother settings (figure shows part of dialog box).*

*Postsmoother*

The default postsmoother is **SORU vector**.
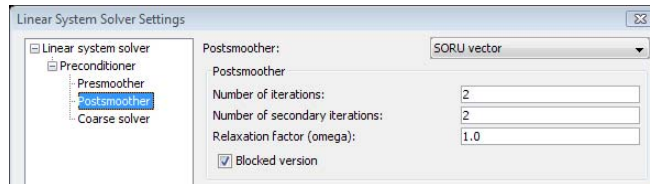


*Figure 2-14: The default postsmoother settings (figure shows part of dialog box).*

*Coarse Solver*

The default coarse solver is **PARDISO**.
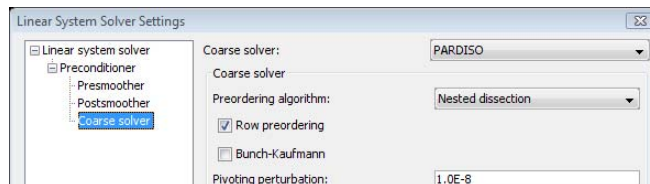


*Figure 2-15: The default coarse solver settings (figure shows part of dialog box).*

---

**Note:** When the 3D Electromagnetic Waves application mode is the single application mode in the model, it automatically adjusts the default values of the solver settings described above to values which are suitable for the calculation. If the 3D Electromagnetic Waves application mode is part of a multiphysics model the default settings might not be suitable for waves. In that case, make sure you adjust all settings as described in the sections above. Models involving periodic 3D wave problems also require manual adjustment of the solver settings as described in the following section.

---

SOLVER SETTINGS FOR PERIODIC 3D WAVE PROBLEMS

If you use periodic boundary conditions in 3D, you cannot use the settings described above due to the added divergence condition (see "Periodic Boundary Conditions" on page 37). This section also apply to problems using assembly interfaces with incompatible meshes (see "Using Assemblies in Electromagnetic Problems" on page 107). To set up an iterative solver for such problems, you must use geometric multigrid with the Vanka preconditioner. Select **Geometric multigrid** in the **Linear system solver** list, and set the presmoother and postsmoother to **Vanka**, which is configured with suitable defaults for large 3D problems. In some cases it is necessary to adjust how the

solver generates the mesh hierarchy and fine tune the settings for the Vanka preconditioner.
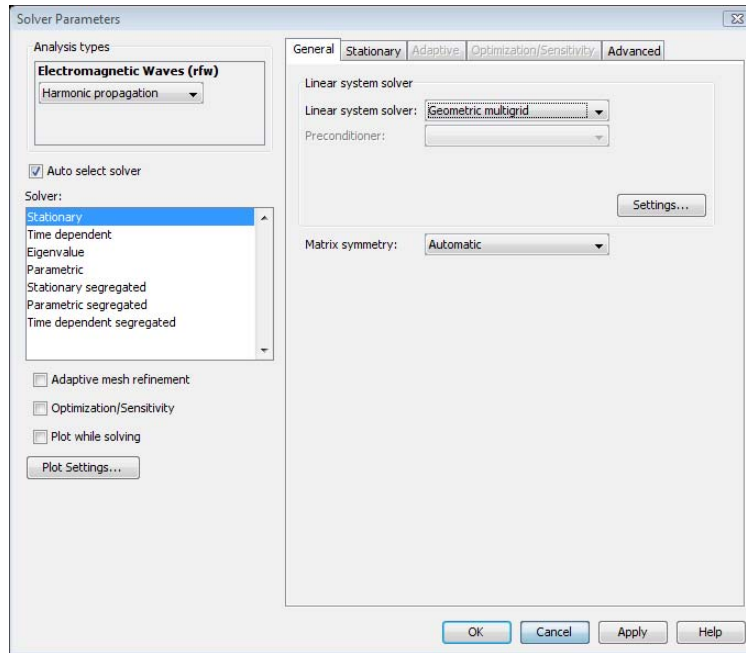


*Figure 2-16: Solver settings for periodic 3D wave problems.*

For geometric multigrid with Vanka the preferred setting for the **Multigrid cycle** is **F-cycle** in contrast to wave problems where it is **V-cycle**.
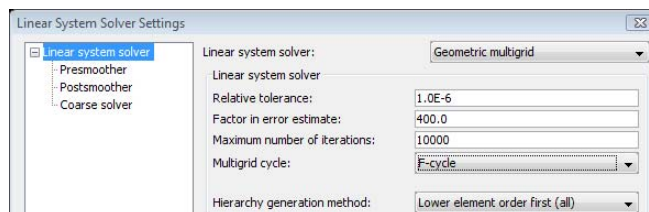


*Figure 2-17: The multigrid settings for periodic 3D wave problems (the figure shows only the upper part of the dialog box).*

As an alternative to the geometric multigrid solver you can also use the GMRES linear system solver. As preconditioner you can either choose Vanka or geometric multigrid with Vanka as pre- and postsmoother. The default settings are the same as those described below.

*Presmoother*

The presmoother must be set to **Vanka**. The dialog box shows the default settings.
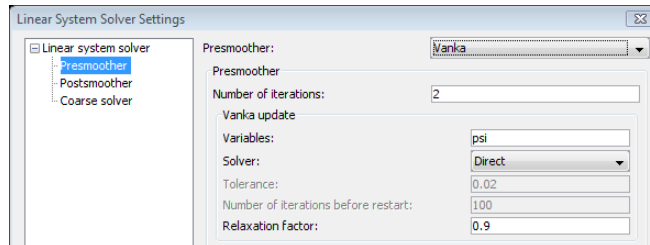


*Figure 2-18: The default presmoother settings (the figure shows only the upper part of the dialog box).*

The default setting for **Variables** is `psi`, which is the default variable name used by the divergence condition. This default setting does not change if you specify a different name for that variable, and then you have to update this setting manually for both the presmoother and the postsmoother.

*Postsmoother*

The postsmoother must be set to **Vanka**. The default settings are the same as for the presmoother.
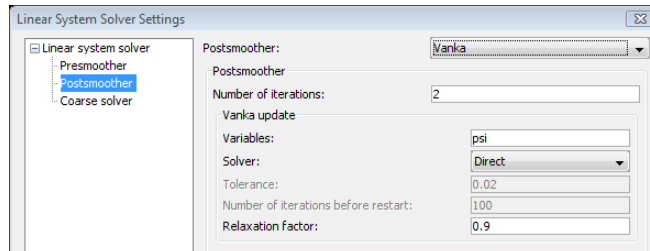


*Figure 2-19: The default postsmoother settings (the figure shows only the upper part of the dialog box).*

*Coarse Solver*

The preferred coarse solver is **PARDISO**.
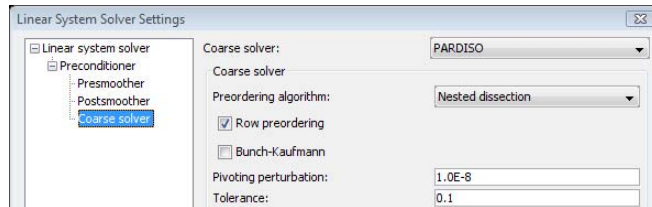


*Figure 2-20: The coarse solver settings (the figure shows only the upper part of the dialog box).*

## After Solving

When you have solved using automatic generation of the mesh hierarchy using the **Refine mesh** method, you can find the meshes as mesh cases in the **Mesh** menu. The finest mesh is the one that the solver used to calculate the final solution, and it shows how well the geometry has been resolved.

When you have solved using the refine mesh hierarchy, the **Hierarchy generation method** in the **Linear Solver Settings** dialog box switches to **Manual**. This means that if you solve again, the solver uses the same meshes.

If you want to use the refine mesh generation again but with different refinement parameters or another coarse mesh, you have to change the **Hierarchy generation method** back to **Refine mesh**. Then another set of meshes is generated. Using the **Mesh Cases** dialog box, you can delete the additional meshes that were used for the previous solution but are no longer used.

If you solve using the **Lower element order first** method, the hierarchy is generated on the current mesh each time you solve, so you do not have to adjust any solver parameters.

# 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 analysis type is available for all application modes 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 |
|---|---|---|
| H | Nonzero conductivity | |
| H, E | Nonzero conductivity | Impedance boundary condition |
| H, E | Nonzero conductivity at adjacent subdomain | Scattering boundary condition |
| H, E | Nonzero conductivity at adjacent subdomain | Scattering boundary condition |
| H, E | Analytical ports | Port boundary condition |

These situations require special treatment, especially since it can lead to singular matrix 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

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

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 the "The Eigenvalue Solver" on page 402 of the *COMSOL Multiphysics User's Guide* for more information). 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 **Solver Parameters** dialog box, under the **Eigenfrequency** tab, by entering a new complex value in the **Eigenvalue linearization point** edit 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.
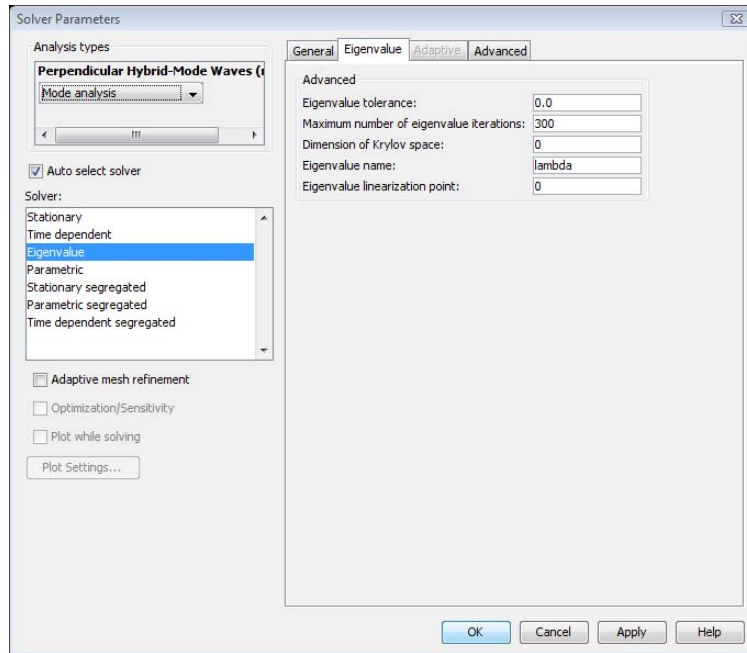


In many cases it is enough to specify a good linearization point and then solve the problem once. If you need a more accurate eigenvalue, an iterative scheme is necessary:

1 Specify that the eigenvalue solver only search for one eigenvalue.

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 application modes. 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, $\delta_z$, 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 |

*Example Model—Circular Waveguide with Lossy Walls*

This model finds the modes of a circular waveguide with walls made of a nonperfect conductor, which is copper in this case. The losses in the walls lead to attenuation of the propagating wave. The attenuation in dB scale is calculated using the formula

$$\Delta_{\mathrm{dB}} = 20\delta_z \log e$$

which is available as the variable dampzdB_rfwv. The first six eigenvalues show the following attenuation per km for a propagating wave with a frequency of $1$ GHz.

| EFFECTIVE MODE INDEX | PROPAGATION CONSTANT | ATTENUATION |
|----------------------|----------------------|-------------|
| 0.931 + 1.9e-6i | 19.51 rad/m | 0.34 dB/km |
| 0.957 + 1.1e-6i | 20.05 rad/m | 0.20 dB/km |
| 0.957 + 1.2e-6i | 20.05 rad/m | 0.22 dB/km |
| 0.973 + 1.3e-6i | 20.40 rad/m | 0.24 dB/km |
| 0.984 + 4.4e-7i | 20.63 rad/m | 0.08 dB/km |
| 0.984 + 7.6e-007i | 20.63 rad/m | 0.14 dB/km |

**Model Library path:** `RF_Module/Tutorial_Models/`
`lossy_circular_waveguide`

## *Modeling Using the Graphical User Interface*

### MODEL NAVIGATOR

**1** In the **Model Navigator** select **2D** from the **Space dimension** list.

**2** Open the **RF Module** folder, then select **Perpendicular Waves>**
**Hybrid-Mode Waves>Mode analysis**.

**3** Click **OK**.

### GEOMETRY MODELING

Use **Specify Objects>Circle** under the **Draw** menu to create a circle with radius `0.5`.

### PHYSICS SETTINGS

*Subdomain Settings*
Use the default settings for the subdomain.

*Boundary Conditions*

**1** Open the **Boundary Settings** dialog box from the **Physics** menu.

**2** Select all boundaries and choose the **Impedance boundary condition** from the
**Boundary condition** list.

**3** Click the **Material Properties** tab, click the **Load** button, and select **Copper** from the
list of materials in the dialog box that appears.

**4** Click **OK** twice to close the **Materials/Coefficients Library** dialog box and the **Boundary**
**Settings** dialog box in that order.

### COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to solve the problem with the default mesh
and solver settings.

The default plot shows the *z*-component of the time-averaged power flow for the first eigenmode in the table on page 104.



neff_rfwv(1)=0.930755-2.208024e-6i
Surface: Power flow, time average, z component [W/m$^2$]
Max: 4.59e5
Min: 13.189
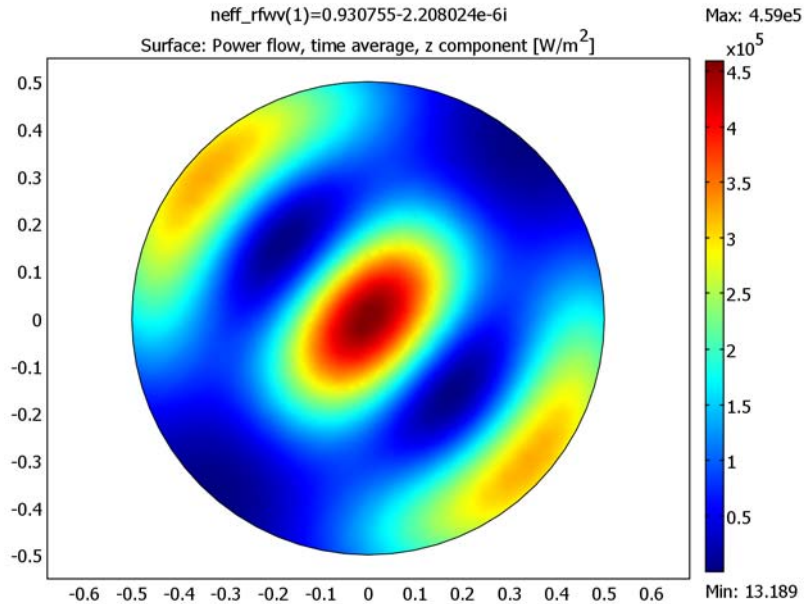
To calculate the attenuation values in the third column of the table, do as follows:

**1** Open the **Global Data Display** dialog box by choosing **Data Display>Global** from the **Postprocessing** menu.

**2** Enter `1000*dampzdB_rfwv` in the **Expression** edit field and click **Apply**.

This produces the first value for the third column in the table on page 104.

**3** Go through the other entries in the **Effective mode index** list to get the full table.

# Using Assemblies in Electromagnetic Problems

The use of assemblies in models can simplify and improve several aspects of modeling. For electromagnetic applications it can be used to make devices move and rotate with the *sliding mesh* technique. Another situation when assemblies come in handy is when you need to introduce a discontinuity for the solution variable at an interface. Such interface conditions are generally called slit boundary conditions. For example, an internal perfect electrical conductor when solving for the magnetic field needs a slit boundary condition because the solution variable becomes discontinuous across that interface. The sliding mesh interface can also be seen as a slit boundary. A third example of assembly usage is when you want to mix mesh element types and mesh resolution, because you can mesh the two neighboring parts of the assembly independently. You can mix element without using assemblies as long as the boundary mesh elements at an interface are the same. This is true for interfaces between quadrilateral and triangular elements in 2D and for tetrahedral elements facing the triangular bases of the prism elements in 3D.

Although assemblies simplifies several task in modeling, there are a few important rules and recommendations that need consideration, and these are covered in the following sections.

### ASSEMBLIES AND VECTOR ELEMENTS

If you use an application mode with vector elements it might be necessary to activate the divergence condition. This is the same type of divergence condition that you need when solving with periodic boundary condition (see "Periodic Boundary Conditions" on page 37). The only situation where you do not need the divergence condition is when the boundary meshes on each side of the assembly interface are identical.

For 2D problems, for which you can use direct solvers, activating the divergence condition usually does not cause any trouble. It is when you solve 3D problems with iterative solvers that you need to know wether the divergence condition is activated or not. When it is activated there is an extra solution variable, `psi`, added to the problem that needs the Vanka preconditioner/solver with `psi` as Vanka variable. You can find details about these solver settings in "Solving Large 3D Problems" on page 92.

The following list summarizes the application modes in the RF Module that use vector elements:

• 3D electromagnetic waves.

• Transient in-plane waves for TM and hybrid-mode polarizations.

• Perpendicular hybrid-mode waves. This application mode does not support the use of a divergence condition.

Generally, if it is possible to avoid activating the divergence condition you should do that. The problem is much tougher to solve when it is activated in terms of solution time and memory usage. There are techniques to create compatible (or identical) meshes, and these are briefly described below.

### Compatible Meshes

Use interactive meshing and mesh only the source or destination side of pair boundaries first. Then select both source and destination boundaries and click the **Copy Mesh** button on the Mesh toolbar. You can read more about the copy mesh feature in "Copying Meshes" on page 360 of the *COMSOL Multiphysics User's Guide*.

---

**Note:** For a sliding mesh interface it is not possible to generate compatible meshes. A mesh that is compatible at one time is probably not compatible at a later time when the destination side has moved slightly.

---

### ASSEMBLIES AND WEAK CONSTRAINTS

It might be necessary to use weak constraints to improve the accuracy at the assembly interface. Especially when the interface is a sliding mesh interface. Weak constraints also add extra complexity for the settings of the iterative solvers, so it might not be possible to use them for large 3D problems. Below are a few recommendations when to use and how to use weak constraints for assemblies:

• If you have a transient problem with a moving assembly interface that fails to converge after some time, you might get better convergence if you turn on weak constraints.

• The recommendation is generally to mesh the destination boundary finer than the source boundary, but it is not as crucial when you use weak constraints.

• When you turn on weak constraints, it is activated for all boundaries with constraints on the solution variable, and for all boundaries with coupling through assembly

pairs (for example, identity pairs). The difference in magnitude of the Lagrange multiplier between the assembly pairs and other constraints can be several orders of magnitude, which can cause convergence problems and even a singular matrix error. If you get such problems, it is recommended to deactivate the weak constraints manually for all other constraints except the assembly pair slave boundaries. Note that you should also deactivate the weak constraints for the master side. You do this by clearing the **Use weak constraints** check box on the **Weak Constr.** tab in the **Boundary Settings** dialog box.

### EXAMPLE MODEL

A model that uses assemblies with vector elements and identical boundary meshes is the "Coaxial to Waveguide Coupling" on page 203 of the *RF Module Model Library.*

# 3

# Review of Electromagnetics

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

# 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* **E**, the *electric displacement* or *electric flux density* **D**, the *magnetic field intensity* **H**, the *magnetic flux density* **B**, the *current density* **J** and the *electric charge density* $\rho$.

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 $\varepsilon_0$ is the *permittivity of vacuum*, $\mu_0$ is the *permeability of vacuum*, and $\sigma$ 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 $\chi_e$ is the *electric susceptibility*. Similarly in linear materials, the magnetization is directly proportional to the magnetic field, $\mathbf{M} = \chi_m \mathbf{H}$, where $\chi_m$ 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 $\varepsilon_r$ is the *relative permittivity* and $\mu_r$ 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 $\varepsilon$ and $\mu$ (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 $\mathbf{D}_r$ is the *remanent displacement*, which is the displacement when no electric field is present.

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

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

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

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

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

## Potentials

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

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

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

$$\mathbf{H} = -\nabla V_m$$

## Electromagnetic Energy

The electric and magnetic energies are defined as

$$W_e = \int_V \left(\int_0^D \mathbf{E} \cdot d\mathbf{D}\right)dV = \int_V \left(\int_0^T \mathbf{E} \cdot \frac{\partial\mathbf{D}}{\partial t}dt\right)dV$$

$$W_m = \int_V \left(\int_0^B \mathbf{H} \cdot d\mathbf{B}\right)dV = \int_V \left(\int_0^T \mathbf{H} \cdot \frac{\partial\mathbf{B}}{\partial t}dt\right)dV$$

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 [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

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

where $\rho_s$ 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.

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

## Phasors

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

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

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

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

The field $\tilde{\mathbf{E}}(\mathbf{r})$ is a *phasor*, 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 postprocessing 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 $\mathrm{Re}(\mathbf{E}(\mathbf{r}))$ by default, which is $\mathbf{E}$ at time $t = 0$. To obtain the solution at a given time, you can specify a phase factor in all postprocessing dialog boxes and in the corresponding functions. For more details about phase factors, see "The Phasor Variable" on page 182 in the *COMSOL Multiphysics User's Guide*.

# 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 $\mathbf{E}_1, \mathbf{E}_2, \mathbf{E}_3$, ... 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

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

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

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

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

It is also possible to calculate the S-parameters from 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 advantage of using the power

flow to calculate the S-parameters is that you do not need to know the modes $\mathbf{E}_1$, $\mathbf{E}_2$, and $\mathbf{E}_3$. When the modes are degenerated, as is the case for circular ports, it is generally impossible to know the orientation of the wave. It is then impossible to find the correct superposition of the degenerated modes to match the calculated wave.

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

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

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

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

Both ways of normalizing the S-parameters are supported and the choice is made from the **Port** page in the **Boundary Settings** dialog box; see "The Coefficient Page" on page 58.

### 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

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

$\omega$ is the angular frequency of the wave, $\mu$ the permeability, and $\beta$ the propagation constant. The power flow then becomes

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

*TM Waves*
For TM waves it holds that

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

where $Z_{TM}$ is the wave impedance

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

and $\varepsilon$ is the permittivity. The power flow then becomes

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

$$= \frac{1}{2Z_{TM}}|\mathbf{n} \times \mathbf{E}|^2$$

*TEM Waves*
For TEM waves it holds that

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

where $Z_{TEM}$ is the wave impedance

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

The power flow then becomes

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

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 $V$.



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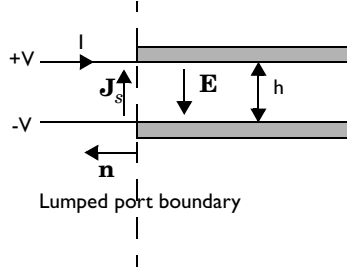
The transmission line current can be represented with a surface current at the lumped port boundary directed opposite to the electric field.

The impedance of a transmission line is defined as

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

and in analogy to this you can define a 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.

## Far-Field Calculations

The far electromagnetic field from, for example, antennas can be calculated from the near field using the Stratton-Chu formula,

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

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 $(\theta, \varphi)$.

In the above formula,

• $\mathbf{E}$ 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 $S$.

- $\eta_0$ is the free-space impedance, $\eta_0 = \sqrt{\mu_0/\varepsilon_0}$.

- $k_0$ is the free-space wave number.

- $\mathbf{r}$ 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 $(\theta, \varphi)$, 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. Although COMSOL Multiphysics supports other unit systems, the equations in the RF Module are written for SI units. 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 $\mu$m as the length scale, you need to have $\varepsilon_0 = 8.854187817 \cdot 10^{-18}$ F/$\mu$m and $\mu_0 = 4\pi \cdot 10^{-13}$ H/$\mu$m.

TABLE 3-1:  ELECTROMAGNETIC QUANTITIES

| QUANTITY | SYMBOL | UNIT | ABBREVIATION |
|---|---|---|---|
| Angular frequency | $\omega$ | radian/second | rad/s |
| Attenuation constant | $\alpha$ | $\text{meter}^{-1}$ | $\text{m}^{-1}$ |
| Capacitance | $C$ | farad | F |
| Charge | $q$ | coulomb | C |
| Charge density (surface) | $\rho_s$ | $\text{coulomb/meter}^2$ | $\text{C/m}^2$ |
| Charge density (volume) | $\rho$ | $\text{coulomb/meter}^3$ | $\text{C/m}^3$ |
| Current | $I$ | ampere | A |
| Current density (surface) | $\mathbf{J}_s$ | ampere/meter | A/m |
| Current density (volume) | $\mathbf{J}$ | $\text{ampere/meter}^2$ | $\text{A/m}^2$ |
| Electric displacement | $\mathbf{D}$ | $\text{coulomb/meter}^2$ | $\text{C/m}^2$ |
| Electric field | $\mathbf{E}$ | volt/meter | V/m |
| Electric potential | $V$ | volt | V |
| Electric susceptibility | $\chi_e$ | (dimensionless) | – |
| Electric conductivity | $\sigma$ | siemens/meter | S/m |
| Energy density | $W$ | $\text{joule/meter}^3$ | $\text{J/m}^3$ |
| Force | $\mathbf{F}$ | newton | N |
| Frequency | $\nu$ | hertz | Hz |
| Impedance | $Z, \eta$ | ohm | $\Omega$ |
| Inductance | $L$ | henry | H |

TABLE 3-1: ELECTROMAGNETIC QUANTITIES

| QUANTITY | SYMBOL | UNIT | ABBREVIATION |
|---|---|---|---|
| Magnetic field | $\mathbf{H}$ | ampere/meter | A/m |
| Magnetic flux | $\Phi$ | weber | Wb |
| Magnetic flux density | $\mathbf{B}$ | tesla | T |
| Magnetic potential (scalar) | $V_m$ | ampere | A |
| Magnetic potential (vector) | $\mathbf{A}$ | weber/meter | Wb/m |
| Magnetic susceptibility | $\chi_m$ | (dimensionless) | – |
| Magnetization | $\mathbf{M}$ | ampere/meter | A/m |
| Permeability | $\mu$ | henry/meter | H/m |
| Permittivity | $\varepsilon$ | 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 | $\beta$ | radian/meter | rad/m |
| Reactance | $X$ | ohm | $\Omega$ |
| Relative permeability | $\mu_r$ | (dimensionless) | – |
| Relative permittivity | $\varepsilon_r$ | (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 | $\lambda$ | meter | m |
| Wave number | $k$ | radian/meter | rad/m |

# Bibliography

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 Application Modes

This chapter reviews the application modes in the RF Module. For each application mode, you find information about its applications, fundamental equations, available boundary conditions, and other related information.

# The Application Mode Formulations

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

The application modes in the RF Module are listed below with the physical quantity solved for and the standard abbreviation that is used for the application modes in COMSOL Multiphysics. The physical quantities used are

- The *electric field*, **E**
- The *magnetic field*, **H**
- The magnetic potential, **A**

You can also use the application modes with MATLAB. See the chapter "Programming Reference" on page 27 in the *RF Module Reference Guide* for details.

## *Application Mode Guide*

Table 4-1 on page 131 lists the available application modes in the RF Module. For a descriptive illustration and more details on each of these modes, see the corresponding section in the table's **Page** column.

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

The **Dependent Variables** column contains the variables that the PDEs are formulated for. For most 2D modes, the PDEs solved in the simulations are formulated for the components that are perpendicular to the modeling plane. For axisymmetric simulations, COMSOL Multiphysics makes a variable transformation to avoid singularities at the rotation axis.

The **Field Components** columns list the nonzero field components. In the application modes using Cartesian coordinates, the components are indexed by $x$, $y$, or $z$; for cylindrical coordinates, $r$, $\varphi$, or $z$ are used.

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

TABLE 4-1: RF MODULE APPLICATION MODES

| APPLICATION MODE | PAGE | NAME | DEP. VAR. | FIELD COMPONENTS | | ANALYSIS CAPABILITIES | | |
|---|---|---|---|---|---|---|---|---|
| | | | | MAGNETIC FIELD | ELECTRIC FIELD | TRANSIENT | TIME-HARMONIC | EIGENFREQUENCY/EIGENMODE |
| **ELECTROMAGNETIC WAVES** | 136 | | | | | | | |
| 3D Electromagnetic Waves | 137 | rfw | $\mathbf{E}$ or $\mathbf{H}$ | $x\ y\ z$ | $x\ y\ z$ | √ | √ | √ |
| In-plane Waves, TE Modes | 148 | rfwe | $E_z$ | $x\ y$ | $z$ | √ | √ | √ |
| Axisymmetric Waves, TE Modes | 158 | rfwe | $E_\varphi$ | $r\ z$ | $\varphi$ | √ | √ | √ |
| In-Plane Waves, TM Modes | 148 | rfwh | $H_z$ | $z$ | $x\ y$ | √ | √ | √ |
| Axisymmetric Waves, TM Modes | 158 | rfwh | $H_\varphi$ | $\varphi$ | $r\ z$ | √ | √ | √ |
| In-Plane Waves, Hybrid Modes | 148 | rfweh | $E_z, H_z$ | $x\ y\ z$ | $x\ y\ z$ | √ | √ | √ |
| Axisymmetric Waves, Hybrid Modes | 158 | rfweh | $E_\varphi, H_\varphi$ | $r\ \varphi\ z$ | $r\ \varphi\ z$ | √ | √ | √ |
| Perpendicular Waves, TM Modes | 160 | rfwev | $E_z$ | $x\ y$ | $x\ y\ z$ | | | √ |
| Perpendicular Waves, TE Modes | 160 | rfwhv | $H_z$ | $x\ y\ z$ | $x\ y$ | | | √ |
| Perpendicular Waves, Hybrid Modes | 160 | rfwv | $H_x, H_y,$ or $\mathbf{H}$, or $E_x, E_y,$ or $\mathbf{E}$ | $x\ y\ z$ | $x\ y\ z$ | | | √ |
| 3D Boundary Mode Analysis, TE Waves | 169 | rfwb | $H_n$ | $x\ y\ z$ | $x\ y\ z$ | | | √ |

TABLE 4-1: RF MODULE APPLICATION MODES

| APPLICATION MODE | PAGE | NAME | DEP. VAR. | FIELD COMPONENTS | | ANALYSIS CAPABILITIES | | |
|---|---|---|---|---|---|---|---|---|
| | | | | MAGNETIC FIELD | ELECTRIC FIELD | TRANSIENT | TIME-HARMONIC | EIGENFREQUENCY/EIGENMODE |
| 3D Boundary Mode Analysis, TM Waves | 169 | rfwb | $E_n$ | $x\ y\ z$ | $x\ y\ z$ | | | √ |
| 3D Boundary Mode Analysis, Hybrid-Mode Waves | 169 | rfwb | $\mathbf{E}, E_n$ or $\mathbf{H}, H_n$ | $x\ y\ z$ | $x\ y\ z$ | | | √ |
| 2D Boundary Mode Analysis, TE Waves | 172 | rfwb | $H_z$ | $z$ | $x\ y$ | | | √ |
| 2D Boundary Mode Analysis, TM Waves | 172 | rfwb | $E_z$ | $x\ y$ | $z$ | | | √ |
| 2D Boundary Mode Analysis, TE Waves, Axisymmetry | 172 | rfwb | $E_\varphi$ | $r\ z$ | $\varphi$ | | | √ |
| 2D Boundary Mode Analysis, TM Waves, Axisymmetry | 172 | rfwb | $H_\varphi$ | $\varphi$ | $r\ z$ | | | √ |

To carry out different kinds of simulations for a given set of parameters in an application mode, you only have to change the *solver type* or specify the *analysis type*, which is an *application mode property*. The concept of application mode properties is introduced for setting up the coefficients in the underlying equations in a way that is consistent with the analysis carried out. The available analysis types are *transient*, *harmonic*, and *eigenfrequency/eigenmode*. Not all analysis types are available in all application modes.

You select the application mode from the Model Navigator when starting a new model. You can also add application modes to an existing model to create a multiphysics model.

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

You specify all scalar properties that are specific to the application mode in the **Application Scalar Variables** dialog box. Their default values are either physical constants or arbitrary values in a value range that is commonly used for modeling, for example, the frequency 1 GHz for time-harmonic propagation.

Enter the application-specific domain properties in the **Subdomain Settings** dialog box. It is possible to define subdomain parameters for problems with regions of different material properties. 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 the section "Modeling Anisotropic Materials" on page 220 of the *COMSOL Multiphysics User's Guide*.

The **Boundary Settings** dialog box also adapts to the current application mode and lets you select application-specific boundary conditions. A certain boundary type might require one or several fields to be specified, while others generate the boundary conditions without user-specified fields.

The **Edge Settings** and **Point Settings** dialog boxes similarly let you specify application-specific conditions on edges and points.

Finally, use the **Plot Parameters**, **Cross-Section Plot Parameters**, and **Domain Plot Parameters** dialog boxes to visualize the relevant physical variables for all application modes in the model. The nonzero components of the electromagnetic vector fields contain the name of the coordinate, for example, Aphi is the φ component of the magnetic vector potential.

The remainder of this chapter contains all the details necessary to get full insight into the different application modes, that is, the physical assumptions and mathematical considerations they are based on, and all the functionality that is available. Each section describing a particular mode is divided into the following sections:

In the *PDE Formulation* section, the equation or equations that are solved in the application mode are derived.

In the *Application Mode Properties* section you find properties that are specific for the application mode. These properties can for example be used to select the analysis type.

The *Application Scalar Variables* section lists the parameters that are specific for the application mode. Their default values are either physical constants or arbitrary values

in a value range that is commonly used for modeling, for example the frequency 1 GHz for harmonic propagation.

The *Boundary and Interface Conditions* section contains the available boundary conditions and explanations of their physical interpretation.

In the *Line Sources* and *Point Sources* sections you find the available settings on edges and points, respectively.

Information about the *Application Mode Variables* can be found in Chapter 2, "Application Mode Reference," of the *RF Module Reference Guide*. That section lists all variables that are available in postprocessing and when formulating the equations. You can use any function of these variables when postprocessing the result of the analysis. It is also possible to use these variables in the expressions for the physical properties in the equations.

**PREDEFINED APPLICATION MODE COMBINATIONS**

In the Model Navigator it is also possible to select a collection of application modes that contains predefined multiphysics couplings and other important combinations. They look like one application mode, but when you click **OK**, the program automatically adds the included application modes, setup predefined couplings, and provide suitable solver defaults. For example, a predefined coupling can be to include the average resistive heating variable, `Qav_rfw`, from Electromagnetic Waves into the **Heat source** edit field in the Heat transfer subdomain settings. This application mode combination is called Microwave heating. Below is a list of the different application mode combination present in the RF Module.

*3D*

- **Electro-Thermal Interaction>Microwave Heating**
  - Heat Transfer (or General Heat Transfer if the Heat Transfer Module is available)
  - Electromagnetic Waves

*2D*

- **Electro-Thermal Interaction>In-Plane Microwave Heating**
  - Heat Transfer (or General Heat Transfer if the Heat Transfer Module is available)
  - In-Plane Waves

*Axial Symmetry (2D)*

- **Electro-Thermal Interaction>Meridional Microwave Heating**

    - Heat Transfer (or General Heat Transfer if the Heat Transfer Module is available)

    - Meridional Waves

**MATERIAL LIBRARY**

All application modes 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 "Using the Materials/Coefficients Library" on page 228 in the *COMSOL Multiphysics User's Guide* for details about the materials libraries.

# Electromagnetic Waves

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

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

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

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.

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

Writing the fields on a time-harmonic form,

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

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

the two laws can be combined into either an equation for the electric field, or an equation for the magnetic field

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

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

where the second equation contains 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 $\varepsilon'$ is the real part of $\varepsilon$, and all losses are given by $\varepsilon''$.

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

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

## Refractive Index in Optics and Photonics

In optics and photonics applications, the refractive index is often used instead of the permittivity. In materials where $\mu_r$ 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 $\kappa$ represents a damping of the electromagnetic wave.

## 3D Electromagnetic Waves Application Mode

In the *3D Electromagnetic Waves* application mode, either of the two time-harmonic equations or the transient equation above can be solved. Both harmonic and transient

propagation problems and eigenfrequency cavity problems can be handled. In harmonic propagation problems and in eigenfrequency cavity problems the application mode lets you solve for either for the electric field or the magnetic field. Solve for the field that is of most interest to you. In transient propagation problems you solve for the magnetic vector potential.

### VECTOR ELEMENTS

In 3D, Lagrange elements cannot be used 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 3D Electromagnetic waves application mode 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.

### PDE FORMULATION

The governing equations can be written in the form

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

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

for the time-harmonic and eigenfrequency problems and in the form

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

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

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

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

$$\mu_0 \varepsilon_0 \frac{\partial}{\partial t}\left(n^2 \frac{\partial \mathbf{A}}{\partial t}\right) + \nabla \times (\nabla \times \mathbf{A}) = 0$$

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.

**DIVERGENCE CONDITION**

Taking the divergence of the two equations above gives the electric and magnetic forms of Gauss' law,

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

In the transient case and in most other cases it is sufficient that Gauss' law is satisfied implicitly from the Helmholtz equation, which the application mode solves. When using periodic boundary conditions, however, you must add Gauss' law explicitly as a separate equation to achieve numerical stability. To do this, set the **Divergence condition** property to **On** in the **Application Mode Properties** dialog box.

## APPLICATION MODE PROPERTIES

The application-mode specific properties are given in the table below.

| PROPERTY | VALUES | DESCRIPTION |
|---|---|---|
| Analysis type | Harmonic propagation<br>Eigenfrequency<br>Transient propagation | Specifies which type of analysis to perform. |
| Solve for | Electric field<br>Magnetic field<br>Scattered electric field<br>Scattered magnetic field | Specifies which field is the dependent variable. |
| Specify wave using | Frequency<br>Free-space wavelength | Specifies if the frequency or the wavelength should be used as input. |
| Specify eigenvalues using | Eigenvalue<br>Eigenfrequency | Specifies if the solver parameters should be given in terms of the eigenvalue $\lambda = j\omega + \delta$ or the eigenfrequency $\nu$. |
| Divergence condition | On<br>Off | Specifies if an explicit divergence condition $\nabla \cdot \mathbf{D} = 0$ or $\nabla \cdot \mathbf{B} = 0$ should be added as a separate equation. Such a condition is necessary when using periodic boundary conditions. |

## APPLICATION SCALAR VARIABLES

The application-specific variables in this mode are given in the following table:

| PROPERTY | NAME | DEFAULT | DESCRIPTION |
|---|---|---|---|
| $\mu_0$ | mu0 | 4*pi*1e-7 | Permeability of vacuum |
| $\varepsilon_0$ | epsilon0 | 8.854187817e-12 | Permittivity of vacuum |
| $\nu$ | nu | 1e9 | Frequency |
| $\lambda_0$ | lambda0 | 0.3 | Free-space wavelength |
| $E_{0ix}$ | E0ix | 0 | Incident electric field |
| $E_{0iy}$ | E0iy | 0 | Incident electric field |
| $E_{0iz}$ | E0iz | exp(-j*k*x) | Incident electric field |
| $H_{0ix}$ | H0ix | 0 | Incident magnetic field |

| PROPERTY | NAME | DEFAULT | DESCRIPTION |
|---|---|---|---|
| $H_{0iy}$ | H0iy | 0 | Incident magnetic field |
| $H_{0iz}$ | H0iz | exp(-j*k*x) | Incident magnetic field |

All values are expressed in SI units. When solving the problem as a harmonic problem, you must specify the frequency $\nu$ or the free-space wavelength $\lambda_0$. In the case of eigenfrequency simulations, the frequency is obtained from the solution of the problem. When solving for the scattered field the incident field needs to be specified to get correct equation and boundary conditions. When solving a harmonic problem for the total field, you can postprocess the scattered field if the incident field is given.

### EIGENFREQUENCY CALCULATIONS

When making eigenfrequency calculations, there are a few important things to note:

- For the **H**-formulation it is possible to have a nonzero conductivity although it leads to nonlinear eigenvalue problems due to the complex permittivity $\varepsilon_c = \varepsilon - j\sigma/\omega$ Such problems have to be treated specially. Using a complex value for the relative permittivity does not create this problem, however.

- Nonlinear eigenvalue problems also appear for impedance boundary conditions with nonzero conductivity and for scattering boundary conditions adjacent to subdomains with nonzero conductivity.

- 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 solver, it may 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 **Solver Parameters** dialog box. Adjust the settings so that the solver searches for eigenvalues closer to the lowest eigenfrequency than to zero. Another way to avoid the false eigenfrequencies is to set the **Divergence condition** to **On** in the **Application Mode Properties** dialog box. Explicitly imposing that the divergence of **D** or **B** is zero removes most of the false eigenfrequencies.

For more information and an example, see "Lossy Eigenvalue Calculations" on page 101.

### 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 $\mathbf{E}$, 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. When solving for $\mathbf{H}$, the tangential component of the magnetic field is always continuous, and thus the second condition is automatically fulfilled. The first condition is equivalent to the natural boundary condition

$$-\mathbf{n} \times [(\varepsilon_{rc}^{-1} \nabla \times \mathbf{H})_1 - (\varepsilon_{rc}^{-1} \nabla \times \mathbf{H})_2] = -\mathbf{n} \times j\omega\varepsilon_0(\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{0}$$

and is therefore also fulfilled. 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.

*Magnetic Field*

The magnetic field boundary condition

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

specifies the tangential component of the magnetic field. When solving for the magnetic field, you can use this boundary condition at exterior and interior

boundaries, but when solving for the electric field or the magnetic potential it is only applicable to exterior boundaries.

*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.

When solving for the magnetic field, surface currents cannot be specified on interior boundaries.

*Perfect Magnetic Conductor*
The perfect magnetic conductor boundary condition

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

is a special case of the above conditions that sets the tangential component of the magnetic field to zero.

*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. When solving for the electric field, this boundary condition is available at both exterior and interior boundaries, but when solving for the magnetic field it can only be applied to exterior boundaries. When solving for the magnetic potential this boundary condition is not available.

*Perfect Electric Conductor*
The perfect electric conductor boundary condition

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

is a special case of the above condition that sets the tangential component of the electric field to zero.

In the transient case this boundary condition sets the tangential component of the magnetic vector potential to zero. Because the perfect electric conductor boundary condition

$$\mathbf{n} \times \mathbf{E} = -\mathbf{n} \times \frac{\partial \mathbf{A}}{\partial t} = \mathbf{0}$$

implies that

$$\mathbf{n} \times \mathbf{A} = \mathbf{A}(t = 0).$$

Normally the initial condition for $\mathbf{A}$ on a perfect electric conductor is zero. In the cases where the initial condition is different from zero, you can use the magnetic potential boundary condition.

*Magnetic Potential*
The magnetic potential boundary condition

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

makes it possible to set the tangential component of the magnetic vector potential. This can be used to give a tangential electric field by integrating the wanted value with respect to time.

*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 wave types which the boundary condition can handle are

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

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

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

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 plane waves with any incidence angle. For the boundary to be perfectly transparent it is important that the boundary represent an

open boundary. If the wave enters a guided structure where only certain modes can be present the boundary condition will give reflections. For such boundaries use the matched boundary condition.

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.

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

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

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

*Matched Boundary*
The matched boundary condition can be used at boundaries which do not represent a physical boundary. If the boundary is a cross section of a waveguide and the wave is an eigenmode with propagation constant $\beta$ of the waveguide, the matched boundary condition makes the boundary totally nonreflecting.

When the electric field $\mathbf{E}$ is the dependent variable, the matched boundary condition is perfectly transparent for TE and TEM modes. When the magnetic field $\mathbf{H}$ is the dependent variable the matched boundary condition is perfectly transparent for TM and TEM modes. If you want a matched boundary for a TM mode when solving for the electric field $\mathbf{E}$, use the port boundary condition.

When $\mathbf{E}$ is the dependent variable the matched boundary condition reads

$$\mathbf{n}\times(\nabla\times\mathbf{E})-j\beta(\mathbf{E}-(\mathbf{n}\cdot\mathbf{E})\mathbf{n}) = -2j\beta(\mathbf{E}_0-(\mathbf{n}\cdot\mathbf{E}_0)\mathbf{n})$$

or

$$\mathbf{n}\times(\nabla\times\mathbf{E})-j\beta(\mathbf{E}-(\mathbf{n}\cdot\mathbf{E})\mathbf{n}) = -2j\omega\mu\mathbf{n}\times\mathbf{H}_0$$

The source field, $\mathbf{E}_0$ or $\mathbf{H}_0$, represents an incident field.

When $\mathbf{H}$ is the dependent variable the matched boundary condition reads

$$\mathbf{n} \times (\nabla \times \mathbf{H}) - j\beta(\mathbf{H} - (\mathbf{n} \cdot \mathbf{H})\mathbf{n}) \ = \ -2j\beta(\mathbf{H}_0 - (\mathbf{n} \cdot \mathbf{H}_0)\mathbf{n})$$

or

$$\mathbf{n} \times (\nabla \times \mathbf{H}) - j\beta(\mathbf{H} - (\mathbf{n} \cdot \mathbf{H})\mathbf{n}) \ = \ 2j\omega\varepsilon\mathbf{n} \times \mathbf{E}_0$$

When solving for the eigenfrequency or the scattered wave the incident field is not included in the boundary condition:

$$\mathbf{n} \times (\nabla \times \mathbf{E}_{sc}) - j\beta(\mathbf{E}_{sc} - (\mathbf{n} \cdot \mathbf{E}_{sc})\mathbf{n}) \ = \ 0$$

or

$$\mathbf{n} \times (\nabla \times \mathbf{H}_{sc}) - j\beta(\mathbf{H}_{sc} - (\mathbf{n} \cdot \mathbf{H}_{sc})\mathbf{n}) \ = \ 0$$

*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 $\mu_1$ and $\varepsilon_1$ are the material properties of the inner domain, is large, that is $|N| \gg 1$.

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

*Transition Boundary Condition*

The transition boundary condition when the electric field is the dependent variable

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

can be used on interior boundaries to model a thin sheet of a conducting or dielectric medium. The surface impedance $\eta$ is a function of the material properties of the sheet and the thickness.

Similarly the transition boundary condition when the magnetic field is solved for

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

can be used on interior boundaries to model a thin sheet of a highly permeable medium.

### Port
Use the port boundary condition for S-parameter calculations. This setting implies the same boundary condition as the matched boundary condition, but has additional functionality for generating the S-parameters. See the section "S-Parameters and Ports" on page 57 for more information.

### Lumped Port
A lumped port is a simplification of the port boundary condition. Use it to apply a voltage signal between two conductors. It is based on 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 the input electric field and surface impedance are calculated from the input voltage and characteristic impedance of the lumped port. They are both the result of an average over the lumped port boundaries, see "Lumped Ports with Voltage Input" on page 68 for more information how to use this condition.

### Circuit Port
This is a special version of the lumped port that is used for connection to SPICE circuits. The actual boundary condition sets a surface current equal to the current from the circuit node divided by a equivalent width of the port. See the section "SPICE Circuit Import" on page 85 for more information.

### Continuity
The continuity boundary condition

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

is the natural boundary condition ensuring continuity of the tangential components of the electric and magnetic fields.

*Periodic Boundary Condition*
The periodic boundary condition sets up a periodicity between the selected boundaries. See "Periodic Boundary Conditions" on page 37 for more details on this boundary condition.

### EDGE CONDITIONS

When solving for the electric field, some conditions are available on edges.

*Current*
The current lets you specify a current $I_0$ flowing along the edge. This current flows in the direction of the tangential vector of the edge. The easiest way to determine this direction is to make an arrow plot on the edges of a vector with the components t1x, t1y, and t1z.

*Perfect Electric Conductor*
The perfect electric conductor edge condition

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

sets the tangential component of the electric field to zero.

*Magnetic Current*
The magnetic current condition lets you specify a magnetic current $I_{m0}$ flowing along the edge. This current flows in the direction of the tangential vector of the edge. The easiest way to determine this direction is to make an arrow plot on the edges of a vector with the components t1x, t1y, and t1z. The condition is only available for the **H**-formulation.

### APPLICATION MODE VARIABLES

See the section "Electromagnetic Waves (3D)" on page 12 of the *RF Module Reference Guide*.

## In-Plane Waves Application Mode

The *In-plane Waves* application mode covers a situation where there is no variation in the *z* direction, and the electromagnetic field propagates in the modeling *xy*-plane. The application mode handles

- Transverse electric (TE) waves,

- Transverse magnetic (TM) waves,

- Hybrid-mode waves. A hybrid-mode wave is a linear combination of a TE wave and a TM wave. Working with hybrid-mode waves allows for specifying elliptical or circular waves as incoming waves.

## PDE FORMULATION FOR TE WAVES

As the field propagates in the modeling $xy$-plane a TE wave has only one electric field component in the $z$ direction, and the magnetic field lies in the modeling plane. Thus the transient and time-harmonic fields can be written

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

$$\mathbf{H}(x, y, t) = H_x(x, y, t)\mathbf{e}_x + H_y(x, y, t)\mathbf{e}_y = (H_x(x, y)\mathbf{e}_x + H_y(x, y)\mathbf{e}_y)e^{j\omega t}$$

To be able to write the fields in this form, it is also required that $\varepsilon_r$, $\sigma$, and $\mu_r$ are nondiagonal only in the $xy$-plane. $\mu_r$ denotes a 2-by-2 tensor, and $\varepsilon_{rzz}$ and $\sigma_{zz}$ are the relative permittivity and conductivity in the $z$ direction.

Given the above constraints, the equation

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

where

$$\varepsilon_{rc} = \varepsilon_r - j\frac{\sigma}{\omega\varepsilon_0}$$

can be simplified to a scalar equation for $E_z$,

$$-\nabla \cdot (\tilde{\mu}_r \nabla E_z) - \varepsilon_{rczz}k_0^2 E_z = 0$$

where

$$\tilde{\mu}_r = \frac{\mu_r^T}{\det(\mu_r)}$$

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

$$-\nabla \cdot \nabla E_z - n_{zz}^2 k_0^2 E_z = 0$$

When the equation is written using the refractive index, the assumption is that $\mu_r = 1$ and $\sigma = 0$.

The wave number in vacuum $k_0$ is defined by

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

where $c_0$ is the speed of light in vacuum.

The equation in the time domain

$$\mu_0\sigma\frac{\partial \mathbf{A}}{\partial t} + \mu_0\varepsilon_0\frac{\partial}{\partial t}\left(\varepsilon_r\frac{\partial \mathbf{A}}{\partial t} - \mathbf{D}_r\right) + \nabla \times (\mu_r^{-1}(\nabla \times \mathbf{A} - \mathbf{B}_r)) = 0 \qquad (4\text{-}1)$$

can be simplified to a scalar equation for $A_z$:

$$\mu_0\sigma\frac{\partial A_z}{\partial t} + \mu_0\varepsilon_0\frac{\partial}{\partial t}\left(\varepsilon_r\frac{\partial A_z}{\partial t} - D_{rz}\right) + \nabla \cdot (\mu_r^{-1}(\nabla A_z - \mathbf{B}_r)) = 0$$

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

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

$$\mu_0\varepsilon_0\frac{\partial}{\partial t}\left(n^2\frac{\partial A_z}{\partial t}\right) + \nabla \cdot (\nabla A_z - \mathbf{B}_r) = 0$$

When using the refractive index, the assumption is that $\mu_r = 1$ and $\sigma = 0$ and only the constitutive relations for linear materials can be used.

## PDE FORMULATION FOR TM WAVES

TM waves has a magnetic field with only a $z$ component and an electric field in the $xy$-plane. Thus the fields can be written as

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

$$\mathbf{E}(x, y, t) = E_x(x, y, t)\mathbf{e}_x + E_y(x, y, t)\mathbf{e}_y = (E_x(x, y)\mathbf{e}_x + E_y(x, y)\mathbf{e}_y)e^{j\omega t}$$

To write the fields in this form, it is also required that $\mu_r$ and $\varepsilon_r$ are nondiagonal only in the $xy$-plane. $\varepsilon_r$ and $\sigma$ denote 2-by-2 tensors, and $\mu_{rzz}$ is the relative permeability in the $z$ direction.

Given the above constraints the time-harmonic equation

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

can be simplified to a scalar equation for $H_z$,

$$-\nabla \cdot (\tilde{\varepsilon}_{rc} \nabla H_z) - \mu_{rzz} k_0^2 H_z = 0$$

where

$$\tilde{\varepsilon}_{rc} = \frac{\varepsilon_{rc}^T}{\det(\varepsilon_{rc})}$$

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

$$-\nabla \cdot (\tilde{n}^2 \nabla H_z) - k_0^2 H_z = 0$$

In the time domain the equation for TM waves equation is simplified to

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

Here the constitutive relations $\mathbf{B} = \mu_0 \mu_r \mathbf{H} + \mathbf{B}_r$ and $\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E} + \mathbf{D}_r$ and are used Other constitutive relations can also be handled by the application mode.

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

$$\mu_0 \varepsilon_0 \frac{\partial}{\partial t}\left(n^2 \frac{\partial \mathbf{A}}{\partial t}\right) + \nabla \times (\nabla \times \mathbf{A}) = 0$$

In this case only the constitutive relations for linear materials can be used.

## PDE FORMULATION FOR HYBRID-MODE WAVES

Hybrid-mode waves are handled simply by solving the two equations for TE and TM waves together. As these two equation do not couple, they can be solved in sequence.

## VECTOR ELEMENTS

For TM waves in the time-domain, Lagrange elements cannot be used. 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. This is only a problem when solving for the components in the plane. To overcome this problem, the transient TM waves application mode uses *vector elements*, which do not have this limitation.

### APPLICATION MODE PROPERTIES

The application mode specific properties are given in the table below.

| PROPERTY | VALUES | DESCRIPTION |
|----------|--------|-------------|
| Analysis type | Harmonic propagation<br>Eigenfrequency<br>Transient | Specifies which type of analysis to perform. |
| Field type | TE waves<br>TM waves<br>Hybrid-mode waves | Specifies which type of waves to handle. |
| Specify wave using | Frequency<br>Free space wavelength | Specifies if the frequency or the wavelength should be used as input. |
| Specify eigenvalues using | Eigenvalue<br>Eigenfrequency | Specifies if the solver parameters should be given in terms of the eigenvalue $\lambda = k_0^2$ or the eigenfrequency $\nu$. |

### APPLICATION SCALAR VARIABLES

The application-specific variables in this mode are given in the following table.

| PROPERTY | NAME | DEFAULT | DESCRIPTION |
|----------|------|---------|-------------|
| $\mu_0$ | mu0 | 4*pi*1e-7 | Permeability of vacuum |
| $\varepsilon_0$ | epsilon0 | 8.854187817e-12 | Permittivity of vacuum |
| $\nu$ | nu | 1e9 | Frequency |
| $\lambda_0$ | lambda0 | 0.3 | Free space wavelength |

All values are expressed in SI units. When solving the problem as a harmonic problem, the frequency $\nu$ or the free-space wavelength $\lambda_0$ needs to be specified. In the case of eigenfrequency simulations, the frequency is obtained from the solution of the problem.

### 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.

When solving for the electric field, $E_z$ is always continuous, and thus the first condition is automatically fulfilled. The second condition is equivalent to the natural boundary condition

$$\mathbf{n} \cdot (\tilde{\mu}_r \nabla E_z)\mathbf{e}_z = -\mathbf{n} \times (\mu_r^{-1} \nabla \times \mathbf{E}) = \mathbf{n} \times (j\omega\mu_0 \mathbf{H})$$

and is therefore also fulfilled. When solving for the magnetic field, $H_z$ is always continuous, and thus the second condition is automatically fulfilled. The first condition is equivalent to the natural boundary condition

$$\mathbf{n} \cdot (\tilde{\varepsilon}_{rc} \nabla H_z)\mathbf{e}_z = -\mathbf{n} \times (\varepsilon_{rc}^{-1} \nabla \times \mathbf{H}) = -\mathbf{n} \times (j\omega\varepsilon_0 \mathbf{E})$$

and is therefore also fulfilled. When solving the transient equation, 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.

*Magnetic Field*
The magnetic field boundary condition

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

specifies the magnetic field at the boundary. For TM waves the magnetic field can also be specified at interior boundaries.

*Surface Current*
The surface current boundary condition

$$-\mathbf{n} \times \mathbf{H} = J_{sz} \mathbf{e}_z$$
$$\mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) = J_{sz} \mathbf{e}_z$$

specifies a surface current flowing in the *z* direction.

*Perfect Magnetic Conductor*

The perfect magnetic conductor boundary condition

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

sets the magnetic field to zero at a boundary. For TM waves the boundary condition can also be used at interior boundaries.

*Electric Field*

The electric field boundary condition

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

specifies the electric field at an outer boundary. For time-harmonic TE waves the electric field can also be specified at interior boundaries.

*Perfect Electric Conductor*

The perfect electric conductor boundary condition

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

sets the electric field to zero at the boundary. For TE waves the boundary condition can also be used at interior boundaries.

In the transient case this boundary condition is used to set the tangential component of the magnetic vector potential to zero. Since the perfect electric conductor boundary condition

$$\mathbf{n} \times \mathbf{E} = -\mathbf{n} \times \frac{\partial \mathbf{A}}{\partial t} = \mathbf{0}$$

implies that

$$\mathbf{n} \times \mathbf{A} = \mathbf{A}(t = 0).$$

Normally the initial condition for **A** on a perfect electric conductor is zero. In the cases where the initial condition is different from zero the magnetic potential boundary condition can be used.

*Electric and Magnetic Field*

The electric and magnetic field boundary condition

$$E_z = E_{0z} \qquad H_z = H_{0z}$$

can be used for hybrid-mode waves to specify the $z$ component of the electric and magnetic fields at interior boundaries.

*Magnetic Potential*

The magnetic potential boundary condition

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

specifies the magnetic vector potential. This boundary condition can be used to specify the electric field in a transient problem by integrating the electric field with respect to $t$ and entering it as a potential.

*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 wave types which the boundary condition can handle are

$$E = E_{sc}e^{-jk(\mathbf{n} \cdot \mathbf{r})} + E_0 e^{-jk(\mathbf{k} \cdot \mathbf{r})} \qquad \text{Plane scattered wave}$$

$$E = E_{sc}\frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{\sqrt{r}} + E_0 e^{-jk(\mathbf{k} \cdot \mathbf{r})} \qquad \text{Cylindrical scattered wave}$$

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 plane waves with any incidence angle. For the boundary to be perfectly transparent it is important that the boundary represent an open boundary. If the wave enters a guided structure where only certain modes can be present, the boundary condition gives reflections. For such boundaries use the matched boundary condition.

For cylindrical waves you have to specify the point $r_0 = (x_0, y_0)$ around which the waves are cylindrical. The radius equals

$$r = \sqrt{(x - x_0)^2 + (y - y_0)^2}$$

*Matched Boundary*

The matched boundary condition

$$\mathbf{e}_z \cdot \mathbf{n} \times (\nabla \times E_z \mathbf{e}_z) - j\beta E_z = -2j\beta E_{0z} \qquad \text{(TE waves)}$$

$$\mathbf{e}_z \cdot \mathbf{n} \times (\nabla \times H_z \mathbf{e}_z) - j\beta H_z = -2j\beta H_{0z} \qquad \text{(TM waves)}$$

is mainly intended to be used at boundaries that do not represent a physical boundary. If the electric field is an eigenmode of the boundary, and if $\beta$ is the propagation constant of the wave, the boundary is exactly nonreflecting.

*Impedance Boundary Condition*

The impedance boundary condition

$$\mathbf{n} \times \mathbf{H} + \sqrt{\frac{\varepsilon_0 \varepsilon_{rc}}{\mu_0 \mu_r}} E_z \mathbf{e}_z = \sqrt{\frac{\varepsilon_0 \varepsilon_{rc}}{\mu_0 \mu_r}} E_{sz} \mathbf{e}_z \qquad \text{(TE waves)}$$

$$-\mathbf{n} \times \mathbf{E} + \sqrt{\frac{\mu_0 \mu_r}{\varepsilon_0 \varepsilon_{rc}}} H_z \mathbf{e}_z = 0 \qquad \text{(TM waves)}$$

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.

The source electric field $E_{sz}$ can be used to specify a surface current on the boundary.

*Transition Boundary Condition*

The transition boundary condition

$$\eta \mathbf{n} \times (\mathbf{H}_1 - \mathbf{H}_2) + E_z \mathbf{e}_z = -E_{sz} \mathbf{e}_z$$

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

can be used at interior boundaries to model a thin conductive or dielectric sheet. The surface impedance $\eta$ is a function of the material properties and thickness of the sheet.

*Lumped Port*

A lumped port is a simplification of the port boundary condition. Use it to apply a voltage signal between two conductors. It is based on 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 the input electric field and surface impedance are calculated from the input voltage and characteristic impedance of the lumped port. They are both the result of an average over the lumped port boundaries, see "Lumped Ports with Voltage Input"

on page 68 for more information how to use this condition. It is only available for the transient TM waves and transient hybrid-mode waves.

*Circuit Port*

This is a special version of the lumped port that is used for connection to SPICE circuits. The actual boundary condition sets a surface current equal to the current from the circuit node divided by a equivalent width of the port. See the section "SPICE Circuit Import" on page 85 for more information. Similar to the lumped port, it is only available for the transient TM waves and transient hybrid-mode waves.

*Neutral*

The neutral boundary condition

$$\mathbf{e}_z \cdot (\mathbf{n} \times \mathbf{E}) = 0 \qquad \mathbf{e}_z \cdot (\mathbf{n} \times \mathbf{H}) = 0$$

is available only for hybrid-mode waves. It should be used at boundaries where periodic boundary conditions are applied to obtain a Neumann boundary condition for both dependent variables, $E_z$ and $H_z$.

*Continuity*

The continuity boundary condition

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

is the natural boundary condition enforcing continuity of the tangential components of the electric and magnetic fields.

*Periodic Boundary Condition*

The periodic boundary condition sets up a periodicity between the selected boundaries. See "Periodic Boundary Conditions" on page 37 for more details on this boundary condition.

**POINT SOURCES**

When working with TE waves, you can specify a current $I_0$ to flow in the $z$ direction at points in the geometry. For TM waves it is possible to specify a magnetic current $I_{m0}$ flowing out of the plane.

**APPLICATION MODE VARIABLES**

See the section "In-Plane Waves Application Mode" on page 13 of the *RF Module Reference Guide*.

## Axisymmetric Waves Application Mode

Use the *Axisymmetric Waves* application mode in situations where you have axial symmetry and thus no variation in the $\varphi$ direction. The application mode handles:

- Transverse electric (TE) waves,

- Transverse magnetic (TM) waves,

- Hybrid-mode waves. A hybrid-mode wave is a linear combination of a TE wave and a TM wave. Working with hybrid-mode waves allows for specifying elliptical or circular waves as incoming waves.

### PDE FORMULATION FOR TE WAVES

A TE wave has only one electric field component in the $\varphi$ direction, and the magnetic field lies in the modeling plane. Thus the fields can be written

$$\mathbf{E}(r, z, t) = E_\varphi(r, z, t) = E_\varphi(r, z)\mathbf{e}_\varphi e^{j\omega t}$$

$$\mathbf{H}(r, z, t) = H_r(r, z, t)\mathbf{e}_r + H_z(r, z, t)\mathbf{e}_z = (H_r(r, z)\mathbf{e}_r + H_z(r, z)\mathbf{e}_z)e^{j\omega t}$$

To write the fields in this form, it is also required that $\varepsilon_r$ and $\mu_r$ are nondiagonal only in the $rz$-plane. $\mu_r$ denotes a 2-by-2 tensor, and $\varepsilon_{r\varphi\varphi}$ and $\sigma_{\varphi\varphi}$ are the relative permittivity and conductivity in the $\varphi$ direction.

Given the above constraints, the equation

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

can be simplified to a scalar equation for $E_\varphi$. For the transient case a scalar equation for $A_\varphi$ can be derived. The derivation of these equations follows exactly the steps taken in the section "PDE Formulation for TE Waves" on page 149. The only difference between 2D in plane and 2D axial symmetry is how the curl operations are implemented. The curl in 2D axial symmetry is

$$\nabla \times (E_\varphi\mathbf{e}_\varphi) = -\frac{\partial E_\varphi}{\partial z}\mathbf{e}_r + \frac{1}{r}\frac{\partial}{\partial r}(rE_\varphi)\mathbf{e}_z$$

For the $z$-component, the application mode uses the `if` operator to avoid the singularity of $r^{-1}$ and implement the following asymptotic relation at $r = 0$:

$$\lim_{r \to 0}\frac{E_\varphi}{r} = \frac{\partial E_\varphi}{\partial r}$$

The relations given in this section are identical for the transient case and the $A_\varphi$-field.

## PDE FORMULATION FOR TM WAVES

TM waves has a magnetic field with only a $\varphi$ component and an electric field in the $rz$-plane. Thus the fields can be written

$$\mathbf{H}(r, z, t) = H_\varphi(r, z, t) = H_\varphi(r, z)\mathbf{e}_\varphi e^{j\omega t}$$

$$\mathbf{E}(r, z, t) = E_r(r, z, t)\mathbf{e}_r + E_z(r, z, t)\mathbf{e}_z = (E_r(r, z)\mathbf{e}_r + E_z(r, z)\mathbf{e}_z)e^{j\omega t}$$

To write the fields in this form it is also required that $\mu_r$ and $\varepsilon_r$ are nondiagonal only in the $rz$-plane. $\varepsilon_r$ and $\sigma$ denote 2-by-2 tensors, and $\mu_{r\varphi\varphi}$ is the relative permeability is the $\varphi$ direction.

Given the above constraints, the equation

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

can be simplified to a scalar equation for $H_\varphi$, see the derivation for PDE Formulation for TM Waves on page 150. The curl operation is handled in the same way here as for the $E_\varphi$-field in PDE Formulation for TE Waves on page 158.

The transient equation for TM waves can be written

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

using the constitutive relations $\mathbf{B} = \mu_0\mu_r\mathbf{H} + \mathbf{B}_r$ and $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E} + \mathbf{D}_r$. Other constitutive relations can also be handled for transient problems. When using the relations, $n^2 = \varepsilon_r$, $\mu_r = 1$ and $\sigma = 0$ the equation becomes

$$\mu_0\varepsilon_0\frac{\partial}{\partial t}\left(n^2\frac{\partial \mathbf{A}}{\partial t}\right) + \nabla \times (\nabla \times \mathbf{A}) = 0$$

Wen giving the refractive index, the formulation only handles constitutive relations for linear materials.

## PDE FORMULATION FOR HYBRID-MODE WAVES

Hybrid-mode waves are handled simply by solving the two equations for TE and TM waves together. Because the two equation do not couple, it is possible to solve them in sequence.

See the corresponding section for In-Plane Waves on page 152.

**APPLICATION SCALAR VARIABLES**

See the corresponding section for In-Plane Waves on page 152.

**BOUNDARY AND INTERFACE CONDITIONS**

For a thorough discussion about the different boundary conditions, see "Boundary and Interface Conditions" on page 152 for the in-plane case. This section only covers the differences from the in-plane case.

*Scattering Boundary Condition*

In the axisymmetric case the scattering boundary condition can handle:

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

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

The cylindrical waves are cylindrical with respect to the $z$ axis and the spherical waves have to be spherical around the origin. The spherical radius is $r_s = \sqrt{r^2 + z^2}$.

*Axial Symmetry*

You must use the axial symmetry boundary condition along the $z$-axis to obtain the axial symmetry. This condition simply forces the $E_\varphi$-component to zero

$$E_\varphi = 0$$

and in the same way for the The relative permeability must also be scalar along the symmetry axis for $A_\varphi$ and $H_\varphi$ for transient TE waves and TM waves.

**APPLICATION MODE VARIABLES**

See the section "Electromagnetic Waves Application Modes (Axial Symmetry, 2D)" on page 15 of the *RF Module Reference Guide*.

## *Perpendicular Waves Application Mode*

The application mode describing perpendicular electromagnetic waves handles 3D waves that can be described by the fields

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

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

The constant $\beta$ in the exponential is the *propagation constant*. In those cases where $\beta$ is a complex number, you can define the *attenuation constant* $\alpha = -\text{Im}(\beta)$, which expresses how the wave decreases, $e^{-\alpha z}$.

The perpendicular electromagnetic waves application mode uses various formulations derived from Maxwell-Ampère's and Faraday's laws:

$$\nabla \times \mathbf{H} = j\omega\varepsilon\mathbf{E}$$

$$\nabla \times \mathbf{E} = -j\omega\mu\mathbf{H}$$

Two general classes of problems are treated, one where $\lambda = -\delta_z - j\beta$ is calculated as the eigenvalue, and one where the angular frequency $\omega$ is found from the eigenvalue $\lambda = -\delta - j\omega$. The derivation in the following section assumes that the eigenvalue is a complex number with the real part equal to zero, $\lambda = -j\beta$ and $\lambda = -j\omega$. All occurrences of $j\beta$ or $-\beta^2$ can be replaced with $\delta + j\beta$ or $(\delta + j\beta)^2$, and you can do the same replacement for $j\omega$. This is only a simplification to make the expressions easier to read.

### PERPENDICULAR TM WAVES

Treating perpendicular TM waves introduces the restriction that the medium is homogeneous, that is, $\mu$ and $\varepsilon$ are constant. In such a homogeneous medium where the magnetic field is transversal, that is, the $z$ component vanishes, a scalar second-order PDE can be formulated for $E_z$.

### PDE FORMULATION FOR TM WAVES

Given that $\mu$ and $\varepsilon$ are constant, and $H_z = 0$, Maxwell-Ampère's and Faraday's laws give

$$\frac{\partial E_z}{\partial y} + j\beta E_y = -j\omega\mu H_x$$

$$\frac{\partial E_z}{\partial x} + j\beta E_x = j\omega\mu H_y$$

$$j\beta H_y = j\omega\varepsilon_c E_x$$

$$j\beta H_x = -j\omega\varepsilon_c E_y$$

$$\frac{\partial H_y}{\partial x} - \frac{\partial H_x}{\partial y} = j\omega\varepsilon_c E_z$$

By combining these equations, the following equation for $E_z$ can be derived

$$-\nabla \cdot (\mu_{\mathrm{r}}^{-1} \nabla E_z) - \varepsilon_{rc} k_0^2 E_z = -\beta^2 \mu_{\mathrm{r}}^{-1} E_z$$

where both $\varepsilon_{\mathrm{rc}}$ and $\mu_{\mathrm{r}}$ are scalar constants. The wave number in vacuum is here defined by

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

Using the relation $\varepsilon_{\mathrm{r}} = n^2$, where $n$ is the refractive index, the equation can alternatively be written

$$-\nabla \cdot (\nabla E_z) - n^2 k_0^2 E_z = -\beta^2 E_z$$

When using the refractive index the assumption is that $\mu_{\mathrm{r}} = 1$ and $\sigma = 0$.

### PERPENDICULAR TE WAVES

As in the TM case, perpendicular TE waves requires that the medium is homogeneous, that is, $\mu$ and $\varepsilon$ are constant. In such a homogeneous medium where the electric field is transversal, that is, the $z$ component vanishes, a scalar second order PDE can be formulated for $H_z$.

### PDE FORMULATION FOR TE WAVES

Given that $\mu$ and $\varepsilon$ are constant, and $E_z = 0$, Maxwell-Ampère's and Faraday's laws give

$$j\beta E_y = -j\omega\mu H_x$$

$$j\beta E_x = j\omega\mu H_y$$

$$\frac{\partial E_y}{\partial x} - \frac{\partial E_x}{\partial y} = -j\omega\mu H_z$$

$$\frac{\partial H_z}{\partial y} + j\beta H_y = j\omega\varepsilon_c E_x$$

$$\frac{\partial H_z}{\partial x} + j\beta H_x = -j\omega\varepsilon_c E_y$$

By combining these equations, the following equation for $H_z$ can be derived

$$-\nabla \cdot (\varepsilon_{\mathrm{rc}}^{-1} \nabla H_z) - \mu_{\mathrm{r}} k_0^2 H_z = -\beta^2 \varepsilon_{\mathrm{rc}}^{-1} H_z \,,$$

Both the permeability and the permittivity are scalar constants. Using the relation $\varepsilon_{\mathrm{r}} = n^2$, where $n$ is the refractive index, the equation can alternatively be written

$$-\nabla \cdot (n^{-2}\nabla H_z) - k_0^2 H_z = -\beta^2 n^{-2} H_z$$

## PERPENDICULAR HYBRID-MODE WAVES

Perpendicular hybrid-mode waves treats the case of transversal fields and inhomogeneous materials. In this case there are two possible formulations available. Either two equations for the two transversal field components are solved, or a set of three equations for all three fields components.

## PDE FORMULATION FOR HYBRID-MODE WAVES

Consider anisotropic complex relative permittivity and relative permeability of the form

$$\varepsilon_{rc} = \begin{bmatrix} \varepsilon_{rcxx} & \varepsilon_{rcxy} & 0 \\ \varepsilon_{rcyx} & \varepsilon_{rcyy} & 0 \\ 0 & 0 & \varepsilon_{rczz} \end{bmatrix} \qquad \mu_r = \begin{bmatrix} \mu_{rxx} & \mu_{rxy} & 0 \\ \mu_{ryx} & \mu_{ryy} & 0 \\ 0 & 0 & \mu_{rzz} \end{bmatrix}$$

### Three-Component Formulation

This derivation is made for the electric field, but the derivation for the magnetic field is almost identical. The starting point is the equation

$$\nabla \times \mathbf{H} = \mathbf{J}_i + j\omega\mathbf{D} = \mathbf{J}$$

Multiplying with $j\omega\mu_0$ and converting into the weak form results in an integral expression

$$-\mu_0 \int_\Omega (j\omega\mathbf{J} \cdot \tilde{\mathbf{E}} - j\omega\mathbf{H} \cdot (\nabla \times \tilde{\mathbf{E}}))d\Omega + \mu_0 \oint_{\partial\Omega} (\mathbf{n} \times s\mathbf{H} \cdot \tilde{\mathbf{E}})d\partial\Omega = \mathbf{0}$$

where $\tilde{\mathbf{E}}$ denotes the vector-valued test function. For electric fields of the form

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}(x,y)e^{-\alpha z} = \mathbf{E}_T(x,y)e^{-\alpha z} + \mathbf{a}_z \alpha e_z(x,y)e^{-\alpha z}$$

$$\tilde{\mathbf{E}}(\mathbf{r}) = \tilde{\mathbf{E}}(x,y)e^{\alpha z} = \tilde{\mathbf{E}}_T(x,y)e^{\alpha z} + \mathbf{a}_z(-\alpha)\tilde{e}_z(x,y)e^{\alpha z}$$

a straightforward calculation gives

$$-\mu_0 \int_\Omega (j\omega \mathbf{J}_T \cdot \tilde{\mathbf{E}}_T - j\omega \mathbf{H}_T \cdot \mathrm{curl}\tilde{E}_T - j\omega H_z \cdot \mathrm{curl}\tilde{E}_{Tz})d\Omega$$

$$+ j\omega\mu_0 \oint_{\partial\Omega} (\mathbf{n} \times \mathbf{H} \cdot \tilde{\mathbf{E}}_T)d\partial\Omega - \mu_0 \int_\Omega ((j\omega J_{iz}) \cdot (-\alpha\tilde{e}_z) - j\omega \mathbf{H}_T \cdot \mathrm{curl}\tilde{\mathbf{E}}_{zT})d\Omega$$

$$+ j\omega\mu_0 \oint_{\partial\Omega} (\mathbf{n} \times \mathbf{H} \cdot \mathbf{a}_z(-\alpha\tilde{e}_z))d\partial\Omega$$

where $\alpha$ is the complex valued propagation constant, usually equal to the pure imaginary propagation constant $j\beta$. The following variable declarations are necessary to simplify the expressions:

$$\mathbf{H} = \frac{\mu^{-1}}{j\omega}[\alpha(\mathbf{a}_z \times \mathbf{E}_T - \nabla \times \mathbf{a}_z e_z) - \mathbf{a}_z(\mathbf{a}_z \cdot (\nabla \times \mathbf{E}_T))] = \frac{j\omega(\mathbf{H}_T + \mathbf{a}_z H_z)}{j\omega}$$

$$\nabla \times \tilde{\mathbf{E}}_T = \alpha\mathbf{a}_z \times \tilde{\mathbf{E}}_T + \mathbf{a}_z(\mathbf{a}_z \cdot (\nabla \times \tilde{\mathbf{E}}_T)) = \mathrm{curl}\tilde{E}_T + \mathbf{a}_z\mathrm{curl}\tilde{E}_{Tz}$$

$$\nabla \times \mathbf{a}_z(-\alpha)\tilde{e}_z = \mathrm{curl}\tilde{\mathbf{E}}_{zT}$$

Two types of eigenvalues can be used, $\lambda = -\delta_z - j\beta$ and $\lambda = -\delta - j\omega\beta$. For simplicity, the previous derivation only included the imaginary part $j\omega$. For damped eigenvalue problems the eigenvalue becomes complex, and the real parts $\delta_z$ and $\delta$ become nonzero. These terms are available as the variables `dampz` and `damp`, respectively.

*Two-Component Formulation*

In the following, let $\varepsilon_{\mathrm{rct}}$ and $\mu_{\mathrm{rt}}$ denote the 2-by-2 tensors in the transversal $xy$-plane,

$$\varepsilon_{\mathrm{rct}} = \begin{bmatrix} \varepsilon_{rcxx} & \varepsilon_{rcxy} \\ \varepsilon_{rcyx} & \varepsilon_{rcyy} \end{bmatrix} \qquad \mu_{\mathrm{rt}} = \begin{bmatrix} \mu_{rxx} & \mu_{rxy} \\ \mu_{ryx} & \mu_{ryy} \end{bmatrix}$$

It is possible to reduce the set of three equations to two equations for the transversal components of the magnetic field $\mathbf{H}_t$. To do this, use that $\nabla \cdot \mathbf{B} = 0$, which for the magnetic field becomes

$$\nabla_t \cdot \mu_{\mathrm{rt}}\mathbf{H}_t - j\beta\mu_{rzz}H_z = 0$$

Substituting this into the equation

$$\nabla_t \times (\varepsilon_{rczz}^{-1}\nabla_t \times \mathbf{H}_t) - j\beta\tilde{\varepsilon}_{\mathrm{rct}}\nabla_t H_z - (k_0^2\mu_{\mathrm{rt}} - \beta^2\tilde{\varepsilon}_{\mathrm{rct}})\mathbf{H}_t = \mathbf{0}$$

gives

$$\nabla_t \times (\varepsilon_{rczz}^{-1} \nabla_t \times \mathbf{H}_t) - \tilde{\varepsilon}_{rct} \nabla_t (\mu_{rzz}^{-1} \nabla_t \cdot \mu_{rt} \mathbf{H}_t) - (k_0^2 \mu_{rt} - \beta^2 \tilde{\varepsilon}_{rt}) \mathbf{H}_t = \mathbf{0}$$

You can solve this equation as an eigenvalue problem both for $\lambda = -j\beta$ and $\lambda = -j\omega$.

*Electric Field Formulations*
To obtain the corresponding formulations for the electric field from the equations above, replace **H** by **E** and interchange $\varepsilon_c$ and $\mu$.

## APPLICATION MODE PROPERTIES
The following table lists the properties for this application mode:

| PROPERTY | VALUES | DESCRIPTION |
| --- | --- | --- |
| Analysis type | Mode analysis<br>Eigenfrequency | Specifies which type of analysis to perform. |
| Field type | TE waves<br>TM waves<br>Hybrid-mode waves | Specifies which type of waves to handle. |
| Specify wave using | Frequency<br>Free-space wavelength | Specifies if the frequency or the wavelength should be used as input. |
| Solve for | Electric field<br>Magnetic field | Specifies which field is the dependent variable. |
| Field components | In-plane components<br>All three components | Specifies if the two or three-component formulation is used for hybrid-mode waves. |
| Specify eigenvalues using | Eigenvalue<br>Eigenfrequency<br>Propagation constant<br>Effective mode index | Specifies if the solver parameters should be given in terms of the eigenvalue or some other quantity. When doing mode analysis, the effective mode index or the propagation constant can be used. When doing eigenfrequency analysis, the eigenfrequency can be used. |

The following table lists the application-specific variables in this application mode:

| PROPERTY | NAME | DEFAULT | UNIT | DESCRIPTION |
|----------|------|---------|------|-------------|
| $\mu_0$ | mu0 | 4*pi*1e-7 | H/m | Permeability of vacuum |
| $\varepsilon_0$ | epsilon0 | 8.854187817e-12 | F/m | Permittivity of vacuum |
| $\nu$ | nu | 1e9 | Hz | Frequency |
| $\lambda_0$ | lambda0 | 0.3 | m | Free-space wavelength |
| $\beta$ | beta | 30 | - | Propagation constant |

The constants $\varepsilon_0$ and $\mu_0$ are expressed in SI units. When solving the problem as an eigenvalue problem for $\beta$, you must specify the frequency $\nu$ or the free-space wavelength $\lambda_0$, and $\beta$ is obtained from the solution. When solving the problem as an eigenvalue problem for $\nu$, you must specify the propagation constant $\beta$. In this case, $\nu$ is obtained from the solution.

**BOUNDARY AND INTERFACE CONDITIONS**

With no surface currents present, the interface conditions

$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{0}$$
$$\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$
$$\mathbf{n}_2 \cdot (\varepsilon_1 \mathbf{E}_1 - \varepsilon_2 \mathbf{E}_2) = 0$$
$$\mathbf{n}_2 \cdot (\mu_1 \mathbf{H}_1 - \mu_2 \mathbf{H}_2) = 0$$

must be fulfilled. Depending on which formulation you use, it is necessary to analyze these conditions differently.

For TE and TM waves it can be shown that they are not fulfilled at interfaces between different media, hence these formulation only handles homogeneous media.

For hybrid-mode waves the following section covers the cases where you solve for the magnetic field. The cases where you solve for the electric field can be analyzed analogously.

*Three-Component Formulation*

When the three-component formulation is used, the condition

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

is the natural boundary condition, which comes from the equation. This can most easily be seen from the equation used to derive the formulation from,

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

It leads to the natural boundary condition

$$-\mathbf{n} \times [(\varepsilon_{rc}^{-1} \nabla \times \mathbf{H})_1 - (\varepsilon_{rc}^{-1} \nabla \times \mathbf{H})_2] = -\mathbf{n} \times j\omega\varepsilon_0(\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{0}$$

Nothing in the formulation explicitly constrains the normal component of $\mathbf{E}$. The condition

$$\mathbf{n}_2 \cdot (\varepsilon_1 \mathbf{E}_1 - \varepsilon_2 \mathbf{E}_2) = 0$$

falls out as a part of the solution.

The conditions for the magnetic field require that $H_z$ as well as the tangential component of $\mathbf{H}_t$ are continuous across interior boundaries. On the other hand the normal component of $\mathbf{H}_t$ must be allowed to be discontinuous if $\mu$ is discontinuous across the interior boundary. These conditions are fulfilled by using vector elements for $\mathbf{H}_t$ and Lagrange elements for $H_z$. Vector elements for $\mathbf{H}_t$ are the best choice also when $\mu$ does not vary, because unlike the Lagrange elements, they do not give rise to spurious modes.

*Two-Component Formulation*
The two-component formulation also has the requirement that the tangential component of $\mathbf{H}_t$ is continuous across interior boundaries. The normal component of $\mathbf{H}_t$ is allowed to be discontinuous if $\mu$ is discontinuous across the interior boundary. It is not possible to use vector elements for $\mathbf{H}_t$ in the two-component formulation, however, because vector elements cannot be used when the equation contains a divergence term. This limits the element selection to Lagrange elements. They force the solution variables to be continuous across interior boundaries.

---

**Note:** The two-component formulation for the magnetic field requires that $\mu$ is everywhere continuous. Similarly the two-component formulation for the electric field requires that $\varepsilon_c$ is everywhere continuous.

---

Although Lagrange elements are used, there is no problem with spurious modes for this equation formulation.

*Perfect Magnetic Conductor*

The perfect magnetic conductor boundary condition,

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

sets the tangential component of the magnetic field to zero at the boundary.

This boundary condition is not available in the two-component hybrid-mode waves formulation for $\mathbf{E}$, because it would require a constraint on the second derivative of $\mathbf{E}_t$.

*Perfect Electric Conductor*

The perfect electric conductor boundary condition,

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

sets the tangential component of the electric field to zero at the boundary.

This boundary condition is not available for the two-component formulation for $\mathbf{H}$, because it would require a constraint on the second derivative of $\mathbf{H}_t$.

*Continuity*

The continuity boundary condition

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

is the natural boundary condition ensuring continuity of the tangential components of the electric and magnetic fields.

### SELECTING EQUATION FORMULATION

As has been shown when deriving the equations, the TE, TM, and the two-component hybrid-mode waves formulation all have their limitations. Some general guidelines are:

- The TE and TM waves formulations require that the whole geometry consists of one single homogeneous medium.
- Some derived quantities, for example the energy, are obtained from the second derivative of the dependent variables in the two-component formulation. Therefore they might be less accurate than in the three-component formulation.
- The two-component formulation does not allow both the permittivity and permeability to be discontinuous.

- The two-component formulation does not support the perfect magnetic conductor (PMC) and the perfect electric conductor (PEC) boundary conditions.
- The three-component formulation is the only formulation that supports absorbing boundaries, perfectly matched layers (PMLs), and impedance boundary conditions.

The following table shows the characteristics of the different equation formulations.

| Analysis Type | TYPE | DEP. VARIABLES | EIGENVALUE | DISCONT. $\varepsilon$ | DISCONT. $\mu$ | PMC | PEC |
|---|---|---|---|---|---|---|---|
| Eigenmode | TE | $H_z$ | $\lambda = -j\beta$ | | | √ | √ |
| Eigenfrequency | TE | $H_z$ | $\lambda = -j\omega$ | | | √ | √ |
| Eigenmode | TM | $E_z$ | $\lambda = -j\beta$ | | | √ | √ |
| Eigenfrequency | TM | $E_z$ | $\lambda = -j\omega$ | | | √ | √ |
| Eigenmode | Hybrid | $H_x, H_y, H_z$ | $\lambda = -j\beta$ | √ | √ | √ | √ |
| Eigenmode | Hybrid | $H_x, H_y$ | $\lambda = -j\beta$ | √ | | √ | |
| Eigenfrequency | Hybrid | $H_x, H_y, H_z$ | $\lambda = -j\omega$ | √ | √ | √ | √ |
| Eigenfrequency | Hybrid | $H_x, H_y$ | $\lambda = -j\omega$ | √ | | √ | |
| Eigenmode | Hybrid | $E_x, E_y, E_z$ | $\lambda = -j\beta$ | √ | √ | √ | √ |
| Eigenmode | Hybrid | $E_x, E_y$ | $\lambda = -j\beta$ | | √ | | √ |
| Eigenfrequency | Hybrid | $E_x, E_y, E_z$ | $\lambda = -j\omega$ | √ | √ | √ | √ |
| Eigenfrequency | Hybrid | $E_x, E_y$ | $\lambda = -j\omega$ | | √ | | √ |

**APPLICATION MODE VARIABLES**

See the section "Perpendicular Waves Application Modes(2D)" on page 18 of the *RF Module Reference Guide*.

## Boundary Mode Analysis Application Mode in 3D

The *Boundary Mode Analysis* application mode in 3D treats the same type of problems as the Perpendicular Waves application mode in 2D. The type of fields that can be described are

$$\mathbf{H}(x, y, z, t) = \mathbf{H}(\mathbf{r}_t)e^{j(\omega t + \beta(\mathbf{n} \cdot \mathbf{r}))}$$

$$\mathbf{E}(x, y, z, t) = \mathbf{E}(\mathbf{r}_t)e^{j(\omega t + \beta(\mathbf{n} \cdot \mathbf{r}))}$$

The mode analysis can be performed on planar boundaries representing a cross section of a waveguide. The coordinates $\mathbf{r}_t$ span the 2D boundary and $\mathbf{n}$ is the outward normal to the boundary. The wave propagates in through the boundary.

For a homogeneous boundary TE and TM waves are found, while in the general case a hybrid-mode wave calculation has to be made.

### TE WAVES

When the medium is homogeneous TE waves can exist. In this case an equation for $H_n$, the component of the magnetic field normal to the boundary, can be derived,

$$-\nabla_t \cdot (\varepsilon_{rc}^{-1} \nabla_t H_n) - \mu_r k_0^2 H_n = -\beta^2 \varepsilon_{rc}^{-1} H_n$$

The operator $\nabla_t$ is the tangential derivative operator.

### TM WAVES

Like for TE waves, the TM waves only appears when the medium is homogeneous. An equation for the electric field component normal to the boundary can be formulated,

$$-\nabla_t \cdot (\mu_r^{-1} \nabla_t E_n) - \varepsilon_{rc} k_0^2 E_n = -\beta^2 \mu_r^{-1} E_n$$

### HYBRID-MODE WAVES

In the general case with a inhomogeneous medium all field components are nonzero. An system of equation can be formulated for either the **H** field or the **E** field. Writing the equations in terms of the **H** field they read

$$\nabla_t \times (\varepsilon_{rc}^{-1} \nabla_t \times \mathbf{h}_t) - k_0^2 \mu_r \mathbf{h}_t = -\beta^2 \varepsilon_{rc}^{-1} \left( \frac{\nabla h_n}{k_0} + \mathbf{h}_t \right)$$

$$0 = \beta^2 \left( \frac{1}{k_0} \nabla_t \cdot \varepsilon_{rc}^{-1} \left( \frac{\nabla h_n}{k_0} + \mathbf{h}_t \right) + \mu_r h_n \right)$$

Here

$$\mathbf{h}_t = \beta \mathbf{H}_t \qquad h_n = -j k_0 H_n$$

For a derivation, see the Perpendicular Waves application on page 163. The equation above is only formulated for an isotropic medium.

The corresponding formulation for the electric field is obtained by replacing **H** by **E** and by interchanging $\varepsilon_c$ and $\mu$.

## APPLICATION MODE PROPERTIES

The application mode specific properties are given in the table below.

| PROPERTY | VALUES | DESCRIPTION |
|---|---|---|
| Field type | TE waves<br>TM waves<br>Hybrid-mode waves | Specifies which type of waves to handle. |
| Specify wave using | Frequency<br>Free-space wavelength | Specifies if the frequency or the wavelength should be used as input. |
| Solve for | Electric field<br>Magnetic field | Specifies which field is the dependent variable. |
| Specify eigenvalues using | Eigenvalue<br>Propagation constant<br>Effective mode index | Specifies if the solver parameters should be given in terms of the eigenvalue, the effective mode index, or the propagation constant. |

## APPLICATION SCALAR VARIABLES

The application-specific variables in this mode are given in the following table:

| PROPERTY | NAME | DEFAULT | UNIT | DESCRIPTION |
|---|---|---|---|---|
| $\mu_0$ | mu0 | 4*pi*1e-7 | H/m | Permeability of vacuum |
| $\varepsilon_0$ | epsilon0 | 8.854187817e-12 | F/m | Permittivity of vacuum |
| $\nu$ | nu | 1e9 | Hz | Frequency |
| $\lambda_0$ | lambda0 | 0.3 | m | Free-space wavelength |

All values are expressed in SI units.

## EDGE CONDITIONS

At interior edges of the boundary, the conditions

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

have to be fulfilled. In can be shown that the equations for TE and TM waves do not fulfill these conditions at interfaces between different media. Therefore these equations only apply to homogeneous media.

*Perfect Magnetic Conductor*
The perfect magnetic conductor edge condition,

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

sets the tangential component of the magnetic field to zero at the edge.

This condition cannot be applied to interior edges when the electric field is the dependent variable.

*Perfect Electric Conductor*
The perfect electric conductor edge condition,

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

sets the tangential component of the electric field to zero at the edge.

This condition cannot be applied to interior edges when the magnetic field is the dependent variable.

*Continuity at Interior Edges*
The continuity edge condition

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

is the natural condition ensuring continuity of the tangential components of the electric and magnetic fields.

**APPLICATION MODE VARIABLES**
See the section "Boundary Mode Analysis Application Modes (3D)" on page 21 of the *RF Module Reference Guide*.

## Boundary Mode Analysis Application Mode in 2D

The *Boundary Mode Analysis* application mode in 2D treats mode analysis problems in 1D. The type of fields that can be described are

$$\mathbf{H}(x, y, t) = \mathbf{H}(\mathbf{r}_t)e^{j(\omega t + \beta(\mathbf{n} \cdot \mathbf{r}))}$$

$$\mathbf{E}(x, y, t) = \mathbf{E}(\mathbf{r}_t)e^{j(\omega t + \beta(\mathbf{n} \cdot \mathbf{r}))}$$

The mode analysis can be performed on planar boundaries representing a cross section of a waveguide. The coordinates $\mathbf{r}_t$ span the 1D boundary and $\mathbf{n}$ is the outward normal to the boundary. The wave propagates in through the boundary.

The application mode can also handle axisymmetric waves. These waves have to travel in the $z$ direction,

$$\mathbf{H}(r, z, t) = \mathbf{H}(r)e^{j(\omega t + \beta z)}$$

$$\mathbf{E}(r, z, t) = \mathbf{E}(r)e^{j(\omega t + \beta z)}$$

## TE WAVES

When the medium is homogeneous TE waves can exist. In this case an equation for $E_z$, the component of the magnetic field normal to the boundary, can be derived,

$$-\nabla_t \cdot (\mu_r^{-1} \nabla_t E_z) - \varepsilon_{rc} k_0^2 E_z = -\beta^2 \mu_r^{-1} E_z$$

The operator $\nabla_t$ is the tangential derivative operator.

For axisymmetric TE waves the equation reads

$$-\frac{\partial}{\partial r}\left(\mu_r^{-1}\frac{1}{r}\frac{\partial}{\partial r}(rE_\varphi)\right) - \varepsilon_{rc} k_0^2 E_\varphi = -\beta^2 \mu_r^{-1} E_\varphi$$

## TM WAVES

Like for TE waves, the TM waves only appears when the medium is homogeneous. An equation for the electric field component normal to the boundary can be formulated,

$$-\nabla_t \cdot (\varepsilon_r^{-1} \nabla_t H_z) - \mu_r k_0^2 E_n = -\beta^2 \varepsilon_r^{-1} H_z$$

For axisymmetric TM waves the equation reads

$$-\frac{\partial}{\partial r}\left(\varepsilon_{rc}^{-1}\frac{1}{r}\frac{\partial}{\partial r}(rH_\varphi)\right) - \mu_r k_0^2 H_\varphi = -\beta^2 \varepsilon_{rc}^{-1} H_\varphi$$

## APPLICATION MODE PROPERTIES

The application mode specific properties are given in the table below.

| PROPERTY | VALUES | DESCRIPTION |
|---|---|---|
| Field type | TE waves<br>TM waves | Specifies which type of waves to handle |

| PROPERTY | VALUES | DESCRIPTION |
|---|---|---|
| Specify wave using | Frequency<br>Free-space wavelength | Specifies if the frequency or the wavelength should be used as input |
| Specify eigenvalues using | Eigenvalue<br>Propagation constant<br>Effective mode index | Specifies if the solver parameters should be given in terms of the eigenvalue, the effective mode index, or the propagation constant |

## APPLICATION SCALAR VARIABLES

The application-specific variables in this mode are given in the following table:

| PROPERTY | NAME | DEFAULT | UNIT | DESCRIPTION |
|---|---|---|---|---|
| $\mu_0$ | mu0 | 4*pi*1e-7 | H/m | Permeability of vacuum |
| $\varepsilon_0$ | epsilon0 | 8.854187817e-12 | F/m | Permittivity of vacuum |
| $\nu$ | nu | 1e9 | Hz | Frequency |
| $\lambda_0$ | lambda0 | 0.3 | m | Free-space wavelength |

All values are expressed in SI units.

## POINT CONDITIONS

At interior points of the boundary, the conditions

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

have to be fulfilled. It can be shown that the equations for TE and TM waves do not fulfill these conditions at interfaces between different media. Therefore these equations only apply to homogeneous media.

*Perfect Magnetic Conductor*
The perfect magnetic conductor point condition,

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

sets the magnetic field to zero at the point.

This condition is not available for interior edges when the magnetic field is the dependent variable.

*Perfect Electric Conductor*
The perfect electric conductor point condition,

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

sets the electric field to zero at the point.

This condition cannot be applied to interior edges when the electric field is the dependent variable.

*Continuity at Interior Points*
The continuity edge condition

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

is the natural condition ensuring continuity of the tangential components of the electric and magnetic fields.

**APPLICATION MODE VARIABLES**
See the section "Boundary Mode Analysis Application Modes (2D)" on page 24 of the *RF Module Reference Guide*.

# 5

# 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 **D** and **E** fields and between the **B** and **H** fields. These relations depend on the material properties.

**cutoff frequency**  The lowest frequency for which a given mode can propagate through, for example, a waveguide or optical fiber.

**edge element**  See *vector element*.

**eigenmode**  A possible propagating mode of, for example, a waveguide or optical fiber.

**electric dipole**  Two equal and opposite charges $+q$ and $-q$ separated a short distance $d$. The electric dipole moment is given by $\mathbf{p} = q\mathbf{d}$, where $\mathbf{d}$ is a vector going from $-q$ to $+q$.

**gauge transformation**  A variable transformation of the electric and magnetic potentials that leaves Maxwell's equations invariant.

**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