COMSOL
MULTIPHYSICS

RF MODULE

VERSION 3.4

COMSOL

RF Module User's Guide
© COPYRIGHT 1994–2007 by COMSOL AB. All rights reserved

Patent pending

Version: October 2007   COMSOL 3.4

C O N T E N T S

Chapter 3: Review of Electromagnetics

Chapter 4: The Application Modes

Chapter 5: Glossary

1

Introduction

The RF Module 3.4 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 in your hands, the *RF Module User's Guide*, introduces you to the basic functionality in the module, reviews new features in the version 3.4 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. It is available in HTML and PDF format from the COMSOL Help Desk.

Typographical Conventions

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

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

- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user interface during geometry modeling. However, nowhere on the screen will you see the term "Draw" referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).

- The symbol **>** indicates a menu item or an item in a folder in the **Model Navigator**. For example, **Physics>Equation System>Subdomain Settings** is equivalent to: On the

Physics menu, point to **Equation System** and then click **Subdomain Settings**. **COMSOL Multiphysics>Heat Transfer>Conduction** means: Open the **COMSOL Multiphysics** folder, open the **Heat Transfer** folder, and select **Conduction**.

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

- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

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. You can also export a simulation to COMSOL Script or MATLAB. Alternatively, save it as a Model M-file, a script file that runs in both COMSOL Script and 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 3.4

This release of the RF Module 3.4 adds a number of valuable new capabilities including these new features:

- Lumped ports with voltage excitation in transient analysis for supplying a voltage signal to, for example, a circuit board or an antenna. See "Lumped Ports with Voltage Input" on page 67.
- Support for multiple numerical ports in models. See "Numeric Modes" on page 58
- Periodic boundary condition in the in-plane and electromagnetic wave application modes that greatly simplifies the process of modeling periodic boundaries, especially for 3D electromagnetic waves. See "Periodic Boundary Conditions" on page 37.
- Faster and easier far-field postprocessing. See "Far-Field Postprocessing" on page 49.

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 "Waveguide H-Bend with S-parameters" on page 80).

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 457 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 107, which provides a summary of all application modes with dependent variables and references to more detailed information.

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 112.

- 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 148.

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 139.

- In-Plane Waves

  See "In-Plane Waves Application Mode" on page 123 or "Axisymmetric Waves Application Mode" on page 132.

- 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 151.

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 on the **New** toolbar button. On the **New** tab in the **Model Navigator** you specify the application mode, names of dependent variables, and the analysis type: static, time-harmonic, transient, mode analysis, or eigenfrequency. You can also set up a combination of application modes from the 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 234 in the *COMSOL Multiphysics User's Guide*. Valid boundary conditions for each electromagnetics mode are summarized in "The Application Mode Formulations" on page 106 of this manual.

SUBDOMAIN SETTING

You specify equation parameters in the **Subdomain Settings** dialog box. For details see "Specifying Subdomain Settings and PDE Coefficients" on page 205 in the *COMSOL Multiphysics User's Guide*. The physical parameters of specific interest for electromagnetics modeling are summarized in "The Application Mode Formulations" on page 106 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 286 of the *COMSOL Multiphysics User's Guide*.

COMPUTING THE SOLUTION

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

POSTPROCESSING AND VISUALIZATION

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

ADDITIONAL POSTPROCESSING

For further postprocessing calculations, you can export the solution to COMSOL Script or MATLAB. Details of modeling by programming are available in "The Programming Language" on page 30 of this manual and in the *COMSOL Multiphysics Scripting 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 "Waveguide H-Bend with S-parameters" on page 80 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.
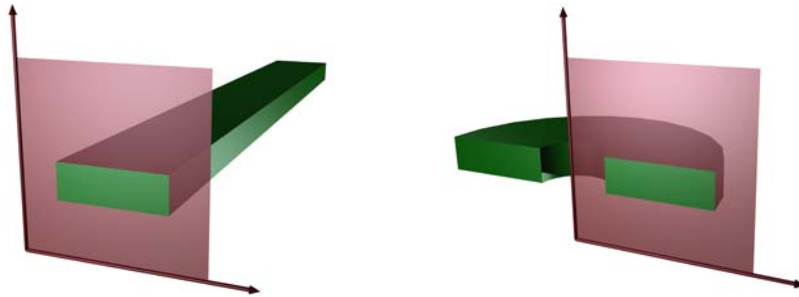
Figure 2-1: The cross sections and their real geometry for Cartesian coordinates and cylindrical coordinates (axial symmetry).

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 αr, where α is the revolution angle (for example, 2π for a full turn). See the model "Monoconical RF Antenna" on page 45 in the *RF Module Model Library* for an example.

3 D   P R O B L E M S

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 45 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 341 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 5 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 359 in the *COMSOL Multiphysics User's Guide*. See also "Solving Large 3D Problems" on page 69.

Hermitian Transpose

The **Use Hermitian transpose of constraint matrix and in symmetry detection** setting on the **Advanced** tab 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, a so-called Floquet periodicity. In this case, select the **Use Hermitian transpose of constraint matrix and in symmetry detection** check box.

An Example—Waveguide H-Bend

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.
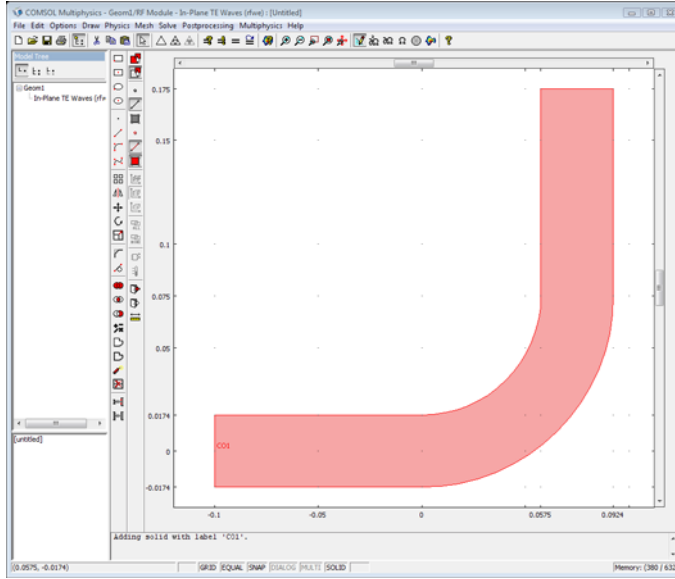
DOMAIN EQUATIONS

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).

$E_z = \cos((\pi/2)(y/a))$
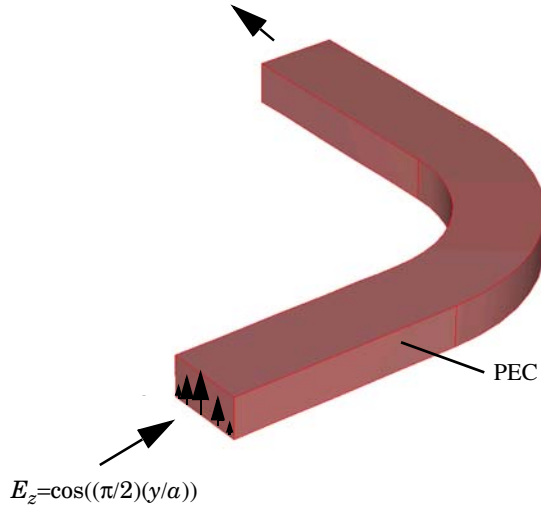
The width of the waveguide is chosen so that it has a cutoff frequency of 4.3 GHz. This makes the waveguide operational between 5.4 GHz up to 8.1 GHz. At higher frequencies other modes than the TE_{10} mode appear, causing a "dirty" signal. The input wave then splits into several modes that are hard 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, β, 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

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

The propagation constant evaluates to

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

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.
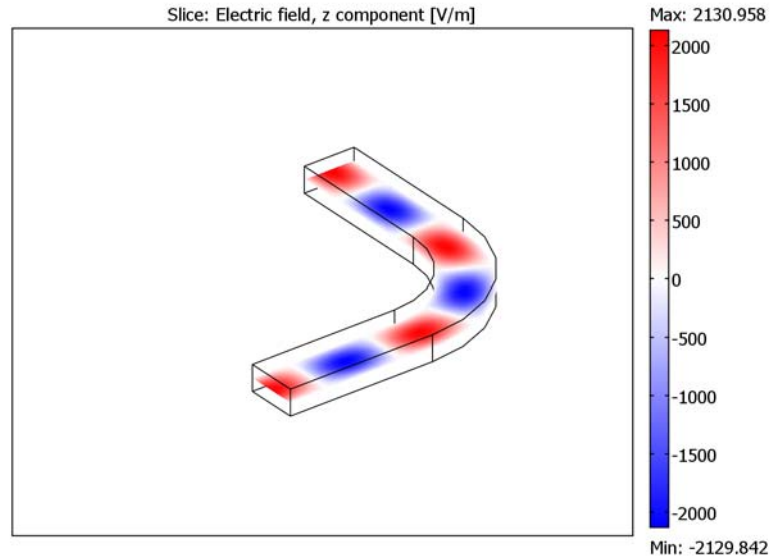
To verify that there is almost no reflection you can compare the power flow of the outgoing wave to that of the incoming wave. The entrance port is excited using the port boundary condition, which automatically normalizes the excitation to unit power (1 W). Comparing this to the power flow over the output 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/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

Define the following constants in the **Constants** dialog box (the descriptions are optional).
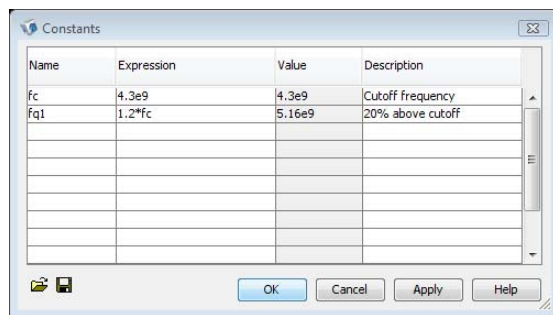
NAME	EXPRESSION	DESCRIPTION
fc	4.3e9	Cutoff frequency
fq1	1.2*fc	20% above cutoff

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 axis and grid settings, and then 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.
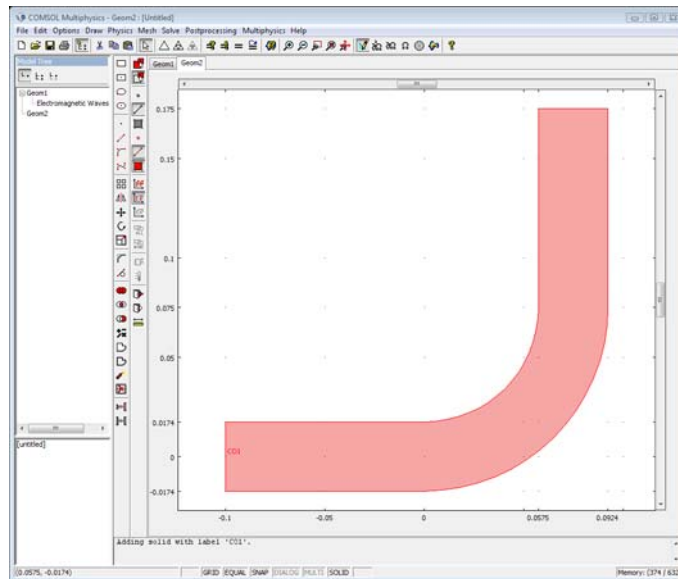
9 Select **Extrude** from the **Draw** menu. Extrude the object using a distance of 0.0174.

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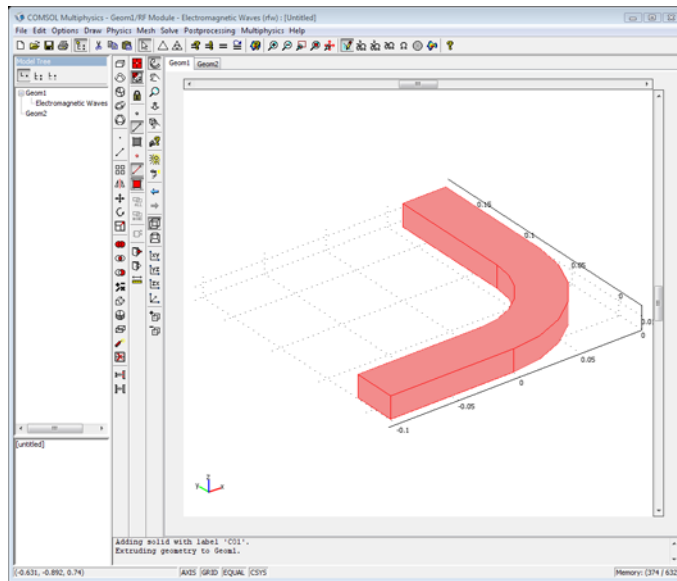
PHYSICS SETTINGS

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 1	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 ε_r, μ_r, and σ, 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.
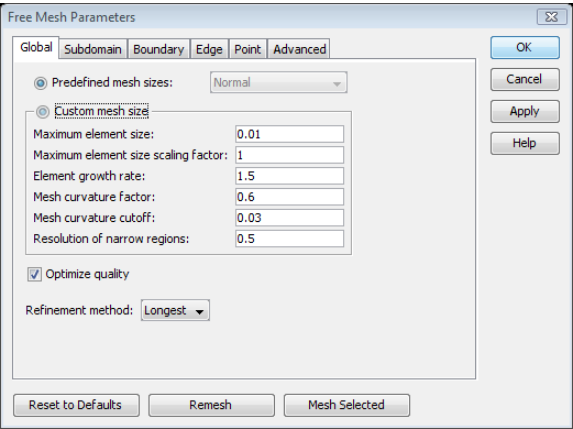
2 Click **Remesh** to generate the mesh; then click **OK**.
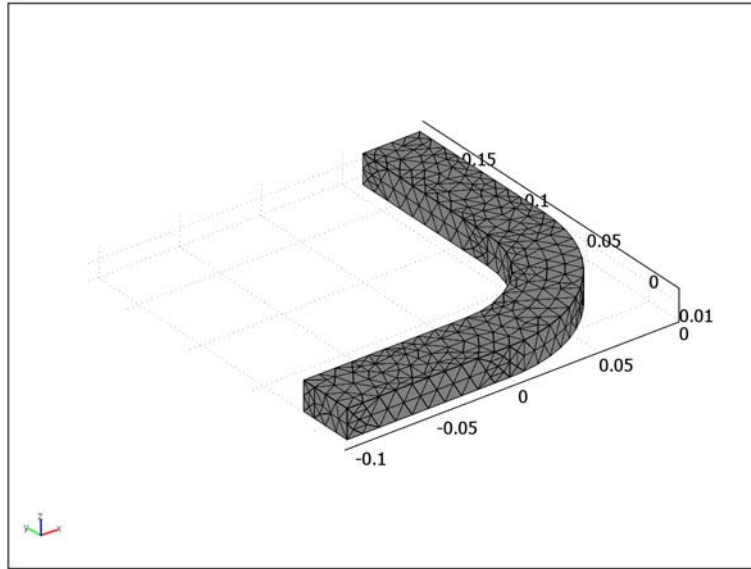
Click the **Solve** button on the Main toolbar to solve the problem.

POSTPROCESSING AND VISUALIZATION

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

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

Slice: Total energy density, time average [J/m³]
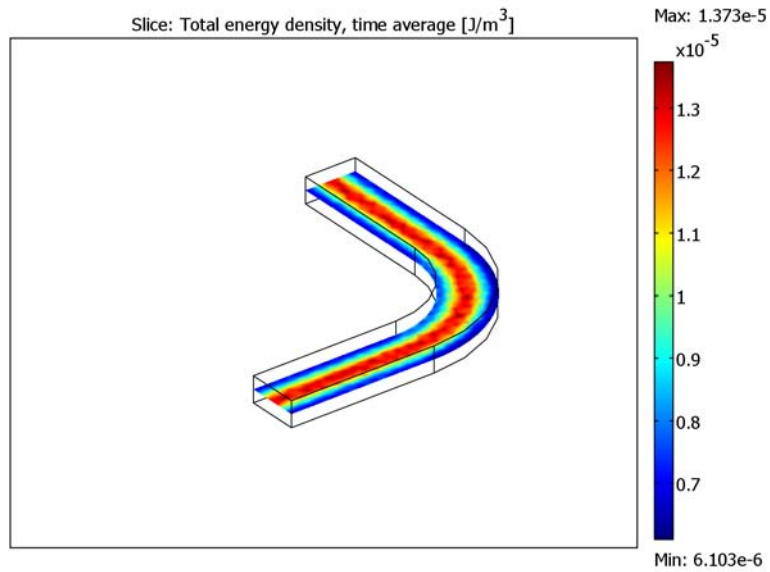
4 To better see the propagating wave, plot the z-component of the electric field. Select **Electric field, z-component** from the **Predefined quantities** list.

5 Choose the **wave** colormap and click **OK**.

Slice: Electric field, z component [V/m]

Max: 2130.958

Min: -2129.842
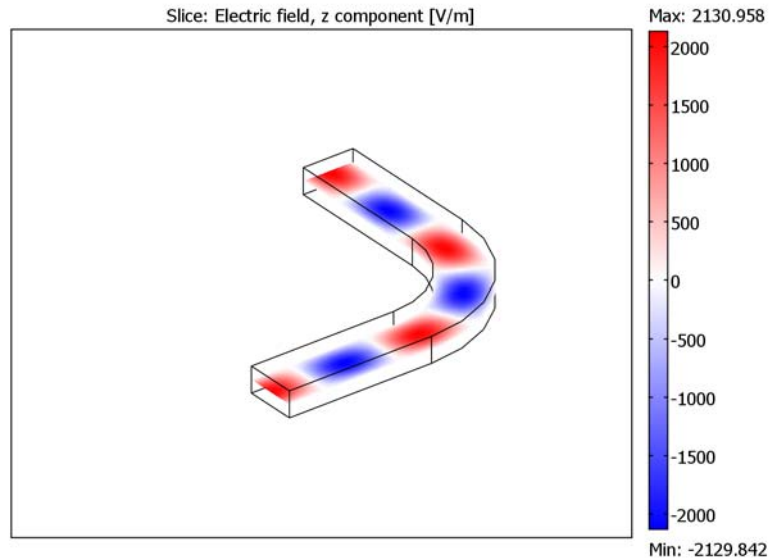
6 To compare the power flow of the incident wave to the power flow of the outgoing wave, perform a boundary integration. The entrance port is excited using the port boundary condition, which provides an excitation with a power level of 1 W. To obtain the power outflow from the exit port, open the **Boundary 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/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 "Waveguide H-Bend with S-parameters" on page 80 in the *RF Module Model Library*. See also the section "S-Parameters and Ports" on page 56 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** tab set the values according to the table below:

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

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 *z* component of the electric field. 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 (1m).
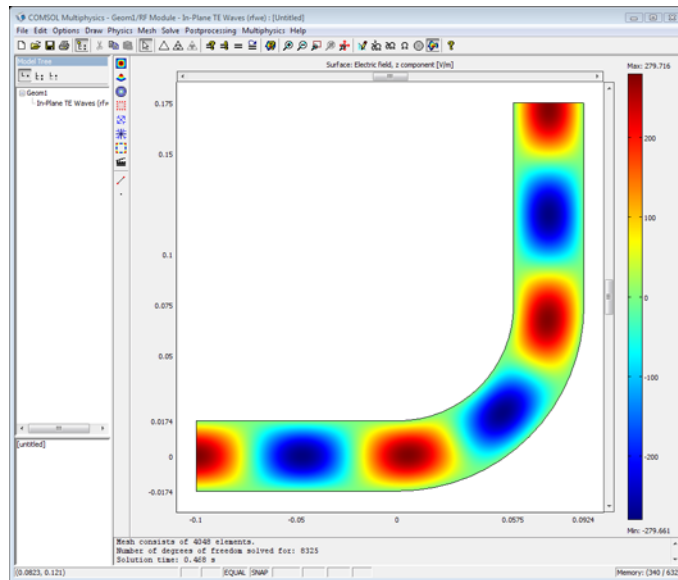
1 To plot the energy density, open the **Plot Parameters** dialog box.

2 Click the **Surface** tab, and select **Total energy density, time average** from the
Predefined quantities list in the **Surface data** area. Click **OK**.
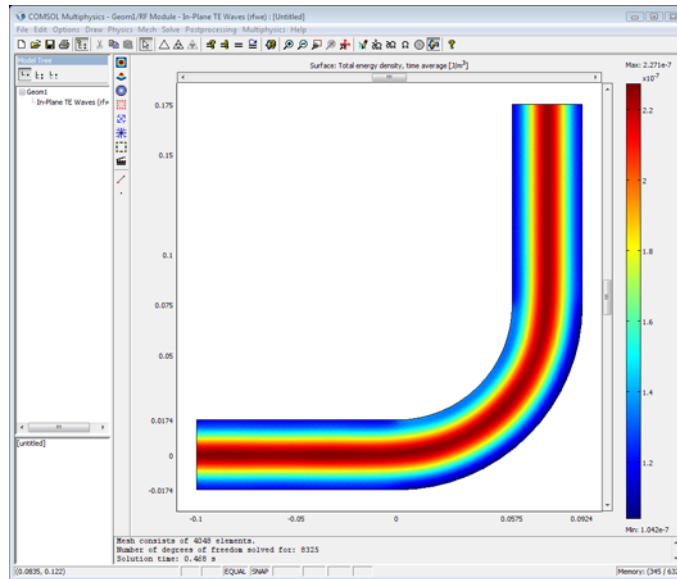
This again shows that the reflections are very small. To further verify this compare the
power flow of the incoming wave to the outgoing wave. To compare the power flow
of the incident wave to the power flow of the outgoing wave, perform a boundary
integration. The entrance port is excited using the 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 245 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 Ψ that implements an extra equation to explicitly set the divergence of the **D** or **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 84 for more details. This variable Ψ 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 set the electric field on the two boundaries parallel to the *yz*-plane equal, and then solve an eigenfrequency problem with this constraint.
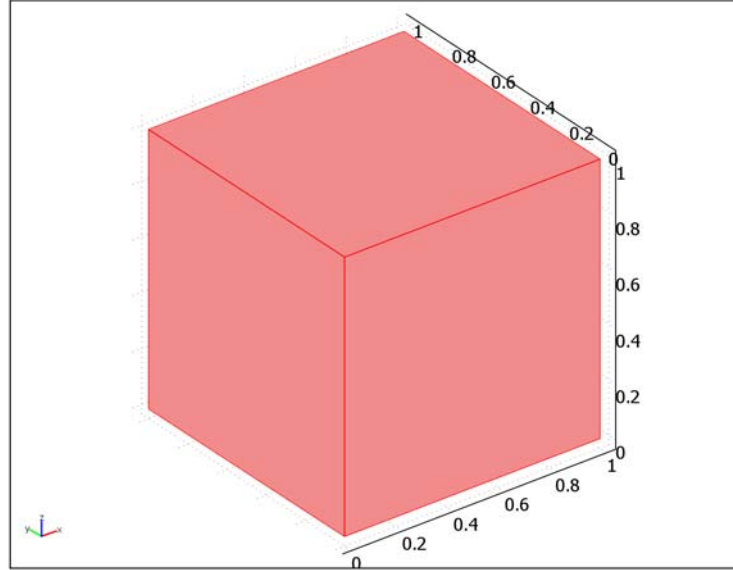
Figure 2-2: The cubic model geometry.

Results and Discussion

The cube has three degenerated eigenfrequencies: 150 MHz, 212 MHz, and 300 MHz. The plot in Figure 2-3 shows the *z* component of the electric field for a 300 MHz mode.
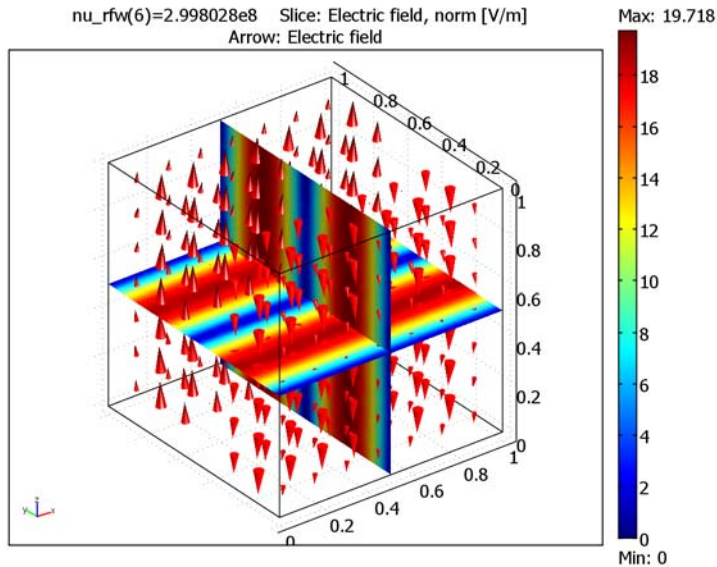
Figure 2-3: The electric field's z component.

Model Library path:
RF_Module/Tutorial_Models/periodic_boundary_condition

Modeling Using the Graphical User Interface

MODEL NAVIGATOR AND APPLICATION MODE PROPERTIES

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

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.

PHYSICS SETTINGS

Boundary Conditions

1 You specify the periodic condition in the **Physics>Boundary Settings** dialog box. Select the boundaries that define one periodic condition. These consist of one or more source boundaries plus one or more destination boundaries. The combined cross section of all source boundaries must be equal in shape to the combined cross section of all destination boundaries. If you want several periodic conditions with different orientations, separate them with a group index entered in the **Group index** edit field.

2 Select Boundaries 1 and 6, and choose **Periodic condition** from the **Boundary condition** list. Leave the **Type of periodicity** list at its default value, **Continuity**. The boundary with the lowest number becomes the source by default. It is possible to change this order by selecting the **Change source and destination order** check box.
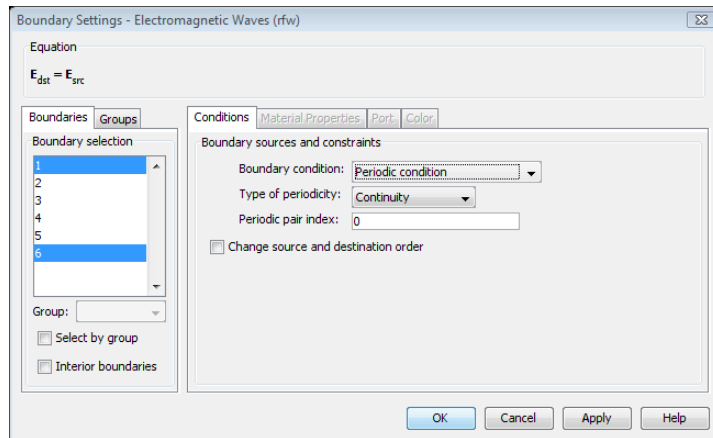
MESH GENERATION

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

2 On the **Boundary** tab, 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 boundaries** 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, enter 250e6 in the **Search for eigenfrequencies around** edit field.

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

POSTPROCESSING AND VISUALIZATION

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

2 Click the **Slice** tab.

3 Select **Electric field, norm** from the **Predefined quantities** list.

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

5 On the **Arrow** tab, select the **Arrow plot** check box and click **OK**.

6 Click the **Scene Light** button.

This will get you an arrow plot of the electric field and a slice plot of its norm, for the lowest eigenfrequency. You can browse between the solutions on the **General** page of the **Plot Parameters** dialog box. One of the 300 MHz eigenfrequencies will get you a plot resembling Figure 2-3.

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** tab in the **Solver Parameters** dialog box. This is the case for a periodic condition of type **Floquet periodicity** with a nonzero k-vector.
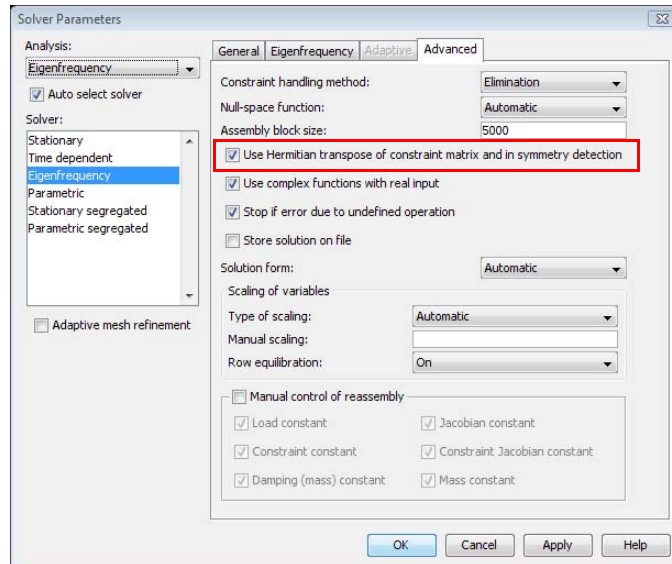
Perfectly Matched Layers (PMLs)

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, δt, 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.
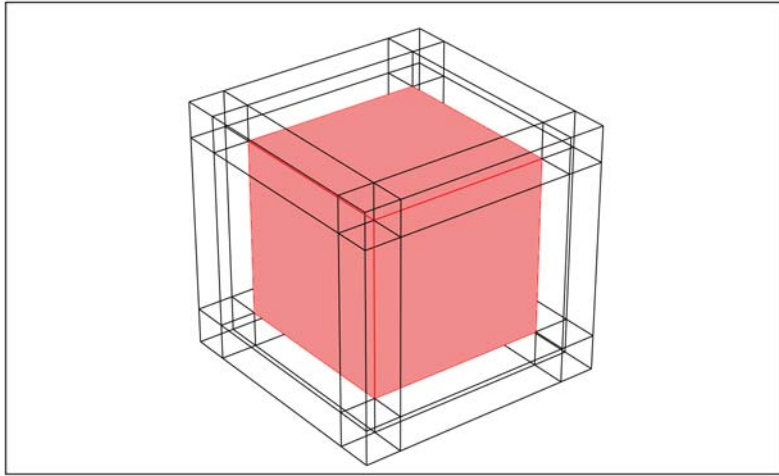
*Figure 2-4: A cube surrounded by typical PML regions of the type "Cartesian."*
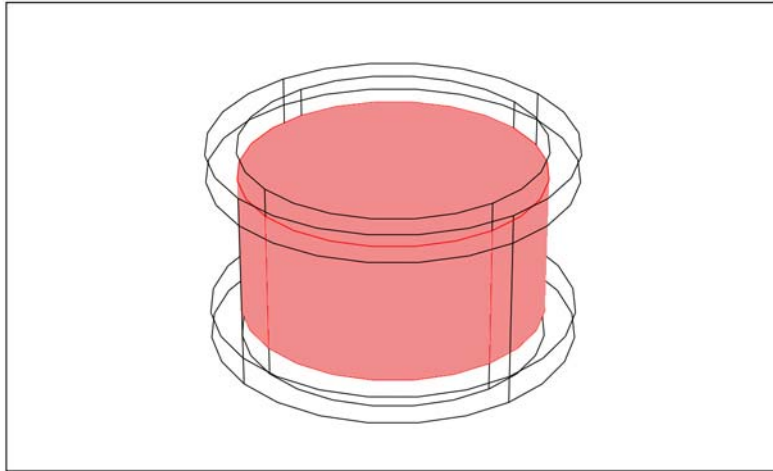
Figure 2-5: A cylinder surrounded by typical cylindrical PML regions.
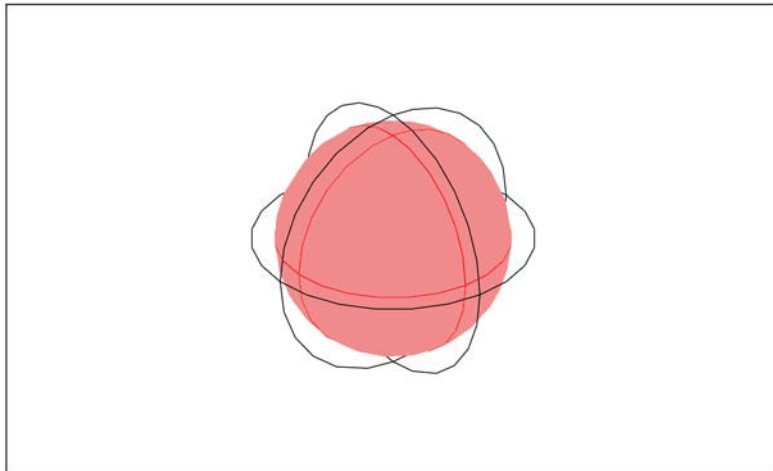
Figure 2-6: A sphere surrounded by a typical spherical PML region.

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.
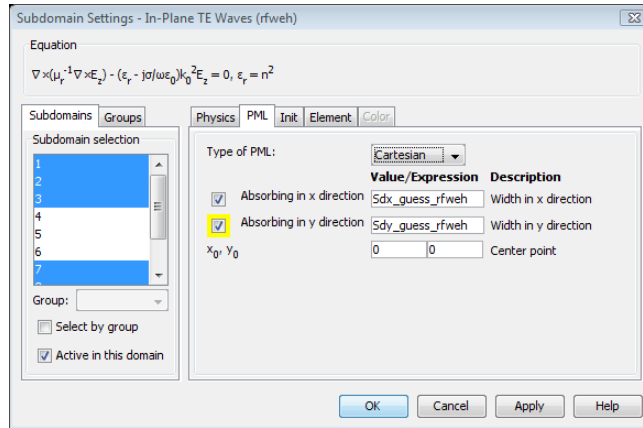
When the width is equal to the actual PML region width in the geometry, the coordinate scaling alters the wavelength so that it is equal to the PML region width. Especially for small geometries, where the size of the geometry is smaller than the wavelength, it might be better to keep the wavelength unchanged 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.
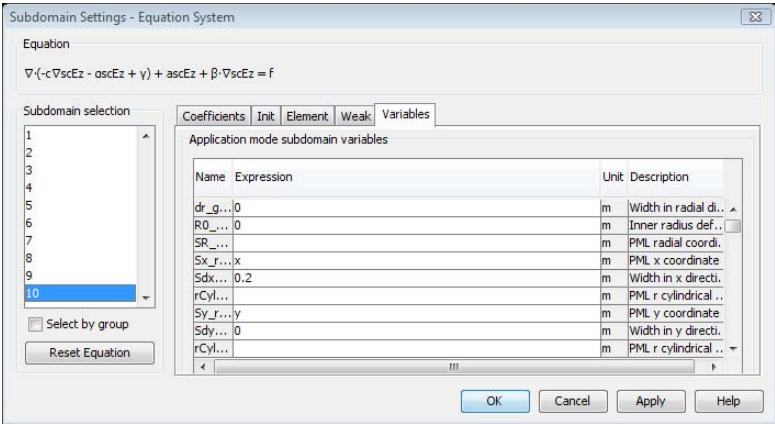
Reference

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 110 (3D cartesian PML).
- "Balanced Patch Antenna for 6 GHz" on page 136 (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 (θ, φ). The far-field radiation pattern is given by evaluating the squared norm of the far-field on a sphere centered at the origin. Each coordinate on the surface of the sphere represents an angular direction.

For the theory behind the far-field postprocessing, see "Far-Field Calculations" on page 99.

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** tab, 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.
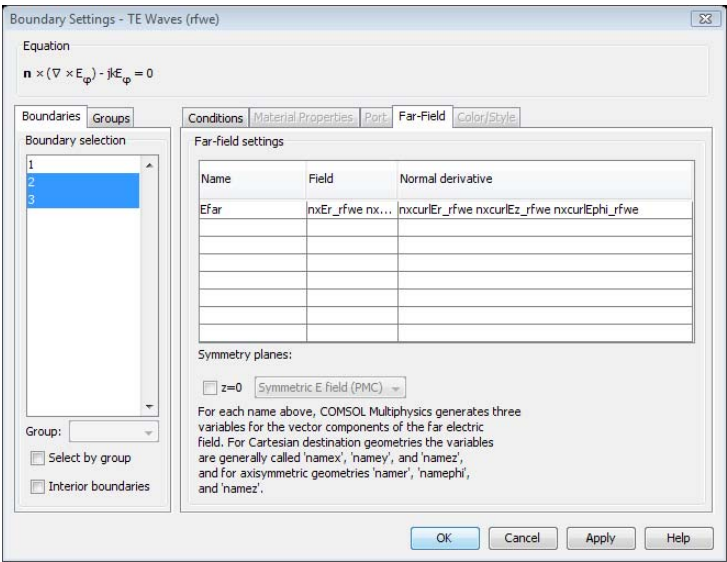
In this dialog box, you can also specify if you want to use symmetry planes in your model when calculating the far-field variable. The symmetry planes have to coincide with one of the Cartesian coordinate planes (the check boxes **x=0**, **y=0**, and **z=0**). For each of these planes it is possible 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 (θ, φ). 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 45 in the *RF Module Model Library.*
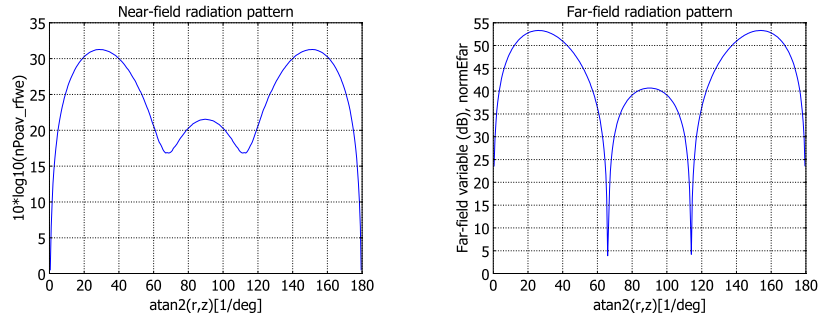
Figure 2-7: The near-field radiation pattern from the dipole (left), and the far-field radiation pattern (right).

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 $(0.2, 0)$.
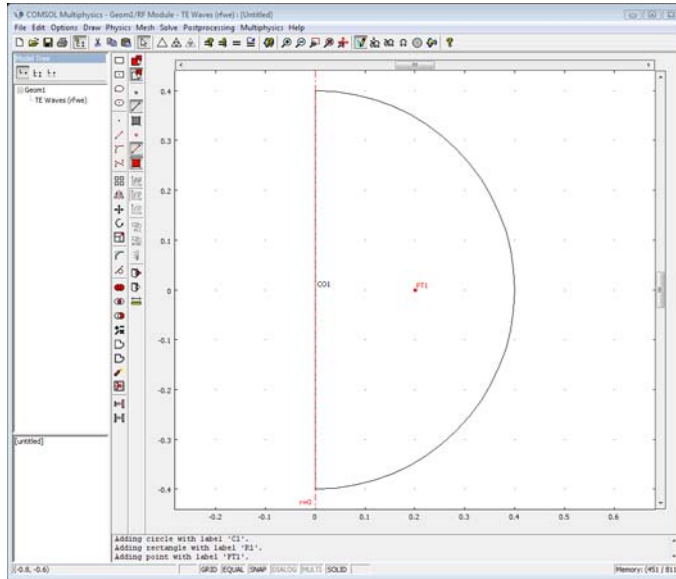
Boundary Conditions

1 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.

2 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.

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.

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.

The plot of the solution shows the electric field E_φ emitted from the circular loop.
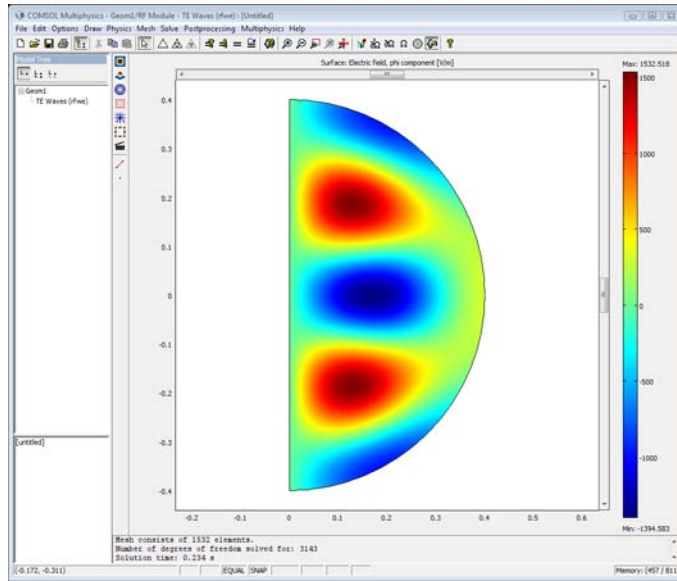
Far-Field Postprocessing

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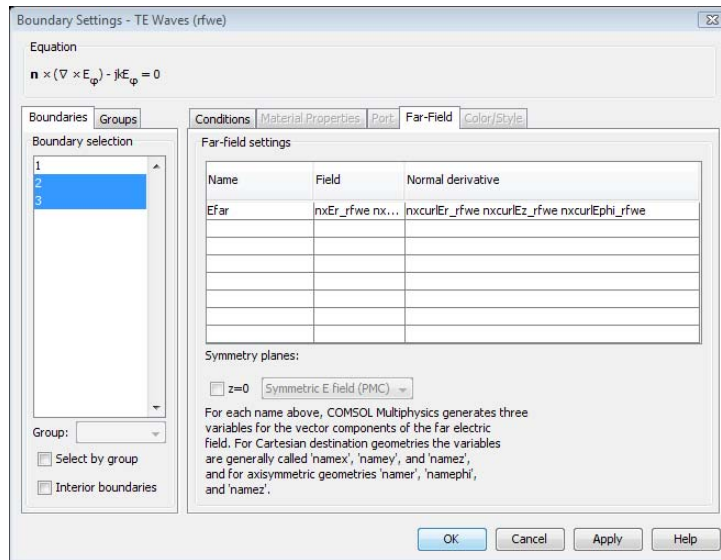
3 The far-field variables are not immediately available. They first have to be evaluated using the previously calculated solution. To do this, choose **Update Model** from the **Solve** menu.

The definition of the variable Efar generates a function, Efarphi(r,z), which is the φ 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 φ 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 **Apply** to plot the far-field radiation pattern on a dB scale.

Far-field radiation pattern
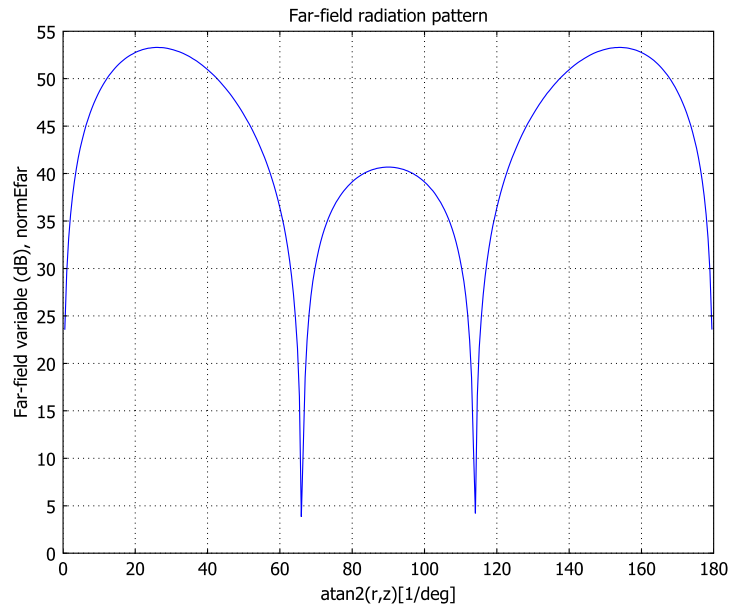
S-Parameters and Ports

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 95.

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** tab 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 TAB

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 in COMSOL Script and 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 TAB

The **Port** tab 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** tab. 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.
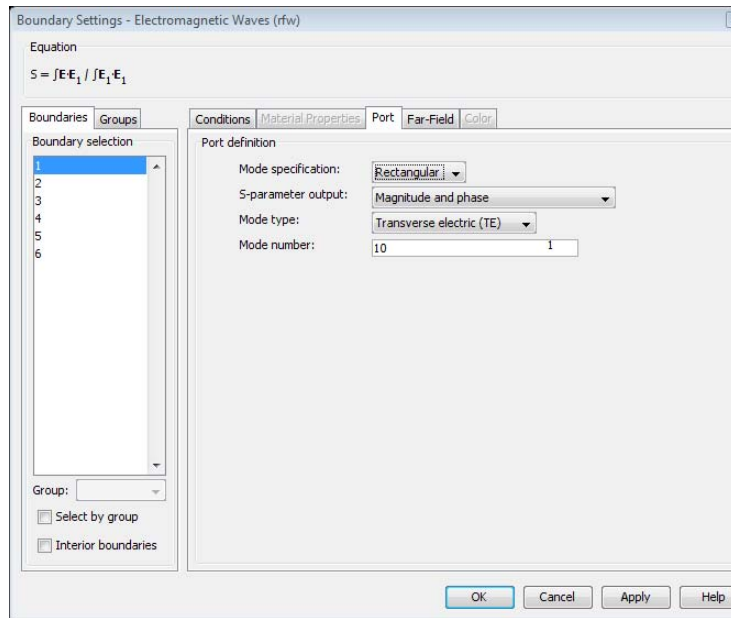
Coaxial Modes

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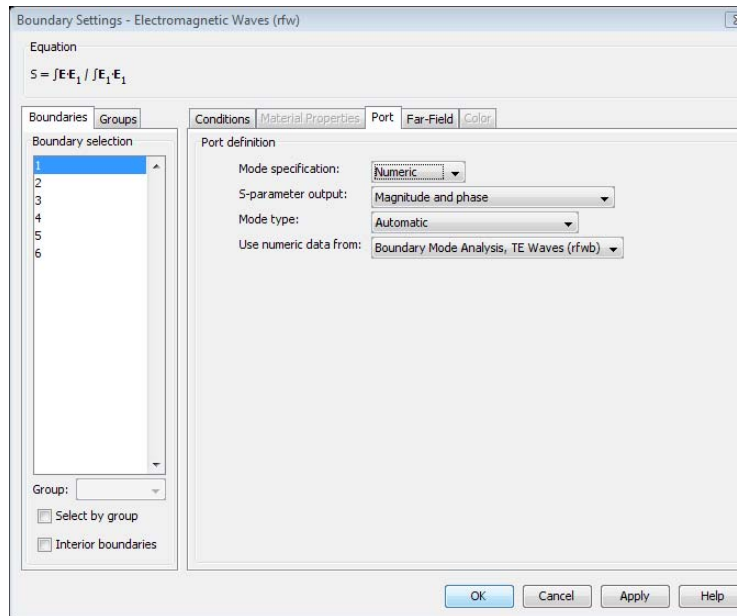
When using a numeric mode the mode analysis and the S-parameter analysis need to be done in sequence:

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.
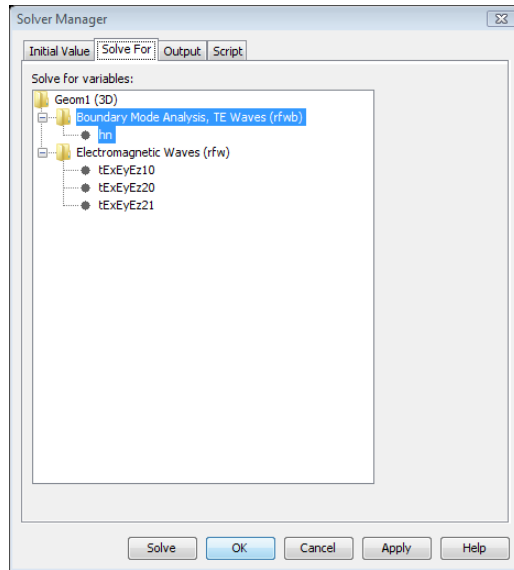
3 From the **Physics** menu, choose **Properties** to open the **Application Mode Properties** dialog box. From the **Specify eigenvalues using** list, choose **Manual eigenvalue** and click **OK**. When you perform this operation, you copy the propagation constant for solution currently plotted into 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.
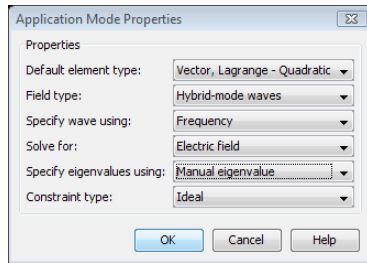
4 If you have several numeric ports, repeat steps 2and 3 for the other Boundary Mode Analysis application modes.

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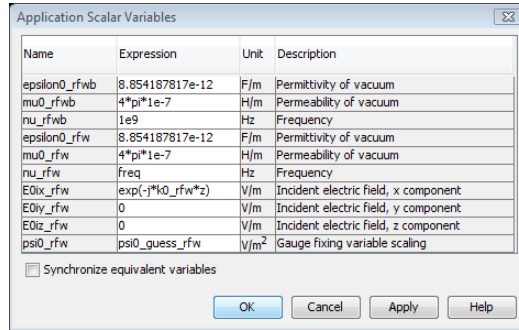
Figure 2-8: The Application Scalar Variables dialog box with the variables for the Boundary Mode Analysis and 3D Electromagnetic Waves application mode.

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 β 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 ν_0 and the propagation constant β_0 for that frequency. The propagation constant as a function of frequency is then calculated as

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

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

$$\beta = \sqrt{4\pi^2 \epsilon \mu (\nu^2 - \nu_{\text{cut-off}}^2)}$$

See Figure 2-9 on page 63, which shows where to enter these settings in the **Subdomain Settings** dialog box.
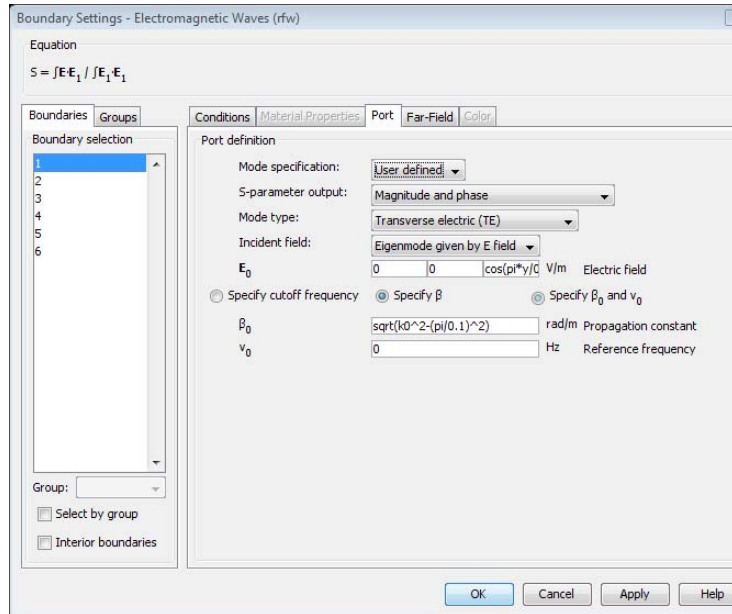
Figure 2-9: Specification of the mode type, electric field components, and the propagation constant.

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:

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

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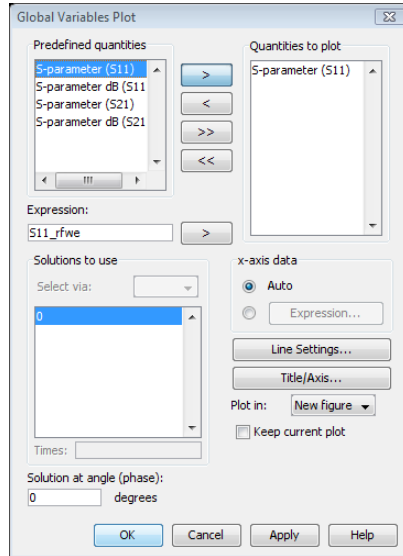
- **Global Data Display**. Open with **Postprocessing>Data Display>Global**.
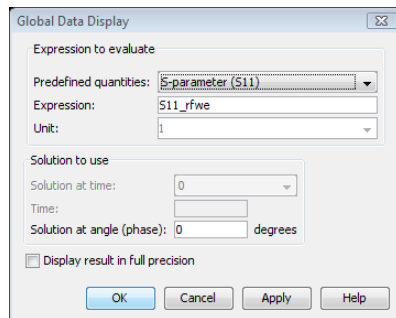
- **Domain Plot Parameters**. Open with **Postprocessing>Domain Plot Parameters**, and under the **Point** tab choose an arbitrary point.
- **Point Evaluation**. Open with **Postprocessing>Point Evaluation**. Choose an arbitrary point.

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** tab.

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 may 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 which becomes negative. The default solver settings do not allow you to take the square root of real numbers. To overcome the problem select the **Use complex functions with real input** check box on the **Advanced** tab 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 85 in the *RF Module Model Library*. The model "Microstrip on a Printed Circuit Board" on page 20 in 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 COMSOL Script 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 32 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 157 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_{port}, and S_{11} S-parameter for that port. The parameters are directly given by the relations

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

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

where V_{port} is the extracted voltage for the port given by the line integral between the terminals averaged over the entire port. The current I_{port} is the averaged total current over all cross sections parallel to the terminals. Ports not specified as input ports only return the extracted voltage and current. You can find more details on how to use voltages and currents in the boundary condition in "Lumped Port Parameters" on page 98.

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.

THE COEFFICIENT PAGE

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.

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 in the *RF Module Model Library*.

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 518 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 1st 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 70.

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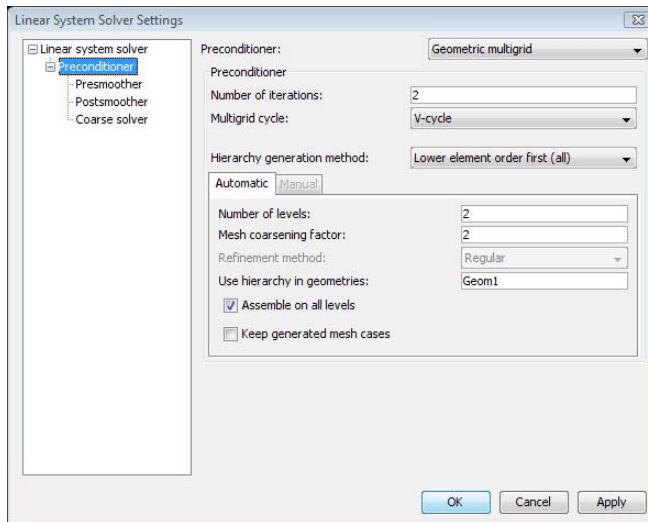
Figure 2-10: The multigrid settings for the GMG preconditioner.

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 1st order on the coarsest level and 2nd order on the finest level. This is a robust setting that often produce convergence. Another efficient option is to use 3rd order as the finest level and 1st order on the coarsest level. The main drawback with 3rd 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 2nd 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** tab 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 327 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.
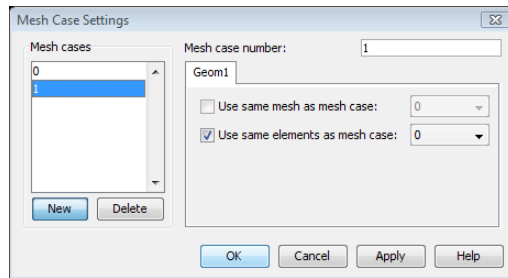
Figure 2-11: Settings for using the same mesh for both mesh cases.

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

On the **Manual** tab 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.
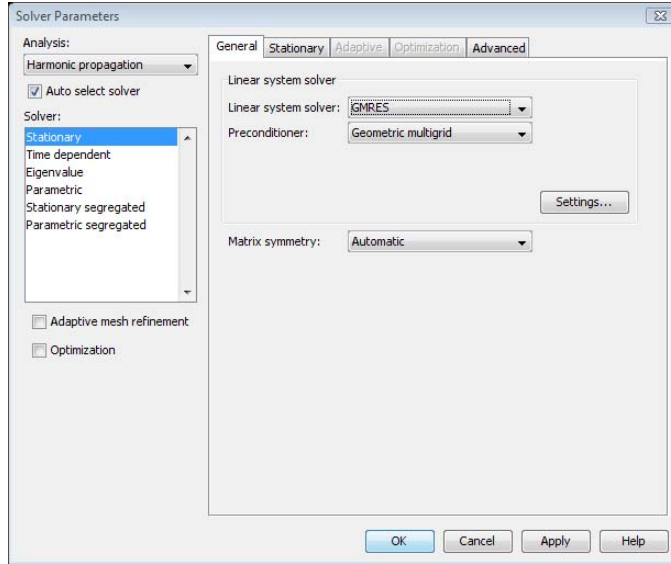
Figure 2-12: The default settings for electromagnetic waves.

Presmoother

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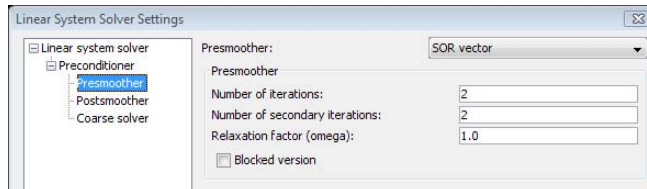
Figure 2-13: The default presmoother settings (figure shows part of dialog box).

Postsmoother

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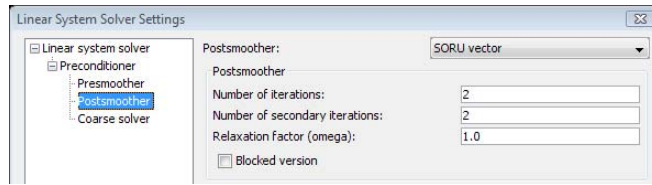
Figure 2-14: The default postsmoother settings (figure shows part of dialog box).

Coarse Solver

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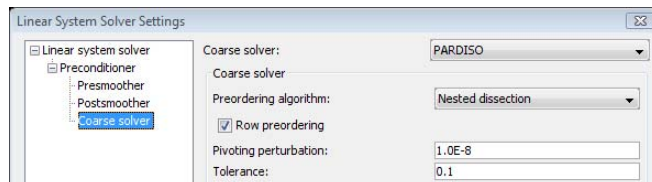
Figure 2-15: The default coarse solver settings (figure shows part of dialog box).

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 84). 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.
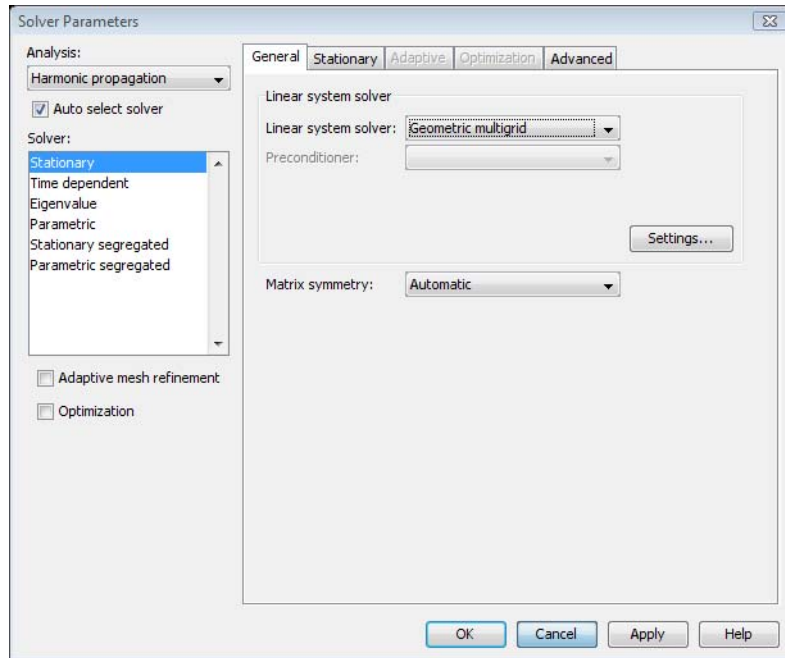
Figure 2-16: Solver settings for periodic 3D wave problems.

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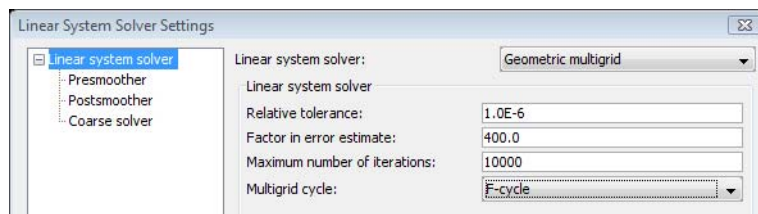
Figure 2-17: The multigrid settings for periodic 3D wave problems (the figure shows only the upper part of the dialog box).

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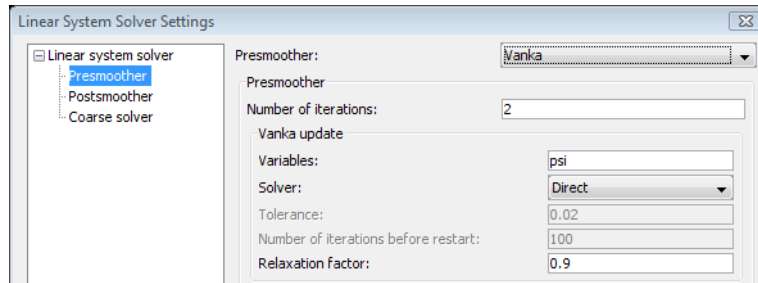
Figure 2-18: The default presmoother settings (the figure shows only the upper part of the dialog box).

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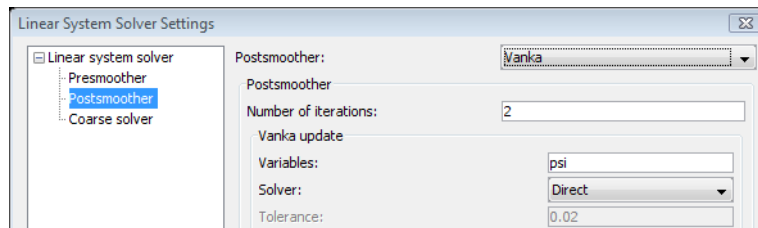
Figure 2-19: The default postsmoother settings (the figure shows only the upper part of the dialog box).

Coarse Solver

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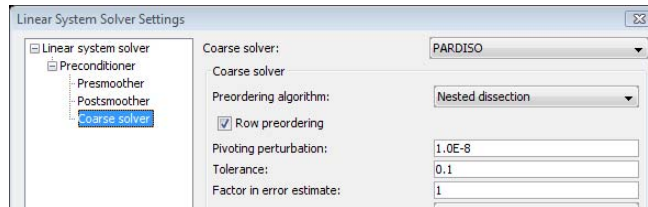
Figure 2-20: The coarse solver settings (the figure shows only the upper part of the dialog box).

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 376 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.
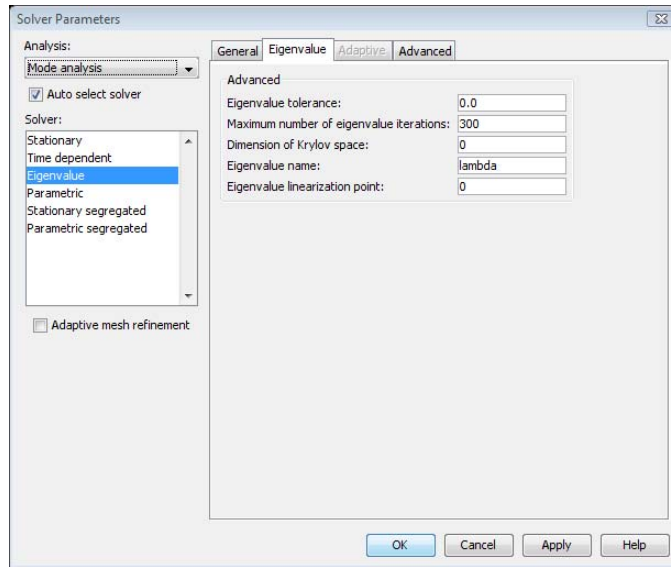
In many cases it is enough to specify a good linearization point and then solve the problem once. If you need a more accurate eigenvalue, an iterative scheme is necessary:

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.

A solver script provides the most convenient way to use this procedure.

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, δ_z, represents the damping along the propagation direction.

VARIABLES

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

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

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 **Material/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 81.

neff_rfwv(1)=0.930755-1.873328e-6i
Surface: Power flow, time average, z component [W/m^2]
Max: 5.727e5

Min: 344.921
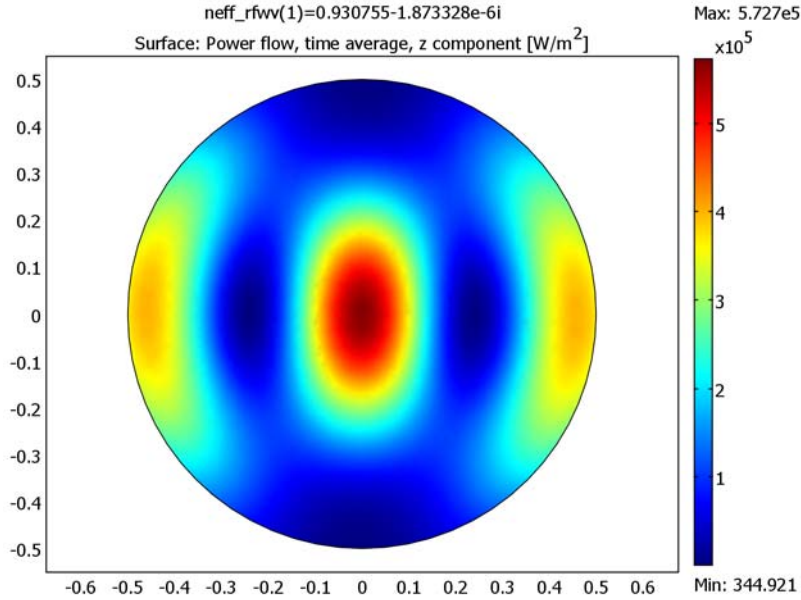
To calculate the attenuation values in the third column of the table, do as follows:

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 81.

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 69.

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 342 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 a 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 157 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* ρ.

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

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

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

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

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

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

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

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

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

Constitutive Relations

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

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

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

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

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

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

For linear materials, the polarization is directly proportional to the electric field, $\mathbf{P} = \varepsilon_0 \chi_e \mathbf{E}$, where χ_e is the *electric susceptibility.* Similarly in linear materials, the magnetization is directly proportional to the magnetic field, $\mathbf{M} = \chi_m \mathbf{H}$, where χ_m is the *magnetic susceptibility.* For such materials, the constitutive relations can be written

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

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

GENERALIZED CONSTITUTIVE RELATIONS

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

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

The field $\mathbf{D_r}$ 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* 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

$$P_e = \int_V \mathbf{E} \cdot \frac{\partial \mathbf{D}}{\partial t} dV$$

$$P_m = \int_V \mathbf{H} \cdot \frac{\partial \mathbf{B}}{\partial t} dV$$

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,

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

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

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

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

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

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

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

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

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

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

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

INTERFACE BETWEEN A DIELECTRIC AND A PERFECT CONDUCTOR

A perfect conductor has infinite electric conductivity and thus no internal electric field. Otherwise, it would produce an infinite current density according to the third fundamental constitutive relation. At an interface between a dielectric and a perfect conductor, the boundary conditions for the $\mathbf{E}$ and $\mathbf{D}$ fields are simplified. If, say, subscript 1 corresponds to the perfect conductor, then $\mathbf{D}_1 = \mathbf{0}$ and $\mathbf{E}_1 = \mathbf{0}$ in the relations above. For the general time-varying case, it holds that $\mathbf{B}_1 = \mathbf{0}$ and $\mathbf{H}_1 = \mathbf{0}$ as well (as a consequence of Maxwell's equations). What remains is the following set of boundary conditions for time-varying fields in the dielectric medium.

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

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 177 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 Tab" on page 57.

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,

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

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

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

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

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

where Z_{TE} is the wave impedance

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

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

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

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 ε is the permittivity. The power flow then becomes

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

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

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

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

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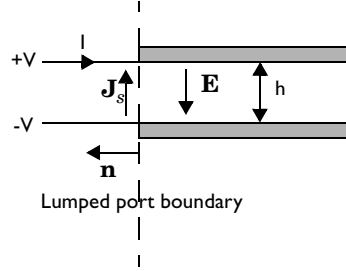
The transmission line current can be represented with a surface current at the lumped port boundary directed opposite to the electric field.

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 (θ, φ).

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.

- η_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 (θ, φ), so $\mathbf{E}_p$ is the far field for this direction.

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

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

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

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

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

Electromagnetic Quantities

The table below shows the symbol and SI unit for most of the physical quantities that appear in the RF Module. 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 μm as the length scale, you need to have $\varepsilon_0 = 8.854187817 \cdot 10^{-18}$ F/μm and $\mu_0 = 4\pi \cdot 10^{-13}$ H/μm.

TABLE 3-1: ELECTROMAGNETIC QUANTITIES

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

TABLE 3-1: ELECTROMAGNETIC QUANTITIES

QUANTITY	SYMBOL	UNIT	ABBREVIATION
Magnetic field	$\mathbf{H}$	ampere/meter	A/m
Magnetic flux	Φ	weber	Wb
Magnetic flux density	$\mathbf{B}$	tesla	T
Magnetic potential (scalar)	V_m	ampere	A
Magnetic potential (vector)	$\mathbf{A}$	weber/meter	Wb/m
Magnetic susceptibility	χ_m	(dimensionless)	–
Magnetization	$\mathbf{M}$	ampere/meter	A/m
Permeability	μ	henry/meter	H/m
Permittivity	ε	farad/meter	F/m
Polarization	$\mathbf{P}$	coulomb/meter^2	C/m^2
Poynting vector	$\mathbf{S}$	watt/meter^2	W/m^2
Propagation constant	β	radian/meter	rad/m
Reactance	X	ohm	Ω
Relative permeability	μ_r	(dimensionless)	–
Relative permittivity	ε_r	(dimensionless)	–
Resistance	R	ohm	Ω
Resistive loss	Q	watt/meter^3	W/m^3
Torque	T	newton-meter	Nm
Velocity	$\mathbf{v}$	meter/second	m/s
Wavelength	λ	meter	m
Wave number	k	radian/meter	rad/m

References

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

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

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

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

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

4

The 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 COMSOL Script or MATLAB. See "Programming Reference" on page 29 in the *RF Module Reference Guide* for details.

Application Mode Guide

Table 4-1 on page 107 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, φ, 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	111							
3D Electromagnetic Waves	112	rfw	$\mathbf{E}$ or $\mathbf{H}$	$x\,y\,z$	$x\,y\,z$	√	√	√
In-plane Waves, TE Modes	123	rfwe	E_z	$x\,y$	z	√	√	√
Axisymmetric Waves, TE Modes	132	rfwe	E_φ/r	$r\,z$	φ	√	√	√
In-Plane Waves, TM Modes	123	rfwh	H_z	z	$x\,y$	√	√	√
Axisymmetric Waves, TM Modes	132	rfwh	H_φ/r	φ	$r\,z$	√	√	√
In-Plane Waves, Hybrid Modes	123	rfweh	E_z, H_z	$x\,y\,z$	$x\,y\,z$	√	√	√
Axisymmetric Waves, Hybrid Modes	132	rfweh	E_φ/r, H_φ/r	$r\,\varphi$ z	$r\,\varphi$ z	√	√	√
Perpendicular Waves, TM Modes	138	rfwev	E_z	$x\,y$	$x\,y\,z$			√
Perpendicular Waves, TE Modes	138	rfwhv	H_z	$x\,y\,z$	$x\,y$			√
Perpendicular Waves, Hybrid Modes	138	rfwv	H_x, H_y, or $\mathbf{H}$, or E_x, E_y, or $\mathbf{E}$	$x\,y\,z$	$x\,y\,z$			√

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, TE Waves	147	rfwb	H_n	$x\,y\,z$	$x\,y\,z$			√
3D Boundary Mode Analysis, TM Waves	147	rfwb	E_n	$x\,y\,z$	$x\,y\,z$			√
3D Boundary Mode Analysis, Hybrid-Mode Waves	147	rfwb	$\mathbf{E}$, E_n or $\mathbf{H}$, H_n	$x\,y\,z$	$x\,y\,z$			√
2D Boundary Mode Analysis, TE Waves	150	rfwb	H_z	z	$x\,y$			√
2D Boundary Mode Analysis, TM Waves	150	rfwb	E_z	$x\,y$	z			√
2D Boundary Mode Analysis, TE Waves, Axisymmetry	150	rfwb	E_φ/r	$r\,z$	φ			√
2D Boundary Mode Analysis, TM Waves, Axisymmetry	150	rfwb	H_φ/r	φ	$r\,z$			√

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 depending on if the material is isotropic or anisotropic.

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, "The Application Modes," 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.

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 223 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 $\mathbf{E} \ = \ -\frac{\partial \mathbf{A}}{\partial t}$ makes 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

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

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

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

Alternatively, an equivalent conductivity can be defined 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 μ_r is 1, the relation between the complex refractive index

$$\bar{n} = n - j\kappa$$

and the complex relative permittivity is

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

that is

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

The inverse relations are

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

The parameter κ represents a damping of the electromagnetic wave.

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

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

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

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

where c_0 is the speed of light in vacuum.

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

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

Using the relation $\varepsilon_r = n^2$, where n is the refractive index, the 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 $\mathbf{E} = \mathbf{E}_{\text{sc}} + \mathbf{E}_i$ and $\mathbf{E}_{\text{sc}}$ 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 Eigenfrequency Transient	Specifies which type of analysis to perform.
Solve for	Electric field Magnetic field Scattered electric field Scattered magnetic field	Specifies which field is the dependent variable.
Specify wave using	Frequency Free-space wavelength	Specifies if the frequency or the wavelength should be used as input.
Specify eigenvalues using	Eigenvalue Eigenfrequency	Specifies if the solver parameters should be given in terms of the eigenvalue $\lambda = j\omega + \delta$ or the eigenfrequency ν.
Divergence condition	On 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
μ_0	mu0	4*pi*1e-7	Permeability of vacuum
ε_0	epsilon0	8.854187817e-12	Permittivity of vacuum
ν	nu	1e9	Frequency
λ_0	lambda0	0.3	Free-space wavelength
E0ix	E0ix	0	Incident electric field
E0iy	E0iy	0	Incident electric field
E0iz	E0iz	exp(-j*k*x)	Incident electric field
H0ix	H0ix	0	Incident magnetic field

PROPERTY	NAME	DEFAULT	DESCRIPTION
H0iy	H0iy	0	Incident magnetic field
H0iz	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 ν or the free-space wavelength λ_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 expression between the **H**-field and the **E**-field. Avoid specifying solution independent sources for these conditions, since 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 78.

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 **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 **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 **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 β 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 $\mathbf{H}_0$, represent 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 μ_1 and ε_1 are the material properties of the inner domain, is large, that is $|N| \gg 1$.

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

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 η 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 56 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 67 for more information how to use this condition.

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 26 for more details on this boundary condition.

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`, `t1z`. The condition is only available for the **H**-formulation.

APPLICATION MODE VARIABLES

See the section "3D Electromagnetic Waves Application Mode" on page 14 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 ε_r, σ, and μ_r are nondiagonal only in the xy-plane. μ_r denotes a 2-by-2 tensor, and ε_{rzz} and σ_{zz} are the relative permittivity and conductivity in the z direction.

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

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

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 μ_r and ε_r are nondiagonal only in the xy-plane. ε_r and σ denote 2-by-2 tensors, and μ_{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,

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

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.

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

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

APPLICATION SCALAR VARIABLES

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

PROPERTY	NAME	DEFAULT	DESCRIPTION
μ_0	mu0	4*pi*1e-7	Permeability of vacuum
ε_0	epsilon0	8.854187817e-12	Permittivity of vacuum
ν	nu	1e9	Frequency
λ_0	lambda0	0.3	Free space wavelength

All values are expressed in SI units. When solving the problem as a harmonic problem, the frequency ν or the free-space wavelength λ_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 around which point $r_0 = (x_0, y_0)$ the waves are cylindrical. The radius $r = \sqrt{(x - x_0)^2 + (y - y_0)^2}$.

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 β 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 η 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 67 for more information how to use this condition. 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 26 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 15 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 φ 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 φ 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 ε_r and μ_r are nondiagonal only in the rz-plane. μ_r denotes a 2-by-2 tensor, and $\varepsilon_{r\varphi\varphi}$ and $\sigma_{\varphi\varphi}$ are the relative permittivity and conductivity in the φ 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_φ,

$$-\begin{bmatrix} \dfrac{\partial}{\partial r} & \dfrac{\partial}{\partial z} \end{bmatrix} \cdot \left(\tilde{\mu}_r \begin{bmatrix} \dfrac{1}{r}\dfrac{\partial}{\partial r}(rE_\varphi) \\ \dfrac{\partial E_\varphi}{\partial z} \end{bmatrix} \right) - \varepsilon_{rc\varphi\varphi}k_0^2 E_\varphi = 0$$

where

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

and where the wave number in vacuum is

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

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

$$-\begin{bmatrix} \dfrac{\partial}{\partial r} & \dfrac{\partial}{\partial z} \end{bmatrix} \cdot \begin{bmatrix} \dfrac{1}{r}\dfrac{\partial}{\partial r}(rE_\varphi) \\ \dfrac{\partial E_\varphi}{\partial z} \end{bmatrix} - n_{\varphi\varphi}^2 k_0^2 E_\varphi = 0$$

The assumption when using the refractive index is that $\mu_r = 1$ and $\sigma = 0$.

The transient equation

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

can for TE waves be written

$$\mu_0 \sigma \frac{\partial A_\varphi}{\partial t} + \mu_0 \varepsilon_0 \frac{\partial}{\partial t}\left(\varepsilon_r \frac{\partial A_\varphi}{\partial t} - D_{r\varphi}\right) - \begin{bmatrix} \dfrac{\partial}{\partial r} & \dfrac{\partial}{\partial z} \end{bmatrix} \cdot \mu_r^{-1} \begin{bmatrix} \dfrac{1}{r}\dfrac{\partial}{\partial r}(rA_\varphi) - B_{rz} \\[2mm] \dfrac{\partial A_\varphi}{\partial z} + B_{rr} \end{bmatrix} = 0$$

using the constitutive relations $\mathbf{B} = \mu_0 \mu_r \mathbf{H} + \mathbf{B}_r$ and $\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E} + \mathbf{D}_r$. When using the relation $\varepsilon_r = n^2$, where n is the refractive index, the equation can be formulated the following way

$$\mu_0 \varepsilon_0 \frac{\partial}{\partial t}\left(n^2 \frac{\partial A_\varphi}{\partial t}\right) - \begin{bmatrix} \dfrac{\partial}{\partial r} & \dfrac{\partial}{\partial z} \end{bmatrix} \cdot \begin{bmatrix} \dfrac{1}{r}\dfrac{\partial}{\partial r}(rA_\varphi) \\[2mm] \dfrac{\partial A_\varphi}{\partial z} \end{bmatrix} = 0$$

In this case the application mode only handles the constitutive relations for linear materials.

Because there are coefficients where a division by r occurs, the PDE is not properly defined close to the z axis. A variable transformation avoids this problem:

$$u = \frac{E_\varphi}{r} \text{ or } u = \frac{A_\varphi}{r}$$

The resulting equations become

$$-\begin{bmatrix} \dfrac{\partial}{\partial r} & \dfrac{\partial}{\partial z} \end{bmatrix} \cdot \left(\tilde{\mu}_r \begin{bmatrix} r\dfrac{\partial u}{\partial r} + 2u \\[2mm] r\dfrac{\partial u}{\partial z} \end{bmatrix}\right) - r\varepsilon_{rc\varphi\varphi}k_0^2 u = 0$$

for either a harmonic problem or an eigenvalue problem with the eigenvalue $\lambda = k_0^2$, and

$$\mu_0\sigma\frac{\partial A_\varphi}{\partial t} + \mu_0\varepsilon_0\frac{\partial}{\partial t}\left(\varepsilon_r\frac{\partial A_\varphi}{\partial t} - D_{r\varphi}\right) - \left[\frac{\partial}{\partial r} \quad \frac{\partial}{\partial z}\right] \cdot \mu_r^{-1}\begin{bmatrix} r\frac{\partial u}{\partial r} + 2u - B_{rz} \\ \\ r\frac{\partial u}{\partial z} + B_{rr} \end{bmatrix} = 0$$

for a transient problem.

PDE FORMULATION FOR TM WAVES

TM waves has a magnetic field with only a φ 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 μ_r and ε_r are nondiagonal only in the rz-plane. ε_r and σ denote 2-by-2 tensors, and $\mu_{r\varphi\varphi}$ is the relative permeability is the φ 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_φ,

$$-\left[\frac{\partial}{\partial r} \quad \frac{\partial}{\partial z}\right] \cdot \left(\tilde{\varepsilon}_{rc}\begin{bmatrix} \dfrac{1}{r}\dfrac{\partial}{\partial r}(rH_\varphi) \\ \\ \dfrac{\partial H_\varphi}{\partial z} \end{bmatrix}\right) - \mu_{r\varphi\varphi}k_0^2 H_\varphi = 0$$

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

$$-\begin{bmatrix} \dfrac{\partial}{\partial r} & \dfrac{\partial}{\partial z} \end{bmatrix} \cdot \left(\tilde{n}^2 \begin{bmatrix} \dfrac{1}{r}\dfrac{\partial}{\partial r}(rH_\varphi) \\[2ex] \dfrac{\partial H_\varphi}{\partial z} \end{bmatrix} \right) - k_0^2 H_\varphi = 0$$

To avoid singularities at the axis of rotation, introduce the dependent variable u, which is defined as the angular component of the magnetic field divided by the radial space coordinate r, that is,

$$u = \frac{H_\varphi}{r}$$

Transforming the equation above using this variable gives the equation

$$-\begin{bmatrix} \dfrac{\partial}{\partial r} & \dfrac{\partial}{\partial z} \end{bmatrix} \cdot \left(\tilde{\varepsilon}_{rc} \begin{bmatrix} r\dfrac{\partial u}{\partial r} + 2u \\[2ex] r\dfrac{\partial u}{\partial z} \end{bmatrix} \right) - r\mu_{r\varphi\varphi}k_0^2 u = 0$$

which can be solved either as a stationary problem or as an eigenvalue problem with the eigenvalue $\lambda = k_0^2$.

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.

APPLICATION MODE PROPERTIES

See the corresponding section for In-Plane Waves on page 127.

APPLICATION SCALAR VARIABLES

See the corresponding section for In-Plane Waves on page 127.

BOUNDARY AND INTERFACE CONDITIONS

For a thorough discussion about the different boundary conditions, see "Boundary and Interface Conditions" on page 127 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 $r_s = \sqrt{r^2 + z^2}$.

Axial Symmetry

You must use the axial symmetry boundary condition along the z-axis, because at the symmetry axis, the meridional fields must satisfy the conditions

$$H_r = 0$$

$$\frac{\partial H_z}{\partial r} = 0$$

The relative permeability must also be scalar along the symmetry axis for the same conditions to be fulfilled for the magnetic flux density **B**.

In the time-harmonic case these conditions are equivalent to the following conditions on the dependent variable u.

$$j\omega\mu H_r = \frac{\partial E_\varphi}{\partial z} = r\frac{\partial u}{\partial z}$$

$$-j\omega\mu\frac{\partial H_z}{\partial r} = \frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(rE_\varphi)\right) = \frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r} + 2u\right) = r\frac{\partial^2 u}{\partial r} + 3\frac{\partial u}{\partial r}$$

The first condition is obviously always fulfilled, while the second one is fulfilled when r is zero, if and only if

$$\frac{\partial u}{\partial r} = 0$$

In the transient case using the variable u, the first of the above equations is automatically fulfilled whenever r is zero, which is shown by the identity

$$H_r = -\mu^{-1}\frac{\partial A_\varphi}{\partial z} = -\mu^{-1} r \frac{\partial u}{\partial z}$$

The second symmetry condition must be given explicitly. Writing the expression using the dependent variable, u, the equation for the symmetry condition becomes

$$\mu\frac{\partial H_z}{\partial r} = \frac{\partial}{\partial r}B_z = \frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(rA_\varphi)\right) = \frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r} + 2u\right) = r\frac{\partial^2 u}{\partial r} + 3\frac{\partial u}{\partial r}$$

This is zero when r is zero if and only if

$$\frac{\partial u}{\partial r} = 0$$

This derivation holds for TE waves. For TM waves a corresponding boundary condition can be derived using the requirements that

$$E_r = 0$$

$$\frac{\partial E_z}{\partial r} = 0$$

at the symmetry axis.

APPLICATION MODE VARIABLES

See the section "Axisymmetric Waves Application Mode" on page 17 of the *RF Module Reference Guide*.

Perpendicular Waves Application Mode

In the application mode describing perpendicular electromagnetic waves, 3D waves that can be described as

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

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

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

Various formulations derived from Maxwell-Ampère's and Faraday's laws

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

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

are used in the perpendicular electromagnetic waves application mode.

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 ω 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, μ and ε 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 μ and ε 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

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

where both ε_{rc} and μ_r are scalar constants. The wave number in vacuum is here defined by

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

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

$$-\nabla \cdot (\nabla E_z) - n^2 k_0^2 E_z = -\beta^2 E_z$$

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

PERPENDICULAR TE WAVES

As in the TM case, perpendicular TE waves requires that the medium is homogeneous, that is, μ and ε 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 μ and ε 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

$$-\nabla \cdot (\varepsilon_{rc}^{-1} \nabla H_z) - \mu_r k_0^2 H_z = -\beta^2 \varepsilon_{rc}^{-1} H_z ,$$

Both the permeability and the permittivity are scalar constants. Using the relation $\varepsilon_r = n^2$, 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

$$-\mu_0 \int_\Omega (j\omega\mathbf{J}_T \cdot \tilde{\mathbf{E}}_T - j\omega\mathbf{H}_T \cdot \mathrm{curl}\tilde{\mathbf{E}}_T - j\omega H_z \cdot \mathrm{curl}\tilde{E}_{Tz})d\Omega$$

$$+j\omega\mu_0 \oint_{\partial\Omega} (\mathbf{n}\times\mathbf{H}\cdot\tilde{\mathbf{E}}_T)d\partial\Omega - \mu_0\int_\Omega ((j\omega J_{iz})\cdot(-\alpha\tilde{e}_z) - j\omega\mathbf{H}_T\cdot\mathrm{curl}\tilde{\mathbf{E}}_{zT})d\Omega$$

$$+j\omega\mu_0 \oint_{\partial\Omega} (\mathbf{n}\times\mathbf{H}\cdot\mathbf{a}_z(-\alpha\tilde{e}_z))d\partial\Omega$$

where α 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 δ_z and δ become nonzero. These terms are available as the variables dampz and damp, respectively.

Two-Component Formulation
In the following, let $\varepsilon_{\mathrm{rct}}$ and μ_{rt} denote the 2-by-2 tensors in the transversal xy-plane,

$$\varepsilon_{\mathrm{rct}} = \begin{bmatrix} \varepsilon_{\mathrm{rcxx}} & \varepsilon_{\mathrm{rcxy}} \\ \varepsilon_{\mathrm{rcyx}} & \varepsilon_{\mathrm{rcyy}} \end{bmatrix} \qquad \mu_{\mathrm{rt}} = \begin{bmatrix} \mu_{\mathrm{rxx}} & \mu_{\mathrm{rxy}} \\ \mu_{\mathrm{ryx}} & \mu_{\mathrm{ryy}} \end{bmatrix}$$

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

$$\nabla_t\cdot\mu_{\mathrm{rt}}\mathbf{H}_t - j\beta\mu_{\mathrm{rzz}}H_z = 0$$

Substituting this into the equation

$$\nabla_t\times(\varepsilon_{\mathrm{rczz}}^{-1}\nabla_t\times\mathbf{H}_t) - j\beta\tilde{\varepsilon}_{\mathrm{rct}}\nabla_t H_z - (k_0^2\mu_{\mathrm{rt}} - \beta^2\tilde{\varepsilon}_{\mathrm{rct}})\mathbf{H}_t = \mathbf{0}$$

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 $\mathbf{H}$ by $\mathbf{E}$ and interchange ε_c and μ.

APPLICATION MODE PROPERTIES

The following table lists the properties for this application mode:

PROPERTY	VALUES	DESCRIPTION
Analysis type	Harmonic propagation Eigenfrequency	Specifies which type of analysis to perform.
Field type	TE waves TM waves Hybrid-mode waves	Specifies which type of waves to handle.
Specify wave using	Frequency Free-space wavelength	Specifies if the frequency or the wavelength should be used as input.
Solve for	Electric field Magnetic field	Specifies which field is the dependent variable.
Field components	In-plane components All three components	Specifies if the two or three-component formulation is used for hybrid-mode waves.
Specify eigenvalues using	Eigenvalue Eigenfrequency Propagation constant 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
μ_0	mu0	4*pi*1e-7	H/m	Permeability of vacuum
ε_0	epsilon0	8.854187817e-12	F/m	Permittivity of vacuum
ν	nu	1e9	Hz	Frequency
λ_0	lambda0	0.3	m	Free-space wavelength
β	beta	30	-	Propagation constant

The constants ε_0 and μ_0 are expressed in SI units. When solving the problem as an eigenvalue problem for β, you must specify the frequency ν or the free-space wavelength λ_0, and β is obtained from the solution. When solving the problem as an eigenvalue problem for ν, you must specify the propagation constant β. In this case, ν 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 **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 μ 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 μ 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 μ 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 μ is everywhere continuous. Similarly the two-component formulation for the electric field requires that ε_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. ε	DISCONT. μ	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 Mode" on page 20 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 ∇_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

$$\nabla_t \times (\varepsilon_{rc}^{-1} \nabla_t \times \mathbf{h}_t) - k_0^2 \mu_r \mathbf{h}_t = -\beta^2 \varepsilon_{rc}^{-1} (\nabla h_n / k_0 + \mathbf{h}_t)$$

$$0 = \beta^2 \left(\frac{1}{k_0} \nabla_t \cdot \varepsilon_{rc}^{-1} (\nabla h_n / k_0 + \mathbf{h}_t) + \mu_r h_n \right)$$

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 141. 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 ε_c and μ.

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

PROPERTY	VALUES	DESCRIPTION
Field type	TE waves TM waves Hybrid-mode waves	Specifies which type of waves to handle.
Specify wave using	Frequency Free-space wavelength	Specifies if the frequency or the wavelength should be used as input.
Solve for	Electric field Magnetic field	Specifies which field is the dependent variable.
Specify eigenvalues using	Eigenvalue Propagation constant 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
μ_0	mu0	4*pi*1e-7	H/m	Permeability of vacuum
ε_0	epsilon0	8.854187817e-12	F/m	Permittivity of vacuum
ν	nu	1e9	Hz	Frequency
λ_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 Mode in 3D" on page 24 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 ∇_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 TM waves	Specifies which type of waves to handle.

PROPERTY	VALUES	DESCRIPTION
Specify wave using	Frequency Free-space wavelength	Specifies if the frequency or the wavelength should be used as input.
Specify eigenvalues using	Eigenvalue Propagation constant 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
μ_0	mu0	4*pi*1e-7	H/m	Permeability of vacuum
ε_0	epsilon0	8.854187817e-12	F/m	Permittivity of vacuum
ν	nu	1e9	Hz	Frequency
λ_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 Mode in 2D" on page 26 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