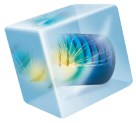


RF Module

User's Guide

VERSION 4.3b

COMSOL

RF Module User's Guide

Version: May 2013 COMSOL 4.3b

Contact Information

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

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

Other useful links include:

- Support Center: www.comsol.com/support
- Download COMSOL: www.comsol.com/support/download
- Product Updates: www.comsol.com/support/updates
- COMSOL Community: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Center: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part No. CM021001

Contents

Chapter 1: Introduction

Chapter 2: RF Modeling

Chapter 3: Electromagnetics Theory

Chapter 4: The Radio Frequency Branch

Chapter 5: The ACDC Branch

Chapter 6: The Electromagnetic Heating Branch

Chapter 7: Glossary

1

Introduction

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

This chapter introduces you to the capabilities of this module. A summary of the physics interfaces and where you can find documentation and model examples is also included. The last section is a brief overview with links to each chapter in this guide.

About the RF Module

In this section:

Overview of the Physics and Building a COMSOL Model in the *COMSOL Multiphysics Reference Manual*

What Can the RF Module Do?

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

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

- Antennas
- Waveguides and cavity resonators in microwave engineering
- Optical fibers
- Photonic waveguides
- Photonic crystals
- Active devices in photonics

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

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

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

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

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

What Problems Can You Solve?

QUASI-STATIC AND HIGH FREQUENCY MODELING

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

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

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

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

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

The RF Module Physics Guide

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

- See Simplifying Geometries for more information about selecting the right space dimension for the model.

In the *COMSOL Multiphysics Reference Manual:*

- Studies and the Study Nodes
- The Physics User Interfaces
- For a list of all the interfaces included with the COMSOL basic license, see Physics Guide.

PHYSICS USER INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
AC/DC				
Electrical Circuit		cir	Not space dependent	stationary; frequency domain; time dependent
Heat Transfer				
Electromagnetic Heating				
Microwave Heating		mh	3D, 2D, 2D axisymmetric	boundary mode analysis; frequency-stationary; frequency transient

PHYSICS USER INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Radio Frequency				
Electromagnetic Waves, Frequency Domain		emw	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal; boundary mode analysis; mode analysis (2D and 2D axisymmetric models only)
Electromagnetic Waves, Time Explicit		ewte	3D, 2D, 2D axisymmetric	time dependent
Electromagnetic Waves, Transient		temw	3D, 2D, 2D axisymmetric	eigenfrequency; time dependent; time dependent modal
Transmission Line		tl	3D, 2D, 1D	eigenfrequency; frequency domain

SELECTING THE STUDY TYPE

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

Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

COMPARING THE TIME DEPENDENT AND FREQUENCY DOMAIN STUDIES

When variations in time are present there are two main approaches to represent the time dependence. The most straightforward is to solve the problem by calculating the changes in the solution for each time step; that is, solving using the Time Dependent study (available with the Electromagnetic Waves, Transient interface). However, this approach can be time consuming if small time steps are necessary for the desired accuracy. It is necessary when the inputs are transients like turn-on and turn-off sequences.

However, if the Frequency Domain study available with the Electromagnetic Waves, Frequency Domain interface is used, this allows you to efficiently simplify and assume that all variations in time occur as sinusoidal signals. Then the problem is

time-harmonic and in the frequency domain. Thus you can formulate it as a stationary problem with complex-valued solutions. The complex value represents both the amplitude and the phase of the field, while the frequency is specified as a scalar model input, usually provided by the solver. This approach is useful because, combined with Fourier analysis, it applies to all periodic signals with the exception of nonlinear problems. Examples of typical frequency domain simulations are wave-propagation problems like waveguides and antennas.

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

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

THE RF MODULE MODELING PROCESS

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

1. Selecting the appropriate physics interface or predefined multiphysics coupling in the **Model Wizard**.
2. Defining model parameters and variables in the **Definitions** branch ( ).
3. Drawing or importing the model geometry in the **Geometry** branch ().
4. Assigning material properties to the geometry in the **Materials** branch ().
5. Setting up the model equations and boundary conditions in the physics interfaces branch.
6. Meshing in the **Mesh** branch ().
7. Setting up the study and computing the solution in the **Study** branch ().
8. Analyzing and visualizing the results in the **Results** branch ().

Even after a model is defined, you can edit to input data, equations, boundary conditions, geometry—the equations and boundary conditions are still available through associative geometry—and mesh settings. You can restart the solver, for example, using the existing solution as the initial condition or initial guess. It is also

easy to add another interface to account for a phenomenon not previously described in a model.

- Building a COMSOL Model in the *COMSOL Multiphysics Reference Manual*
- The RF Module Physics Guide
- Selecting the Study Type

SHOW MORE PHYSICS OPTIONS

There are several general options available for the physics user interfaces and for individual nodes. This section is a short overview of these options, and includes links to additional information when available.

The links to the features described in the *COMSOL Multiphysics Reference Manual* (or any external guide) do not work in the PDF, only from within the online help.

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

To display additional options for the physics interfaces and other parts of the model tree, click the **Show** button ( ) on the **Model Builder** and then select the applicable option.

After clicking the **Show** button ( ), additional sections get displayed on the settings window when a node is clicked and additional nodes are available from the context menu when a node is right-clicked. For each, the additional sections that can be displayed include **Equation**, **Advanced Settings**, **Discretization**, **Consistent Stabilization**, and **Inconsistent Stabilization**.

You can also click the **Expand Sections** button ( ) in the **Model Builder** to always show some sections or click the **Show** button ( ) and select **Reset to Default** to reset to display only the **Equation** and **Override and Contribution** sections.

For most nodes, both the **Equation** and **Override and Contribution** sections are always available. Click the **Show** button ( ) and then select **Equation View** to display the **Equation View** node under all nodes in the **Model Builder**.

Availability of each node, and whether it is described for a particular node, is based on the individual selected. For example, the **Discretization**, **Advanced Settings**, **Consistent Stabilization**, and **Inconsistent Stabilization** sections are often described individually throughout the documentation as there are unique settings.

SECTION	CROSS REFERENCE
Show More Options and Expand Sections	Advanced Physics Sections The Model Wizard and Model Builder
Discretization	Show Discretization Discretization (Node)
Discretization—Splitting of complex variables	Compile Equations
Consistent and Inconsistent Stabilization	Show Stabilization Numerical Stabilization
Constraint Settings	Weak Constraints and Constraint Settings
Override and Contribution	Physics Exclusive and Contributing Node Types

OTHER COMMON SETTINGS

At the main level, some of the common settings found (in addition to the Show options) are the Interface Identifier, Domain, Boundary, or Edge Selection, and Dependent Variables.

At the nodes' level, some of the common settings found (in addition to the Show options) are Domain, Boundary, Edge, or Point Selection, Material Type, Coordinate System Selection, and Model Inputs. Other sections are common based on application area and are not included here.

SECTION	CROSS REFERENCE
Coordinate System Selection	Coordinate Systems
Domain, Boundary, Edge, and Point Selection	About Geometric Entities About Selecting Geometric Entities

SECTION	CROSS REFERENCE
Interface Identifier	Predefined Physics Variables Variable Naming Convention and Scope Viewing Node Names, Identifiers, Types, and Tags
Material Type	Materials
Model Inputs	About Materials and Material Properties Selecting Physics Adding Multiphysics Couplings
Pair Selection	Identity and Contact Pairs Continuity on Interior Boundaries

Where Do I Access the Documentation and Model Library?

A number of Internet resources provide more information about COMSOL, including licensing and technical information. The electronic documentation, context help, and the Model Library are all accessed through the COMSOL Desktop.

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

THE DOCUMENTATION

The *COMSOL Multiphysics Reference Manual* describes all user interfaces and functionality included with the basic COMSOL Multiphysics license. This book also has instructions about how to use COMSOL and how to access the documentation electronically through the COMSOL Help Desk.

To locate and search all the documentation, in COMSOL Multiphysics:

- Press F1 or select **Help>Help** () from the main menu for context help.

- Press Ctrl+F1 or select **Help>Documentation** ( ) from the main menu for opening the main documentation window with access to all COMSOL documentation.
- Click the corresponding buttons (or) on the main toolbar.

and then either enter a search term or look under a specific module in the documentation tree.

If you have added a node to a model you are working on, click the **Help** button ( ) in the node's settings window or press F1 to learn more about it. Under **More results** in the **Help** window there is a link with a search string for the node's name. Click the link to find all occurrences of the node's name in the documentation, including model documentation and the external COMSOL website. This can help you find more information about the use of the node's functionality as well as model examples where the node is used.

THE MODEL LIBRARY

Each model comes with documentation that includes a theoretical background and step-by-step instructions to create the model. The models are available in COMSOL as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual models as a template for your own modeling and applications.

In most models, SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

To open the Model Library, select **View>Model Library** ( ) from the main menu, and then search by model name or browse under a module folder name. Click to highlight any model of interest, and select **Open Model and PDF** to open both the model and the documentation explaining how to build the model. Alternatively, click the **Help** button ( ) or select **Help>Documentation** in COMSOL to search by name or browse by module.

The model libraries are updated on a regular basis by COMSOL in order to add new models and to improve existing models. Choose **View>Model Library Update** ( ) to update your model library to include the latest versions of the model examples.

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

CONTACTING COMSOL BY EMAIL

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

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

COMSOL WEBSITES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
Support Center	www.comsol.com/support
Download COMSOL	www.comsol.com/support/download
Support Knowledge Base	www.comsol.com/support/knowledgebase
Product Updates	www.comsol.com/support/updates
COMSOL Community	www.comsol.com/community

Overview of the User's Guide

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

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

TABLE OF CONTENTS, GLOSSARY, AND INDEX

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

MODELING WITH THE RF MODULE

The RF Modeling chapter familiarize you with the modeling procedures. A number of models available through the Model Library also illustrate the different aspects of the simulation process. Topics include Preparing for RF Modeling, Simplifying Geometries, and Scattered Field Formulation.

RF THEORY

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

RADIO FREQUENCY

The Radio Frequency Branch chapter describes:

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

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

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

ELECTRICAL CIRCUIT

The ACDC Branch chapter describes The Electrical Circuit User Interface, which simulates the current in a conductive and capacitive material under the influence of an electric field. All three study types (stationary, frequency domain, and time-dependent) are available. The underlying theory is also included at the end of the chapter.

HEAT TRANSFER

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

2

RF Modeling

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

In this chapter:

Preparing for RF Modeling

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

This section is intended to help answer questions such as:

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

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

Overview of the Physics and Building a COMSOL Model in the *COMSOL Multiphysics Reference Manual*

Simplifying Geometries

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

In this section:

2D Models

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

CARTESIAN COORDINATES

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

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

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

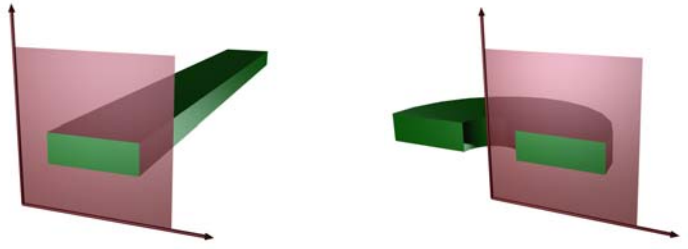


Figure 2-1: The cross sections and their real geometry for Cartesian coordinates and cylindrical coordinates (axial symmetry).

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

AXIAL SYMMETRY (CYLINDRICAL COORDINATES)

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

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

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

POLARIZATION IN 2D

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

3D Models

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

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

Using Efficient Boundary Conditions

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

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

  A more accurate option is to use perfectly matched layers (PMLs). PMLs are layers that absorbs all radiated waves with small reflections.
- Replace thin layers with boundary conditions where possible. There are several types of boundary conditions in COMSOL Multiphysics suitable for such replacements. For example, replace materials with high conductivity by the *perfect electric conductor (PEC)* boundary condition.
- Use boundary conditions for known solutions. For example, an antenna aperture can be modeled as an equivalent surface current density on a 2D face (boundary) in a 3D model.

Applying Electromagnetic Sources

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

Meshing and Solving

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

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

SOLVERS

In most cases the solver sequence generated by COMSOL Multiphysics can be used. The choice of solver is optimized for the typical case for each physics interface and study type in this module. However, in special cases tuning the solver settings may be required. This is especially important for 3D problems because they can require a large amount of memory. For large 3D problems, a 64-bit platform may be needed.

In the *COMSOL Multiphysics Reference Manual:*

- Meshing
- Studies and Solvers

Periodic Boundary Conditions

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

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

Periodic boundary conditions must have compatible meshes.

If more advanced periodic boundary conditions are required, for example, when there is a known rotation of the polarization from one boundary to another, see Model Couplings in the *COMSOL Multiphysics Reference Manual* for tools to define more general mappings between boundaries.

To learn how to use the Copy Mesh feature to ensure that the mesh on the destination boundary is identical to that on the source boundary, see Fresnel Equations: Model Library path **RF_Module/Verification_Models/fresnel_equations**.

In the *COMSOL Multiphysics Reference Manual*:

- Periodic Condition and Destination Selection
- Periodic Boundary Conditions

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 does not need a solution computed 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 the scattered field is solved for. Furthermore, a plane wave excitation is easier to set up, because for scattered-field problems it is specified as a global plane wave. Otherwise matched boundary conditions must be set up around the 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, the damping in the PML must be taken into account when exciting the plane wave, because the excitation appears outside the PML. With the scattered-field formulation the plane wave for all non-PML regions is specified, so it is not at all affected by the PML design.

SCATTERED FIELDS SETTING

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

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

Modeling with Far-Field Calculations

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

In this section:

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

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

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

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

For each variable name entered, the software generates functions and variables, which represent the vector components of the far electric field. The names of these variables are constructed by appending the names of the independent variables to the name entered in the field. For example, the name `Efar` is entered and the geometry is Cartesian with the independent variables `x`, `y`, and `z`, the generated variables get the

names `Efarx`, `Efary`, and `Efarz`. If, on the other hand, the geometry is axisymmetric with the independent variables `r`, `phi`, and `z`, the generated variables get the names `Efarr`, `Efarphi`, and `Efarz`. In 2D, the software only generates the variables for the nonzero field components. The physics interface name also appears in front of the variable names so they may vary, but typically look something like `emw.Efarz` and so forth.

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

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

The expression

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

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

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

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

Far-Field Calculations Theory

The Far Field Plots

The **Far Field** plots are available with this module to plot the value of a global variable (the far field norm, `normEfar` and `normdBEfar`, or components of the far field variable `Efar`). The variables are plotted for a selected number of angles on a unit circle (in 2D) or a unit sphere (in 3D). The angle interval and the number of angles can be manually specified. Also the circle origin and radius of the circle (2D) or sphere (3D) can be specified. For 3D **Far Field** plots you also specify an expression for the surface color.

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

Default **Far Field** plots are automatically added to any model that uses far field calculations.

- 2D model example with a Polar Plot Group—Radar Cross Section: Model Library path **RF_Module/Scattering_and_RCS/radar_cross_section**.
- 2D axisymmetric model example with a Polar Plot Group and a 3D Plot Group—Conical Antenna: Model Library path **RF_Module/Antennas/conical_antenna**.
- 3D model example with a Polar Plot Group and 3D Plot Group—Radome with Double-layered Dielectric Lens: Model Library path **RF_Module/Antennas/radome_antenna**.

- Far-Field Support in the Electromagnetic Waves, Frequency Domain User Interface
- Far Field in the *COMSOL Multiphysics Reference Manual*

S-Parameters and Ports

In this section:

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 waves at different ports of devices like filters, antennas, waveguide transitions, and transmission lines. S-parameters originate from transmission-line theory and are defined in terms of transmitted and reflected voltage waves. All ports are assumed to be connected to matched loads, that is, there is no reflection directly at a port.

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

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

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

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

For details on how COMSOL Multiphysics calculates the S-parameters, see S-Parameter Calculations.

S-Parameter Calculations: Ports

The RF interfaces have a built-in support for S-parameter calculations. To set up an S-parameter study use a *Port* boundary feature for each port in the model. Also use a *lumped port* that approximates connecting transmission lines. The lumped ports should only be used when the port width is much smaller than the wavelength.

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

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

S-Parameter Variables

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

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

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

Port Sweeps and Touchstone Export

The Port Sweep Settings section in the Electromagnetic Waves interface cycles through the ports, computes the entire S-matrix and exports it to a Touchstone file.

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

Lumped Ports with Voltage Input

In this section:

About Lumped Ports

The ports described in the S-Parameters and Ports section require a detailed specification of the mode, including the propagation constant and field profile. In situations when the mode is difficult to calculate or when there is an applied voltage to the port, a lumped port might be a better choice. This is also the appropriate choice when connecting a model to an electrical circuit. The lumped port is not as accurate as the ordinary port in terms of calculating S-parameters, but it is easier to use. For example, attach a lumped port as an internal port directly to a printed circuit board or to the transmission line feed of a device. The lumped port must be applied between two metallic objects separated by a distance much smaller than the wavelength, that is a local *quasi-static approximation* must be justified. This is because the concept of port or gap voltage breaks down unless the gap is much smaller than the local wavelength.

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

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

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

where V_{port} is the extracted voltage for the port given by the electric field line integral between the terminals averaged over the entire port. The current I_{port} is the averaged

total current over all cross sections parallel to the terminals. Ports not specified as input ports only return the extracted voltage and current.

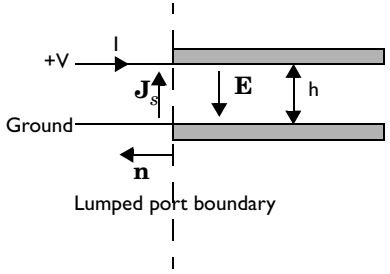
Lumped Port Parameters

Lumped Port Parameters

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

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

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 an equivalent surface impedance is defined at the lumped port boundary

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

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

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

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

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

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

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

Lumped Ports in the RF Module

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

LUMPED PORT VARIABLES

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

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

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

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

Lossy Eigenvalue Calculations

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

In this section:

- Eigenfrequency Analysis
- Mode Analysis

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

Eigenfrequency Analysis

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

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

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 AFFECTED BY EIGENFREQUENCY ANALYSIS

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

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

NONLINEAR EIGENFREQUENCY PROBLEMS

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

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

These situations require special treatment, especially since it can lead to "singular matrix" or "undefined value" messages if not treated correctly. The complication is not only the nonlinearity itself, it is also the way it enters the equations. For example the impedance boundary conditions with nonzero boundary conductivity has the term

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

where $(-\lambda) = -\delta + j\omega$. When the solver starts to solve the eigenfrequency problem it linearizes the entire formulation with respect to the eigenvalue around a certain linearization point. 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. Do this in the **Eigenvalue** node (not the Eigenfrequency node) under the **Solver Sequence** node in the **Study** branch of the Model Builder. A solver sequence may need to be generated first. In the **Linearization Point** section, select the **Transform point** check box and enter a suitable value in the

Point field. For example, it is known that the eigenfrequency is close to 1 GHz, enter the eigenvalue `1[GHz]` in the field.

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

1. Specify that the eigenvalue solver only search for one eigenvalue. Do this either for an existing solver sequence in the **Eigenvalue** node or, before generating a solver sequence, in the **Eigenfrequency** node.
2. Solve the problem with a "good" linearization point. As the 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.

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

Mode Analysis

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

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

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 INFLUENCED BY MODE ANALYSIS

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

NAME	EXPRESSION	CAN BE COMPLEX	DESCRIPTION
`beta`	`imag(-lambda)`	No	Propagation constant
`dampz`	`real(-lambda)`	No	Attenuation constant

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

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

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

Connecting to Electrical Circuits

In this section:

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

About Connecting Electrical Circuits to Physics User Interfaces

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

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

- External I vs. U
- External U vs. I
- External I-Terminal

These features either accept a voltage measurement from the connecting non-circuit physics interface and return a current from the circuit interface or the other way around.

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

Connecting Electrical Circuits Using Predefined Couplings

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

This coupling is typically activated when:

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

These circuit connections are supported in Lumped Ports.

Connecting Electrical Circuits by User-Defined Couplings

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

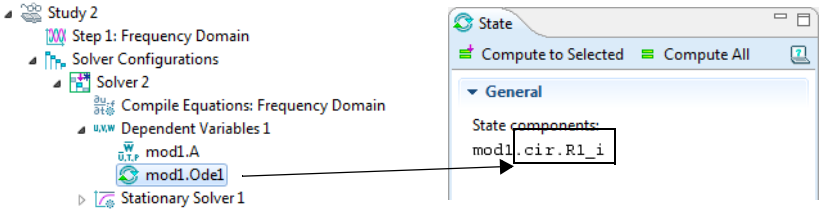
- Apply the voltage or current from the connecting "External" circuit feature as an excitation in the non-circuit physics interface.
- Define your own voltage or current measurement in the non-circuit physics interface using variables, coupling operators and so forth.
- In the settings window for the Electrical Circuit interface feature, selecting the User-defined option and entering the name of the variable or expression using coupling operators defined in the previous step.

DETERMINING A CURRENT OR VOLTAGE VARIABLE NAME

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

1 In the **Model Builder**, right-click the **Study** node and select **Show Default Solver**.

2 Expand the **Solver>Dependent Variables** node and click the state node, in this example, **mod1Ode1**. The variable name is shown on the **State** settings window.



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

3

Electromagnetics Theory

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

In this chapter:

- Maxwell's Equations
- Special Calculations
- Electromagnetic Quantities

See also:

- Theory for the Electromagnetic Waves User Interfaces
- Theory for the Electrical Circuit User Interface
- Heat Transfer Theory in the *COMSOL Multiphysics Reference Manual*

Maxwell's Equations

In this section:

Introduction to Maxwell's Equations

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

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

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

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

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

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

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

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

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

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, however, have a nonzero $\mathbf{M}$ also when there is no magnetic field present.

For linear materials, the polarization is directly proportional to the electric field, $\mathbf{P} = \varepsilon_0 \chi_e \mathbf{E}$, 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 magnetic Gauss' law. The electric potential results from Faraday's law.

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

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

Electromagnetic Energy

The electric and magnetic energies are defined as

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

$$w = w_{\mathrm{e}} + w_{\mathrm{m}} = \frac{1}{2}\varepsilon \mathbf{E} \cdot \mathbf{E} + \frac{1}{2\mu}\mathbf{B} \cdot \mathbf{B}$$

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.

MATERIAL PROPERTIES AND THE MATERIAL BROWSER

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

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

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

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

Materials and Modeling Anisotropic Materials in the *COMSOL Multiphysics Reference Manual*

Boundary and Interface Conditions

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

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

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

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

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

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

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

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

For example, all plot functions visualize

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

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

Special Calculations

In this section:

Lumped Ports with Voltage Input

S-Parameter Calculations

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

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

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

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

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

S-PARAMETERS IN TERMS OF POWER FLOW

For a guiding structure in single mode operation, it is also possible to interpret the S-parameters in terms of the power flow through the ports. Such a definition is only the absolute value of the S-parameters defined in the previous section and does not have any phase information.

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

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

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

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

POWER FLOW NORMALIZATION

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

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

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

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

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

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

TE Waves

For TE waves it holds that

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

where Z_{TE} is the wave impedance

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

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

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

TM Waves

For TM waves it holds that

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

where Z_{TM} is the wave impedance

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

and ε is the permittivity. The power flow then becomes

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

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

TEM Waves

For TEM waves it holds that

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

where Z_{TEM} is the wave impedance

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

The power flow then becomes

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

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

Far-Field Calculations Theory

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

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

and in 2D it looks slightly different:

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

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

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

In the above formulas,

- $\mathbf{E}$ and $\mathbf{H}$ are the fields on the “aperture”—the surface S enclosing the antenna.

- $\mathbf{r}_0$ is the unit vector pointing from the origin to the field point p. If the field points lie on a spherical surface S', $\mathbf{r}_0$ is the unit normal to S'.
- $\mathbf{n}$ is the unit normal to the surface S.
- η is the impedance:

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

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

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

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

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

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

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

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

References

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

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

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

4. R.K. Wangsness, *Electromagnetic Fields*, 2nd ed., John Wiley & Sons, 1986.

Electromagnetic Quantities

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

TABLE 3-1: ELECTROMAGNETIC QUANTITIES

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

TABLE 3-1: ELECTROMAGNETIC QUANTITIES

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

4

The Radio Frequency Branch

This chapter reviews the physics interfaces in the RF Module, which are under the **Radio Frequency** branch () in the **Model Wizard**.

In this chapter:

The Electromagnetic Waves, Frequency Domain User Interface

The **Electromagnetic Waves, Frequency Domain (emw)** interface (), found under the **Radio Frequency** branch () in the **Model Wizard**, solves the electric field based time-harmonic wave equation, which is strictly valid for linear media only.

The physics interface supports the study types Frequency domain, Eigenfrequency, Mode analysis, and Boundary mode analysis. The frequency domain study type is used for source driven simulations for a single frequency or a sequence of frequencies. The Eigenfrequency study type is used to find resonance frequencies and their associated eigenmodes in cavity problems.

When this interface is added, these default nodes are also added to the **Model Builder**—**Wave Equation, Electric**, **Perfect Electric Conductor**, and **Initial Values**.

Right-click the **Electromagnetic Waves, Frequency Domain** node to add other features that implement, for example, boundary conditions and sources. The following sections provide information about all feature nodes in the interface.

	The Mode analysis study type is applicable only for 2D cross-sections of waveguides and transmission lines where it is used to find allowed propagating modes.

	Boundary mode analysis is used for the same purpose in 3D and applies to boundaries representing waveguide ports.

INTERFACE IDENTIFIER

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

The default identifier (for the first interface in the model) is `emw`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the dependent variables and the equations. To choose specific domains, select **Manual** from the **Selection** list.

SETTINGS

From the **Solve for** list, select whether to solve for the **Full field** (the default) or the **Scattered field**. If **Scattered field** is selected, enter the component expressions for the **Background electric field** $\mathbf{E}_b$ (SI unit: V/m).

ELECTRIC FIELD COMPONENTS SOLVED FOR

This section is available for 2D models.

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

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

OUT-OF-PLANE WAVE NUMBER

This section is available for 2D models, when solving for **Three-component vector** or **In-plane vector**. Assign a wave vector component to the **Out-of-plane wave number** field.

This section is available for axisymmetric 2D models, when solving for **Three-component vector** or **In-plane vector**. Assign an integer constant or an integer parameter expression to the **Azimuthal mode number** field.

PORT SWEEP SETTINGS

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

If **Activate port sweep** is selected, enter a **Sweep parameter name** to assign a specific name to the variable that controls the port number solved for during the sweep.

For this interface, the lumped parameters are subject to **Touchstone file export**. Click **Browse** to locate the file, or enter a file name and path. Select an **Output format**—**Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

Enter a **Reference impedance**, **Touchstone file export** Z_{ref} (SI unit: Ω). The default is 50 Ω.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select **Linear**, **Quadratic** (the default), or **Cubic** for the **Electric field**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).

DEPENDENT VARIABLES

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

- Show More Physics Options
- Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Frequency Domain Interface
- Theory for the Electromagnetic Waves User Interfaces

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

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

The Electromagnetic Waves, Frequency Domain User Interface has these domain, boundary, edge, point, and pair nodes and subnodes available.

DOMAIN

- Archie's Law
- Divergence Constraint
- External Current Density
- Far-Field Calculation
- Far-Field Domain
- Initial Values
- Porous Media
- Wave Equation, Electric

BOUNDARY CONDITIONS

With no surface currents present the boundary conditions

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

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

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

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

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

EDGE, POINT, AND PAIR

- Circular Port Reference Axis
- Edge Current
- Electric Field
- Electric Point Dipole
- Line Current (Out-of-Plane)
- Magnetic Current
- Magnetic Point Dipole
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Port Reference Point
- Surface Current

For 2D axisymmetric models, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node to the model that is valid on the axial symmetry boundaries only.

In the *COMSOL Multiphysics Reference Manual*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Periodic Condition and Destination Selection
- Periodic Boundary Conditions

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

Wave Equation, Electric

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

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

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

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

where c_0 is the speed of light in vacuum.

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

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

The wave equation is thereby rewritten as

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

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

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

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

For this case, the wave equation is rewritten as

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

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

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

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

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

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

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

Also right-click the **Wave Equation, Electric** node to add a Divergence Constraint subnode.

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the settings include such model inputs. By default, this section is empty.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

ELECTRIC DISPLACEMENT FIELD

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

Relative Permittivity

When **Relative permittivity** is selected, the default **Relative permittivity** ε_r takes values **From material**. Select **Porous media** to add a Porous Media subnode, or if **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter values or expressions in the field or matrix.

Refractive Index

When **Refractive index** is selected, the default **Refractive index** n and **Refractive index, imaginary part** k take the values **From material**. To specify the real and imaginary parts of the refractive index and assume a relative permeability of unity and zero

conductivity, for one or both of the options, select **User defined** then choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic**. Enter values or expressions in the field or matrix.

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

Loss Tangent

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

Dielectric Loss

When **Dielectric loss** is selected, the default **Relative permittivity** ε' and **Relative permittivity (imaginary part)** ε'' take values **From material**. If **User defined** is selected for one or both options, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter values or expressions in the field or matrix.

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

Drude-Lorentz Dispersion Model

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

$$\varepsilon_r(\omega) = \varepsilon_\infty + \sum_{j=1}^{M} \frac{f_j \omega_P^2}{\omega_{0j}^2 - \omega^2 + i\Gamma_j \omega}$$

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

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

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

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

Debye Dispersion Model

The **Debye dispersion model** is given by

$$\varepsilon(\omega) = \varepsilon_\infty + \sum_k \frac{\Delta\varepsilon_k}{1 + i\omega\tau_k}$$

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

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

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

MAGNETIC FIELD

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

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

- If **Relative permeability** is selected, the **Relative permeability** μ_r uses values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix. If **Porous media** is selected, right-click to add a Porous Media subnode.
- If **Magnetic losses** is selected, the default values for **Relative permeability (real part)** μ' and **Relative permeability (imaginary part)** μ'' are taken **From material**. Select **User defined** to enter different values.

CONDUCTION CURRENT

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

- If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the current and enter values or expressions in the field or matrix.
- If **Linearized resistivity** is selected, the default values for the **Reference temperature** T_{ref} (SI unit: K), **Resistivity temperature coefficient** α (SI unit: 1/K), and **Reference resistivity** ρ_0 (SI unit: Ω·m) are taken **From material**. Select **User defined** to enter other values or expressions for any of these variables.
- When **Porous media** is selected, right-click to add a Porous Media subnode.
- When **Archie's law** is selected, right-click to add an Archie's Law subnode.

Divergence Constraint

Right-click the Wave Equation, Electric node to add a **Divergence Constraint** subnode. It is used for numerical stabilization when the frequency is low enough for the total electric current density related term in the wave equation to become numerically insignificant. For The Electromagnetic Waves, Frequency Domain User Interface and The Microwave Heating User Interface the divergence condition is given by

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

and for The Electromagnetic Waves, Transient User Interface it is

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

DOMAIN SELECTION

From the **Selection** list, choose the domains to define.

DIVERGENCE CONSTRAINT

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

For the Electromagnetic Waves, Frequency Domain and Microwave Heating interfaces, the SI unit is kg/(m·s^3·A)). The default is 1 kg/(m·s^3·A).

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

Initial Values

The **Initial Values** node adds an initial value for the electric field that can serve as an initial guess for a nonlinear solver. Right-click to add additional **Initial Values** node from the **Other** menu.

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

INITIAL VALUES

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

External Current Density

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

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

and in the equation that the interface defines.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

EXTERNAL CURRENT DENSITY

Based on space dimension, enter the components (**x**, **y**, and **z** for 3D models for example) of the **External current density** $\mathbf{J}_e$ (SI unit: A/m^2).

This subfeature is available only when **Archie's law** is selected as the **Electrical conductivity** material parameter in the parent feature (for example, the **Wave Equation, Electric** node). Then right-click the parent node to add this subnode.

Use the **Archie's Law** subnode to provide an electrical conductivity computed using Archie's Law. This subnode can be used to model nonconductive porous media saturated (or variably saturated) by conductive liquids, using the relation:

$$\sigma = s_L^n \varepsilon_p^m \sigma_L$$

Archie's Law Theory

DOMAIN SELECTION

From the **Selection** list, choose the domains to define.

MATERIAL TYPE

Select a **Material type**—**Solid**, **Non-solid**, or **From material**.

CONDUCTION CURRENTS

By default, the **Electrical conductivity** σ_L (SI unit: S/m) for the fluid is defined **From material**. This uses the value of the conductivity of the material domain.

If **User defined** is selected, enter a value or expression. If another type of temperature dependence is used other than a linear temperature relation, enter any expression for the conductivity as a function of temperature.

Enter these dimensionless parameters as required:

- **Cementation exponent** m
- **Saturation exponent** n
- **Fluid saturation** S_L
- **Porosity** ε_p to set up the volume fraction of the fluid.

This subfeature is available only when **Porous media** is selected as the material parameter (for example, **Relative permeability** or **Relative permittivity**) in the parent feature node on any Radio Frequency interface (for example, the **Wave Equation, Electric** node). Then right-click the parent node to add this subnode.

Use the **Porous Media** subfeature to specify the material properties of a domain consisting of a porous medium using a mixture model. The **Porous Media** subfeature is available for all the Radio Frequency branch physics interfaces and, depending on the specific physics interface, can be used to provide a mixture model for the electric conductivity σ, the relative dielectric permittivity ε_r, or the relative magnetic permeability μ_r.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define.

POROUS MEDIA

This section is always available and is used to define the mixture model for the domain.

Select the **Number of materials** (up to 5) to be included in the mixture model.

For each material (**Material 1**, **Material 2**, and so on), select either **Domain material**, to use the material specified for the domain, or one of the other materials specified in the **Materials** node. For each material, enter a **Volume fraction** θ_1, θ_2, and so on.

The Volume fractions specified for the materials should be fractional (between 0 and 1) and should add to 1 in normal cases.

The availability of the **Effective Electrical Conductivity**, **Effective Relative Permittivity**, and **Effective Relative Permeability** sections depend on the material properties used in the interface. In addition, these sections are only active if **Porous media** is selected in the corresponding material property for the parent feature node.

EFFECTIVE ELECTRICAL CONDUCTIVITY, EFFECTIVE RELATIVE PERMITTIVITY, OR EFFECTIVE RELATIVE PERMEABILITY

Select the averaging method to use in the mixture model between the **Volume average** of the material property (for example, **conductivity** or **permittivity**), the volume average of its inverse (for example, the **resistivity**), or the **Power law**. For each material, specify either **From material**, to take the value from the corresponding material specified in the **Porous Media** section, or **User defined** to manually input a value.

Effective Material Properties in Porous Media and Mixtures

Far-Field Domain

To set up a far-field calculation, add a **Far-Field Domain** node and specify the far-field domains in its settings window. Use Far-Field Calculation subnodes (one is added by default) to specify all other settings needed to define the far-field calculation. Select a homogeneous domain or domain group that is outside of all radiating and scattering objects and which has the material settings of the far-field medium.

DOMAIN SELECTION

From the **Selection** list, choose the domains to define The default setting is to include **All domains** in the model.

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

- Radar Cross Section: Model Library path **RF_Module/Scattering_and_RCS/radar_cross_section**
- Biconical Antenna: Model Library path **RF_Module/Antennas/biconical_antenna**

Far-Field Calculation

A **Far-Field Calculation** subnode is added by default to the Far-Field Domain node and is used to select boundaries corresponding to a single closed surface surrounding all radiating and scattering objects. Symmetry reduction of the geometry makes it relevant to select boundaries defining a non-closed surface. Also use this feature to indicate symmetry planes and symmetry cuts applied to the geometry, and whether the selected boundaries are defining the inside or outside of the far field domain; that is, to say whether facing away from infinity or toward infinity. Right-click the **Far-Field Domain** node to add additional subnodes as required.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define and that make up the source aperture for the far field.

FAR-FIELD CALCULATION

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

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

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

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

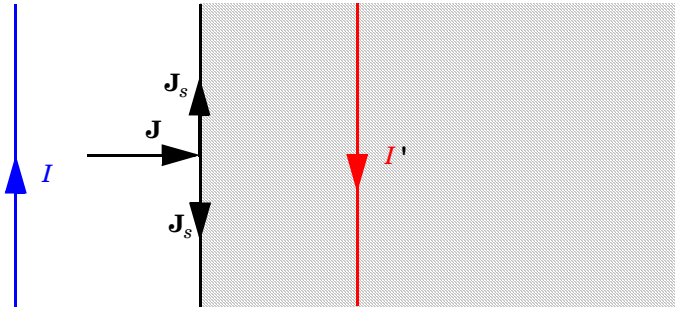
Dielectric Resonator Antenna: Model Library path
RF_Module/Antennas/dielectric_resonator_antenna

Perfect Electric Conductor

The **Perfect Electric Conductor** boundary condition

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

is a special case of the electric field boundary condition that sets the tangential component of the electric field to zero. It is used for the modeling of a lossless metallic surface, for example a ground plane or as a symmetry type boundary condition. It imposes symmetry for magnetic fields and "magnetic currents" and antisymmetry for electric fields and electric currents. It supports induced electric surface currents and thus any prescribed or induced electric currents (volume, surface or edge currents) flowing into a *perfect electric conductor* boundary is automatically balanced by induced surface currents.



The perfect electric conductor boundary condition is used on exterior and interior boundaries representing the surface of a lossless metallic conductor or (on exterior boundaries) representing a symmetry cut. The shaded (metallic) region is not part of the model but still carries effective mirror images of the sources. Note also that any current flowing into the boundary is perfectly balanced by induced surface currents. The tangential electric field vanishes at the boundary.

BOUNDARY OR EDGE SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific boundaries or edges, or select **All boundaries** or **All edges** as required.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. To **Apply reaction terms on** all dependent variables, select **All physics (symmetric)**.

Otherwise, select **Current physics (internally symmetric)** or **Individual dependent variables** to restrict the reaction terms as required. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation.

Show More Physics Options

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

Perfect Magnetic Conductor

The **Perfect Magnetic Conductor** boundary condition

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

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

in addition to setting the surface current density to zero also makes the tangential electric field discontinuous.

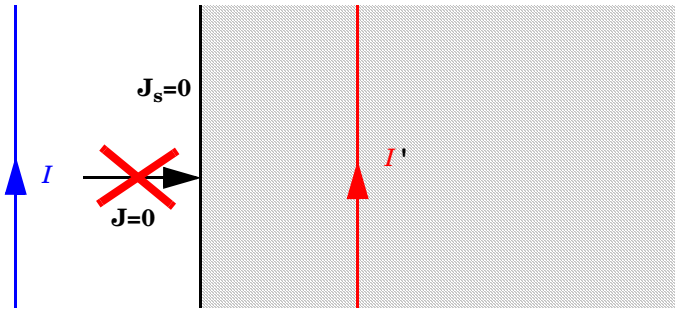


Figure 4-1: The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut. The shaded (high impedance) region is not part of the model but nevertheless carries effective mirror images of the sources. Note also that any electric current flowing into the boundary is forbidden as it cannot be balanced by induced electric surface currents. The tangential magnetic field vanishes at the boundary. On interior boundaries, the perfect magnetic conductor boundary condition literally sets the tangential magnetic field to zero which in addition to setting the surface current density to zero also makes the tangential electric field (and in dynamics the tangential electric field) discontinuous.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

Magnetic Frill: Model Library path **RF_Module/Antennas/magnetic_frill**

Port

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

In 3D, also right-click the **Port** node to select the:

- Circular Port Reference Axis to determine a reference direction for the modes. This subfeature is selected from the **Points** submenu when **Circular** is chosen as the type of port.
- Periodic Port Reference Point to uniquely determine reciprocal lattice vectors. This subfeature is selected from the **Points** submenu when **Periodic** is chosen as the type of port.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

PORT PROPERTIES

Enter a unique **Port name**. It is recommended to use a numeric name as it is used to define the elements of the S-parameter matrix and numeric port names are also required for port sweeps and Touchstone file export.

Select the **Type of Port**—**User defined**, **Numeric**, **Rectangular**, **Periodic**, **Coaxial**, or **Circular**.

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

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

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

Wave Excitation at this Port

To set whether it is an inport or a listener port, select **On** or **Off** from the **Wave excitation at this port** list. If **On** is selected, enter a **Port input power** P_{in} (SI unit: W), and **Port phase** θ_{in} (SI unit: rad).

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

PORT MODE SETTINGS

The input is based on the **Type of Port** selected above—User Defined, Rectangular, Circular, or Periodic. No entry is required if **Numeric** or **Coaxial** are selected. The **Port phase** field in the previous section has no impact for this mode type because the phase is determined by the entered fields.

User Defined

If **User defined** is selected, specify the eigenmode of the port.

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

Rectangular

If **Rectangular** is selected, specify a unique rectangular mode.

	In 3D, select a **Mode type**—**Transverse electric (TE)** or **Transverse magnetic (TM)**. Enter the **Mode number**, for example, 10 for a TE_{10} mode, or 11 for a TM_{11} mode.

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

Circular

If **Circular** is selected, specify a unique circular mode.

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

	In 3D, enter the **Mode number**, for example, 11 for a TE_{11} mode, or 01 for a TM_{01} mode. When **Circular** is chosen as the type of port in 3D, also right-click the **Port** node to add the Circular Port Reference Axis subfeature that defines the orientation of fields on a port boundary.

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

Periodic

If **Periodic** is selected, specify parameters for the incident wave and the periodic domain. When **Periodic** is chosen, also right-click the Port node to add a Diffraction Order port.

- Select a **Input quantity**—**Electric field** or **Magnetic field** and define the field amplitude.

For 2D models and if the **Input quantity** is set to **Electric field**, define the **Electric field amplitude**. For example, for a TE wave set the x, y, and z components to 0, 0, 1. Similarly, if the **Input quantity** is set to **Magnetic field**, define the **Magnetic field amplitude**. For a TM wave set the x, y, and z components to 0, 0, 1.

- Define the **Angle of incidence**.

In 3D, define the **Elevation angle of incidence** and **Azimuth angle of incidence**. The **Elevation angle of incidence** α_1 and **Azimuth angle of incidence** α_2 are used in the relations

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

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

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

In 2D, define the **Angle of incidence**. The **Angle of incidence** α is defined by the relation

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

where $\mathbf{k}$ is the projection of the wave vector in the xy-plane, $\mathbf{n}$ is the normalized normal vector to the boundary, k is the magnitude of the projected wave vector in the xy-plane, and $\mathbf{z}$ is the unit vector in the z-direction. Note that for a periodic structure with a top and a bottom side, the **Angle of incidence** for the two sides is of a different sign, since the normals point in opposite directions.

- Define the **Refractive index** at the boundary.
- Define the **Maximum frequency**. If a single frequency is used, insert the frequency, or if a frequency sweep is performed, insert the maximum frequency of the sweep.
- When all parameters are defined, click the **Compute Diffraction Orders** button to automatically create **Diffraction Order** ports as subnodes to the **Periodic** port.

- S-Parameters and Ports
- S-Parameter Variables

- Waveguide Adapter: Model Library path **RF_Module/Transmission_Lines_and_Waveguides/waveguide_adapter**
- Three-Port Ferrite Circulator: Model Library path **RF_Module/Ferrimagnetic_Devices/circulator**
- Plasmonic Wire Grating: Model Library path: **RF_Module/Tutorial_Models/plasmonic_wire_grating**

Circular Port Reference Axis

The **Circular Port Reference Axis** is available only in 3D. When the **Type of port** is set to **Circular** under Port Properties, right-click the Port node to add the **Circular Port Reference Axis** subfeature. Two points are used to define the orientation of fields on a port boundary. If there are more than two points on the selection list, the first and last points are used.

For the fundamental TE_{11} mode, the direction of the reference axis corresponds to the polarization of the electric field at the port center.

POINT SELECTION

From the **Selection** list, choose the points to define.

Diffraction Order

The **Diffraction Order** port is available in 3D and 2D. When the **Type of Port** is set to **Periodic** under Port Properties, right-click the Port node to add this feature.

Use the **Diffraction Order** port to define diffraction orders from a periodic structure. Normally a **Diffraction Order** node is added automatically during the **Periodic** port setup. You can also right-click the Port node to add additional **Diffraction Order** ports.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

PORT PROPERTIES

Enter a unique **Port name**. It is recommended to use a numeric name as it is used to define the elements of the S-parameter matrix and numeric port names are also required for port sweeps and Touchstone file export.

The Diffraction Order port is a listener port feature. Enter a value or expression for the **Port phase** θ_{in} (SI unit: rad). The default is 0 radians.

PORT MODE SETTINGS

These settings define the diffracted plane wave.

Components

Select the **Components** setting for the port—**In-plane vector** (the default) or **Out-of-plane vector**.

In-plane vector is available when the settings for the physics interface is set to either **In-plane vector** or **Three-component vector** under Electric Field Components Solved For.

Out-of-plane vector is available when the settings for the physics interface is set to either **Out-of-plane vector** or **Three-component vector** under Electric Field Components Solved For.

Diffraction Order

Specify an integer constant or and integer parameter expression for the **Diffraction order** setting.

- S-Parameters and Ports
- S-Parameter Variables

Plasmonic Wire Grating: Model Library path
RF_Module/Tutorial_Models/plasmonic_wire_grating

Periodic Port Reference Point

The **Periodic Port Reference Point** is available only in 3D. When the **Type of Port** is set to **Periodic** under Port Properties, right-click the Port node to add the **Periodic Port Reference Point** subfeature.

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

$$\mathbf{a}_1 \times \mathbf{a}_2 = \mathbf{n}$$

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

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

POINT SELECTION

From the **Selection** list, choose the points to define.

Lumped Port

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

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

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

PORT PROPERTIES

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

Type of Port

Select a **Type of Port**—**Uniform**, **Coaxial**, or **User defined**.

Select **User defined** for non uniform ports, for example, a curved port and enter values or expressions in the fields—**Height of lumped port** h_{port} (SI unit: m), **Width of lumped port** w_{port} (SI unit: m), and **Direction between lumped port terminals** $\mathbf{a}_{\text{h}}$.

Terminal Type

Select a **Terminal type**—a **Cable** port for a voltage driven transmission line, a **Current** driven port, or a **Circuit** port.

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

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

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

SETTINGS

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

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

- S-Parameters and Ports
- Lumped Ports with Voltage Input

Balanced Patch Antenna for 6 GHz: Model Library path **RF_Module/Antennas/patch_antenna**

Lumped Element

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

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

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

LUMPED ELEMENT PROPERTIES

Enter a unique **Lumped element name**. See Lumped Port for the rest of the settings.

SETTINGS

Select a **Lumped element type**—**User defined** (the default), **Inductor**, or **Capacitor**.

- If **User defined** is selected, enter a **Lumped element impedance** Z_{element} (SI unit: Ω) The default is 50 Ω.

- If **Inductor** is selected, enter a **Lumped element inductance** L_{element} (SI unit: H) The default is 1 nH.
- If **Capacitor** is selected, enter a **Lumped element capacitance** C_{element} (SI unit: F) The default is 1 pF.

Inductor and **capacitor** are available only in the frequency domain study type.

SMA Connectorized Wilkinson Power Divider: Model Library path **RF_Module/Passive_Devices/wilkinson_power_divider**

Electric Field

The **Electric Field** boundary condition

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

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

BOUNDARY OR EDGE SELECTION

From the **Selection** list, choose the geometric entity (boundaries or edges) to define.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

ELECTRIC FIELD

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

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. To **Apply reaction terms on** all dependent variables, select **All physics (symmetric)**. Otherwise, select **Current physics (internally symmetric)** or **Individual dependent variables** to restrict the reaction terms as required. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation.

Show More Physics Options

Magnetic Field

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

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

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

MAGNETIC FIELD

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

Scattering Boundary Condition

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

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

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

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

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

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

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

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

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

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

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

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

SCATTERING BOUNDARY CONDITION

Select a **Wave type** for which the boundary is absorbing—**Spherical wave**, **Cylindrical wave**, or **Plane wave**.

For all **Wave types**, enter coordinates for the **Wave direction** $\mathbf{k}_{\text{dir}}$ (dimensionless).

- If **Cylindrical wave** is selected, also enter coordinates for the **Source point** $\mathbf{r}_o$ (SI unit: m) and **Source axis direction** $\mathbf{r}_{\text{axis}}$ (dimensionless).
- If **Spherical wave** is selected, enter coordinates for the **Source point** $\mathbf{r}_o$ (SI unit: m).

Select an **Incident field**—**Wave given by E field** or **Wave given by H field**. Enter the expressions for the components for the **Incident electric field** $\mathbf{E}_0$ or **Incident magnetic field** $\mathbf{H}_0$. This setting is not available in 2D axisymmetry.

Conical Antenna: Model Library path
RF_Module/Antennas/conical_antenna

Impedance Boundary Condition

The **Impedance Boundary Condition**

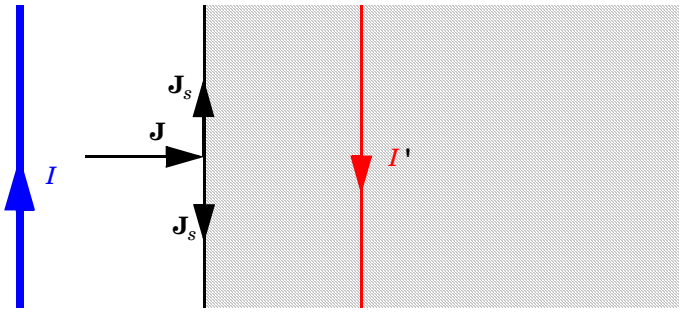
$$\sqrt{\frac{\mu_0\mu_r}{\varepsilon_c}}\mathbf{n}\times\mathbf{H}+\mathbf{E}-(\mathbf{n}\cdot\mathbf{E})\mathbf{n}\;=\;(\mathbf{n}\cdot\mathbf{E}_s)\mathbf{n}-\mathbf{E}_s$$

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 refractive index

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

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

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



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

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

IMPEDANCE BOUNDARY CONDITION

The following default material properties for the domain outside the boundary, which this boundary condition approximates, are all taken **From material**:

- **Relative permeability** μ_r (dimensionless)
- **Relative permittivity** ε_r (dimensionless)
- **Electrical conductivity** σ (SI unit: S/m)

Select **User defined** for any of these to enter a different value or expression.

Enter the values or expressions for the components of a **Source electric field** $\mathbf{E}_s$ (SI unit: V/m).

For **Relative permittivity** ε_r, you can alternatively select **Porous media** and then right-click to add a Porous Media subnode.

- Coaxial to Waveguide Coupling: Model Library path **RF_Module/Transmission_Lines_and_Waveguides/coaxial_waveguide_coupling**
- Computing Q-Factors and Resonant Frequencies of Cavity Resonators: Model Library path **RF_Module/Verification_Models/cavity_resonators**

Surface Current

The **Surface Current** boundary condition

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

specifies a surface current density at both exterior and interior boundaries. The current density is specified as a three-dimensional vector, but because it needs to flow along the boundary surface, COMSOL Multiphysics projects it onto the boundary surface and neglects its normal component. This makes it easier to specify the current density and avoids unexpected results when a current density with a component normal to the surface is given.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to define. An identity pair has to be created first. Ctrl-click to deselect.

SURFACE CURRENT

Enter values or expressions for the components of the **Surface current density** $\mathbf{J}_{s0}$ (SI unit: A/m).

Transition Boundary Condition

The **Transition Boundary Condition** is used on interior boundaries to model a sheet of a medium that should be geometrically thin but does not have to be electrically thin. It represents a discontinuity in the tangential electric field. Mathematically it is described

by a relation between the electric field discontinuity and the induced surface current density:

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

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

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

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

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

Where indices 1 and 2 refer to the different sides of the layer. This feature is not available with the Electromagnetic Waves, Transient interface.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

TRANSITION BOUNDARY CONDITION

The following default material properties for the thin layer which this boundary condition approximates, are all taken **From material**:

- **Relative permeability** μ_r (dimensionless)
- **Relative permittivity** ε_r (dimensionless)
- **Electrical conductivity** σ (SI unit: S/m).

For **Relative permittivity** ε_r, you can alternatively select **Porous media** and then right-click to add a Porous Media subnode.

Select **User defined** for any of these to enter a different value or expression.Enter a **Thickness** d (SI unit: m). The default is 0.01 m.

Periodic Condition

The **Periodic Condition** sets up a periodicity between the selected boundaries. Right-click to add a **Destination Selection** node as required.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define. The software automatically identifies the boundaries as either source boundaries or destination boundaries.

This works fine for cases like opposing parallel boundaries. To control the destination, right-click to add a **Destination Selection** node. By default it contains the selection that COMSOL Multiphysics has identified.

PERIODICITY SETTINGS

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

- **Continuity** to make the electric field periodic (equal on the source and destination),
- **Antiperiodicity** to make it antiperiodic, or
- **Floquet periodicity** (The Electromagnetic Waves, Frequency Domain User Interface only) to use a Floquet periodicity (Bloch-Floquet periodicity).
  - If **Floquet periodicity** is selected, also enter the source for the **k-vector for Floquet periodicity**.
  - If **User defined** is selected, specify the components of the **k-vector for Floquet periodicity** $\mathbf{k}_F$ (SI unit: rad/m).
  - If **From periodic port** is selected the **k-vector for Floquet periodicity** $\mathbf{k}_F$ is obtained from the **Periodic Port** settings.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. To **Apply reaction terms on** all dependent variables, select **All physics (symmetric)**. Otherwise, select **Current physics (internally symmetric)** or **Individual dependent variables** to restrict the reaction terms as required. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation.

- Periodic Boundary Conditions
- Show More Physics Options

In the *COMSOL Multiphysics Reference Manual*:

- Periodic Condition and Destination Selection
- Periodic Boundary Conditions

- Fresnel Equations: Model Library path **RF_Module/Verification_Models/fresnel_equations**
- Plasmonic Wire Grating: Model Library path: **RF_Module/Tutorial_Models/plasmonic_wire_grating**

Magnetic Current

For 2D and 2D axisymmetric models the **Magnetic Current** node is applied to **Points**, representing magnetic currents directed out of the model plane.

For 3D models, the **Magnetic Current** is applied to **Edges**.

The **Magnetic Current** node specifies a magnetic line current along one or more edges. For a single **Magnetic Current** source, the electric field is orthogonal to both the line and the distance vector from the line to the field point.

EDGE OR POINT SELECTION

From the **Selection** list, choose the edges or points to define.

MAGNETIC CURRENT

Enter a value for the **Magnetic current** I_{m} (SI unit: V).

Edge Current

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

EDGE SELECTION

From the **Selection** list, choose the edges to define.

EDGE CURRENT

Enter an **Edge current** I_0 (SI unit: A).

Electric Point Dipole

Add **Electric Point Dipole** nodes to 3D and 2D models. This represents the limiting case of when the length d of a current filament carrying uniform current I approaches zero while maintaining the product between I and d. The dipole moment is a vector entity with the positive direction set by the current flow.

POINT SELECTION

From the **Selection** list, choose the points to define.

DIPOLE SPECIFICATION

Select a **Dipole specification**—**Magnitude and direction** or **Dipole moment**.

DIPOLE PARAMETERS

Based on the **Dipole specification** selection:

- If **Magnitude and direction** is selected, enter coordinates for the **Electric current dipole moment direction** $\mathbf{n}_p$ and **Electric current dipole moment, magnitude** p (SI unit: mA).
- If **Dipole moment** is selected, enter coordinates for the **Electric current dipole moment** $\mathbf{p}$ (SI unit: mA).

Magnetic Point Dipole

Add a **Magnetic Point Dipole** to 3D and 2D models. The point dipole source represents a small circular current loop I in the limit of zero loop area a at a fixed product $I*a$.

POINT SELECTION

From the **Selection** list, choose the points to define.

DIPOLE SPECIFICATION

Select a **Dipole specification**—**Magnitude and direction** or **Dipole moment**.

DIPOLE PARAMETERS

Based on the **Dipole specification** selection:

- If **Magnitude and direction** is selected, enter coordinates for the **Magnetic dipole moment direction** $\mathbf{n}_m$ and **Magnetic dipole moment, magnitude** m (SI unit: $\mathrm{m}^2\mathrm{A}$).
- If **Dipole moment** is selected, enter coordinates for the **Magnetic dipole moment** $\mathbf{m}$ (SI unit: $\mathrm{m}^2\mathrm{A}$).

Line Current (Out-of-Plane)

Add a **Line Current (Out-of-Plane)** node to 2D or 2D axisymmetric models. This specifies a line current out of the modeling plane. In axially symmetric geometries this is the rotational direction, in 2D geometries it is the z direction.

POINT SELECTION

From the **Selection** list, choose the points to define.

LINE CURRENT (OUT-OF-PLANE)

Enter an **Out-of-plane current** I_0 (SI unit: A).

The Electromagnetic Waves, Transient User Interface

The **Electromagnetic Waves, Transient (temw)** user interface (), found under the **Radio Frequency** branch () in the **Model Wizard**, solves a transient wave equation for the magnetic vector potential.

When this interface is added, these default nodes are also added to the **Model Builder**—**Wave Equation, Electric**, **Perfect Electric Conductor**, and **Initial Values**.

Right-click the **Electromagnetic Waves, Transient** node to add other features that implement, for example, boundary conditions and mass sources.

Except where indicated, most of the settings are the same as for The Electromagnetic Waves, Frequency Domain User Interface.

INTERFACE IDENTIFIER

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

The default identifier (for the first interface in the model) is `temw`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the dependent variables and the equations. To choose specific domains, select **Manual** from the **Selection** list.

COMPONENTS


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

Select the **Electric field components solved for**. Select:

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

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select **Quadratic** (the default), **Linear**, or **Cubic** for the **Magnetic vector potential**. Specify the **Value type when using splitting of complex variables**—**Real** (the default) or **Complex**.

DEPENDENT VARIABLES

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

- Show More Physics Options
- Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Transient User Interface
- Theory for the Electromagnetic Waves User Interfaces

Transient Modeling of a Coaxial Cable: Model Library path **RF_Module/Verification_Models/coaxial_cable_transient**

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

The Electromagnetic Waves, Transient User Interface shares most of its nodes with The Electromagnetic Waves, Frequency Domain User Interface. The domain, boundary, edge, point, and pair nodes are available as indicated.

DOMAIN

These nodes are unique for this interface and described in this section:

- Wave Equation, Electric
- Initial Values

BOUNDARY CONDITIONS

With no surface currents present the boundary conditions

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

need to be fulfilled. Depending on the field being solved for, it is necessary to analyze these conditions differently. When solving for $\mathbf{A}$, the first condition can be formulated in the following way.

$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = \mathbf{n}_2 \times \left(\frac{\partial \mathbf{A}_2}{\partial t} - \frac{\partial \mathbf{A}_1}{\partial t} \right) = \frac{\partial}{\partial t}(\mathbf{n}_2 \times (\mathbf{A}_2 - \mathbf{A}_1))$$

The tangential component of the magnetic vector potential is always continuous and thus the first condition is fulfilled. The second condition is equivalent to the natural boundary condition.

$$-\mathbf{n} \times (\mu_r^{-1} \nabla \times \mathbf{A}_1 - \mu_r^{-1} \nabla \times \mathbf{A}_2) = -\mathbf{n} \times \mu_r^{-1}(\mathbf{H}_1 - \mathbf{H}_2) = \mathbf{0}$$

and is therefore also fulfilled.

These nodes and subnodes are available and described for the Electromagnetic Waves, Frequency Domain interface (listed in alphabetical order):

- Archie's Law
- Lumped Port
- Magnetic Field
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Condition
- Porous Media
- Scattering Boundary Condition
- Surface Current

EDGE, POINT, AND PAIR

These edge, point, and pair nodes are available and described for the Electromagnetic Waves, Frequency Domain interface (listed in alphabetical order):

- Edge Current
- Electric Point Dipole (2D and 3D models)
- Line Current (Out-of-Plane) (2D and 2D axisymmetric models)
- Lumped Port
- Magnetic Point Dipole (2D and 3D models)
- Perfect Electric Conductor
- Perfect Electric Conductor
- Surface Current

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

In the *COMSOL Multiphysics Reference Manual*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs

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

Wave Equation, Electric

The **Wave Equation, Electric** node is the main node for the Electromagnetic Waves, Transient interface. The governing equation can be written in the form

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

for transient problems with the constitutive relations $\mathbf{B} = \mu_0\mu_r\mathbf{H}$ and $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E}$. Other constitutive relations can also be handled for transient problems. Also right-click the **Wave Equation, Electric** node to add a **Divergence Constraint** subnode.

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

MODEL INPUTS

This section contains field variables that appear as model inputs, if the settings include such model inputs. By default, this section is empty.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

ELECTRIC DISPLACEMENT FIELD

Select an **Electric displacement field model**—**Relative permittivity** (the default), **Refractive index**, **Polarization**, or **Remanent electric displacement**.

Relative Permittivity

When **Relative permittivity** is selected, the default **Relative permittivity** ε_r (dimensionless) takes values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter values or expressions in the field or matrix. If **Porous media** is selected, right-click to add a **Porous Media** subnode.

Refractive Index

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

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

Polarization

If **Polarization** is selected enter coordinates for the **Polarization** $\mathbf{P}$ (SI unit: $\mathrm{C/m^2}$).

Remanent Electric Displacement

If **Remanent electric displacement** is selected, enter coordinates for the **Remanent electric displacement** $\mathbf{D}_\mathrm{r}$ (SI unit: $\mathrm{C/m^2}$). Then select **User defined** or **From Material** as above for the **Relative permittivity** ε_r.

MAGNETIC FIELD

This section is available if **Relative permittivity**, **Polarization**, or **Remanent electric displacement** are chosen as the **Electric displacement field model**.

Select the **Constitutive relation**—**Relative permeability** (the default), **Remanent flux density**, or **Magnetization**.

Relative Permeability

If **Relative permeability** is selected, the **Relative permeability** μ_r uses values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix. If **Porous media** is selected, right-click to add a **Porous Media** subnode.

Remanent Flux Density

If **Remanent flux density** is selected, the **Relative permeability** μ_r uses values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix. Then enter coordinates for the **Remanent flux density** $\mathbf{B}_\mathrm{r}$ (SI unit: T).

Magnetization

If **Magnetization** is selected, enter coordinates for $\mathbf{M}$ (SI unit: A/m).

CONDUCTION CURRENT

This section is available if **Relative permittivity**, **Polarization**, or **Remanent electric displacement** are chosen as the **Electric displacement field model**.

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

- If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the current and enter values or expressions in the field or matrix.
- If **Linearized resistivity** is selected, the default values for the **Reference temperature** T_{ref} (SI unit: K), **Resistivity temperature coefficient** α (SI unit: 1/K), and **Reference resistivity** ρ_0 (SI unit: Ωm) use values **From material**. Select **User defined** to enter other values or expressions for any of these variables.
- If **Porous media** is selected, right-click to add a **Porous Media** subnode.
- If **Archie's Law** is selected, right-click to add an **Archie's Law** subnode.

Initial Values

The **Initial Values** node adds an initial value for the magnetic vector potential and its time derivative that serves as initial conditions for the transient simulation.

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

INITIAL VALUES

Enter values or expressions for the initial values of the components of the magnetic vector potential $\mathbf{A}$ (SI unit: Wb/m) and its time derivative $\partial\mathbf{A}/\partial t$ (SI unit: V/m). The default values are 0 Wb/m and 0 V/m, respectively.

The Transmission Line User Interface

The **Transmission Line (tl)** interface (), found under the **Radio Frequency** branch () in the **Model Wizard**, solves the time-harmonic transmission line equation for the electric potential. The interface is used when solving for electromagnetic wave propagation along one-dimensional transmission lines and is available in 1D, 2D and 3D.

The physics interface has Eigenfrequency and Frequency Domain study types available. The frequency domain study is used for source driven simulations for a single frequency or a sequence of frequencies.

When this interface is added, these default nodes are also added to the **Model Builder**—**Transmission Line Equation**, **Absorbing Boundary**, and **Initial Values**.Right-click the **Transmission Line** node to add other features that implement boundary conditions.

INTERFACE IDENTIFIER

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

The default identifier (for the first interface in the model) is `tl`.

DOMAIN, EDGE, OR BOUNDARY SELECTION

The default setting is to include **All edges** (3D models), **All boundaries** (2D models), or **All domains** (1D models) to define the dependent variables and the equations. To choose specific geometric entities, select **Manual** from the **Selection** list.

Select **Edges** for 3D models, **Boundaries** for 2D models, and **Domains** for 1D models. **Points** are available for all space dimensions (3D, 2D, and 1D).

PORT SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

Select the **Activate port sweep** check box to switch on the port sweep. When selected, this invokes a parametric sweep over the ports/terminals in addition to the automatically generated frequency sweep. The generated lumped parameters are in the form of an impedance or admittance matrix depending on the port/terminal settings which consistently must be of either fixed voltage or fixed current type.If **Activate port sweep** is selected, enter a **Sweep parameter name** (the default is `PortName`) to assign a specific name to the variable that controls the port number solved for during the sweep.

For this interface, the lumped parameters are subject to **Touchstone file export**. Click **Browse** to locate the file, or enter a file name and path. Select an **Output format**—**Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

DEPENDENT VARIABLES

The dependent variable (field variable) is the **Electric potential V** (SI unit: V). The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select **Linear**, **Quadratic** (the default), **Cubic**, **Quartic**, or **Quintic** for the **Electric potential**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).

- Show More Physics Options
- Domain, Boundary, Edge, Point, and Pair Nodes for the Transmission Line Equation User Interface
- Theory for the Transmission Line User Interface
- Selecting and Visualizing in Models in the *COMSOL Multiphysics Reference Manual*

Quarter-Wave Transformer: Model Library path **RF_Module/Transmission_Lines_and_Waveguides/quarter_wave_transformer**

Domain, Boundary, Edge, Point, and Pair Nodes for the Transmission Line Equation User Interface

The Transmission Line User Interface has these domain, boundary, edge, point, and pair nodes available and listed in alphabetical order:

Select **Edges** for 3D models, **Boundaries** for 2D models, and **Domains** for 1D models. **Points** are available for all space dimensions (3D, 2D, and 1D).

For all space dimensions, select **Points** for the boundary condition.

- Absorbing Boundary
- Incoming Wave
- Initial Values
- Open Circuit
- Terminating Impedance
- Transmission Line Equation
- Short Circuit
- Lumped Port

- Theory for the Transmission Line Boundary Conditions

In the *COMSOL Multiphysics Reference Manual*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs

Transmission Line Equation

The **Transmission Line Equation** node is the main feature of the Transmission Line interface. It defines the 1D wave equation for the electric potential. The wave equation is written in the form

$$\frac{\partial}{\partial x}\left(\frac{1}{R+i\omega L}\frac{\partial V}{\partial x}\right)-(G+i\omega C)V = 0$$

where R, L, G, and C are the distributed resistance, inductance, conductance, and capacitance, respectively.

DOMAIN, EDGE, OR BOUNDARY SELECTION

The default setting is to include **All edges** (3D models), **All boundaries** (2D models), or **All domains** (1D models) in the model. This cannot be edited.

TRANSMISSION LINE EQUATION

Enter the values for the following:

- **Distributed resistance** R (SI unit: m·kg/(s^3·A^2)). The default is 0.
- **Distributed inductance** L (SI unit: H/m). The default is 2.5e-6 H/m.
- **Distributed conductance** G (SI unit: S/m). The default is 0.
- **Distributed capacitance** C (SI unit: F/m). The default is 1e-9 F/m.

The default values give a characteristic impedance for the transmission line of 50 Ω.

Initial Values

The **Initial Values** node adds an initial value for the electric potential that can serve as an initial guess for a nonlinear solver.

DOMAIN, EDGE, OR BOUNDARY SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains (1D models), edges (3D models), or boundaries (2D models). Or select **All domains**, **All edges** or, **All boundaries** as required.

INITIAL VALUES

Enter values or expressions for the initial values of the **Electric potential** V (SI unit: V). The default is 0 V.

Absorbing Boundary

The **Absorbing Boundary** condition is stated as

$$\frac{\mathbf{n} \cdot \nabla V}{R + j\omega L} + \frac{V}{Z_0} = 0$$

where γ is the complex propagation constant defined by

$$\gamma = \sqrt{(R + i\omega L)(G + i\omega C)}$$

and **n** is the normal pointing out of the domain. The *absorbing boundary* condition prescribes that propagating waves are absorbed at the boundary and, thus, that there is no reflection at the boundary.

The **Absorbing Boundary** condition is only available on external boundaries.

BOUNDARY OR POINT SELECTION

The default setting is to include **All points** (3D and 2D models) or **All boundaries** (1D models) in the model. This cannot be edited.

Theory for the Transmission Line Boundary Conditions

Incoming Wave

The **Incoming Wave** boundary condition

$$\frac{\mathbf{n}\cdot\nabla V}{R+j\omega L}+\frac{V-2V_0}{Z_0}=0$$

lets a wave of complex amplitude V_{in} enter the domain. The complex propagation constant γ and the outwards-pointing normal **n** are defined in the section describing the Absorbing Boundary node.

The **Incoming Wave** boundary condition is only available on external boundaries.

BOUNDARY OR POINT SELECTION

From the **Selection** list, choose the points or boundaries to define.

For 2D and 3D models, select **Points** for the boundary condition. For 1D models, select **Boundaries**.

VOLTAGE

Enter the value or expression for the input **Electric potential** V_0 (SI unit: V). The default is 1 V.

Theory for the Transmission Line Boundary Conditions

Open Circuit

The **Open Circuit** boundary condition is a special case of the Terminating Impedance boundary condition, assuming an infinite impedance, and, thus, zero current at the boundary. The condition is thus

$$\mathbf{n} \cdot \nabla V = 0$$

The **Open Circuit** boundary condition is only available on external boundaries.

BOUNDARY OR POINT SELECTION

From the **Selection** list, choose the points or boundaries to define.

For 2D and 3D models, select **Points** for the boundary condition. For 1D models, select **Boundaries**.

Theory for the Transmission Line Boundary Conditions

Terminating Impedance

The **Terminating Impedance** boundary condition

$$\frac{\mathbf{n} \cdot \nabla V}{R + j\omega L} + \frac{V}{Z_L} = 0$$

specifies the terminating impedance to be Z_L. Notice that the Absorbing Boundary condition is a special case of this boundary condition for the case when

$$Z_L = Z_0 = \sqrt{\frac{R + j\omega L}{G + j\omega C}}$$

The Open Circuit and Short Circuit boundary conditions are also special cases of this condition. The **Terminating Impedance** boundary condition is only available on external boundaries.

BOUNDARY OR POINT SELECTION

From the **Selection** list, choose the points or boundaries to define.

For 2D and 3D models, select **Points** for the boundary condition. For 1D models, select **Boundaries**.

IMPEDANCE

Enter the value or expression for the **Impedance** Z_L (SI unit: Ω). The default is 50 Ω.

Theory for the Transmission Line Boundary Conditions

Short Circuit

The **Short Circuit** node is a special case of the Terminating Impedance boundary condition, assuming that impedance is zero and, thus, the electric potential is zero. The constraint at this boundary is, thus, $V = 0$.

BOUNDARY OR POINT SELECTION

From the **Selection** list, choose the points or boundaries to define.

For 2D and 3D models, select **Points** for the boundary condition. For 1D models, select **Boundaries**.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. To **Apply reaction terms on** all dependent variables, select **All physics (symmetric)**.

Otherwise, select **Current physics (internally symmetric)** or **Individual dependent variables** to restrict the reaction terms as required. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation.

- Theory for the Transmission Line Boundary Conditions
- Show More Physics Options

Lumped Port

Use the **Lumped Port** node to apply a voltage or current excitation of a model or to connect to a circuit. The **Lumped Port** node also defines S-parameters (reflection and transmission coefficients) that can be used in later post-processing steps.

BOUNDARY OR POINT SELECTION

From the **Selection** list, choose the points or boundaries to define.

For 2D and 3D models, select **Points** for the boundary condition. For 1D models, select **Boundaries**.

PORT PROPERTIES

Enter a unique **Port Name**. It is recommended to use a numeric name as it is used to define the elements of the S-parameter matrix and numeric port names are also required for port sweeps and Touchstone file export.

Select a **Type of Port**—**Cable** (the default), **Current**, or **Circuit**.

SETTINGS

If a **Circuit** port type is selected under **Port Properties**, this section does not require any selection.

- If a **Cable** port type is selected under **Port Properties**, enter the **Characteristic impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.
- If a **Current** terminal type is selected under **Port Properties**, enter a **Terminal current** I_0 (SI unit: A). The default is 1 A.

If **Cable** is selected as the port type, select the **Wave excitation at this port** check box to enter values or expressions for the:

- **Electric potential** V_0 (SI unit: V). The default is 1 V.
- **Port phase** Θ_{in} (SI unit: rad). The default is 0.

- S-Parameters and Ports
- Lumped Ports with Voltage Input
- Theory for the Transmission Line Boundary Conditions

The Electromagnetic Waves, Time Explicit User Interface

The **Electromagnetic Waves, Time Explicit (ewte)** user interface (), found under the **Radio Frequency** branch () in the **Model Wizard**, solves a transient wave equation for both the electric and magnetic fields. It is available for 3D, 2D axisymmetric, and 2D models.

When this interface is added, these default nodes are also added to the **Model Builder**—**Wave Equations**, **Perfect Electric Conductor**, and **Initial Values**.

Right-click the **Electromagnetic Waves, Time Explicit** node to add other features that implement other boundary conditions.

INTERFACE IDENTIFIER

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

The default identifier (for the first interface in the model) is `ewte`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the dependent variables and the equations. To choose specific domains, select **Manual** from the **Selection** list.

COMPONENTS


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

Select the **Field components solved for**:

- **Full wave** (the default) to solve using a full three-component vector for the electric field $\mathbf{E}$ and the magnetic field $\mathbf{H}$.
- **E in plane (TM wave)** to solve for the electric field vector components in the modeling plane and one magnetic field vector component perpendicular to the plane, assuming that there is no electric field perpendicular to the plane and no magnetic field components in the plane.
- **H in plane (TE wave)** to solve for the magnetic field vector components in the modeling plane and one electric field vector component perpendicular to the plane.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. Select **Cubic** (the default), **Linear**, **Quadratic** or **Quartic** for the vector field components. Specify the **Value type when using splitting of complex variables**—**Real** (the default) or **Complex**.

DEPENDENT VARIABLES

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

- Domain, Boundary, and Pair Nodes for the Electromagnetic Waves, Time Explicit User Interface
- Show More Physics Options
- Theory for the Electromagnetic Waves, Time Explicit User Interface

Domain, Boundary, and Pair Nodes for the Electromagnetic Waves, Time Explicit User Interface

The Electromagnetic Waves, Time Explicit User Interface has these domain and boundary nodes available and listed in alphabetical order.

- Electric Field
- Electric Current Density
- Flux/Source
- Initial Values
- Low Reflecting Boundary
- Magnetic Field
- Magnetic Current Density
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Surface Current Density
- Wave Equations

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

Wave Equations

The **Wave Equations** node is the main node for the Electromagnetic Waves, Time Explicit interface. The governing transient equations can be written in the form

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

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

with the constitutive relations $\mathbf{B} = \mu_0 \mu_r \mathbf{H}$ and $\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E}$, which reads

$$\varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} + \sigma \mathbf{E} = 0$$

$$\mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \times \mathbf{E} = 0$$

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the

interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

MATERIAL PROPERTIES

The default **Relative permittivity** ε_r (dimensionless), **Relative permeability** μ_r (dimensionless), and **Electrical conductivity** σ (SI unit: S/m) take values **From material**. If **User defined** is selected for any of the properties, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** and enter values or expressions in the field or matrix.

NUMERICAL PARAMETERS

The defaults for each parameter are as follows:

- **Lax-Friedrichs flux parameter for E field** τ_E (SI unit: S), the default is $0.5/Z$ for Ampere's law.
- **Lax-Friedrichs flux parameter for H field** τ_H (SI unit: Ω), the default is $0.5Z$ for Faraday's law, where Z is the impedance of vacuum.
- **Estimate of maximum wave speed** $c_{\max}$ (SI unit: m/s) the default is taken from the speed of light in a vacuum `c_const`.

FILTER PARAMETERS

The filter provides higher-order smoothing of nodal discontinuous Galerkin formulations and is intended to be used for absorbing layers, but you can also use it to stabilize linear wave problems with highly varying coefficients. The filter is constructed by transforming the solution (in each global time step) to an orthogonal polynomial representation, multiplying with a damping factor and then transforming back to the (Lagrange) nodal basis. Select the **Activate** check box to use this filter.

The exponential filter can be described by the matrix formula

$$V\Lambda V^{-1}$$

where V is a Vandermonde matrix induced by the node points, and Λ is a diagonal matrix with the exponential damping factors on the diagonal:

$$\Lambda_{mm} = \sigma(\eta) = \begin{cases} 1, 0 \le \eta \le \eta_c \\ e^{-\alpha\left(\frac{\eta - \eta_c}{1 - \eta_c}\right)^{2s}}, \eta_c \le \eta \le 1 \end{cases}$$

where

$$\eta = \eta(m) = \frac{i_m}{N_p}$$

and N_p is the basis function and i_m the polynomial order for coefficient m. α (default value: 36), η_c (default value: 1), and s (default value: 3) are the filter parameters that you specify in the corresponding text fields. The damping is derived from an a spatial dissipation operator of order $2s$. For $s = 1$, you obtain a damping that is related to the classical 2nd-order Laplacian. Higher order (larger s) gives less damping for the lower-order polynomial coefficients (a more pronounced low-pass filter), while keeping the damping property for the highest values of η, which is controlled by α. The default values 36 for α correspond to maximal damping for $\eta = 1$. It is important to realize that the effect of the filter is influenced by how much of the solution (energy) is represented by the higher-order polynomial coefficients. For a well resolved solution this will be a smaller part than for a poorly resolved solution. The effect will be stronger for poorly resolved solutions than for well resolved ones. This is one of the reasons why this filter is useful in an absorbing layer where the energy is transferred to the higher-order coefficients through a coordinate transformation. See Ref. 1 (Chapter 5) for more information.

α must be positive; $\alpha = 0$ means no dissipation, and the maximum value is related to the machine precision, $-\log(\varepsilon)$, which is approximately 36. η_c should be between 0 and 1, where $\eta_c = 0$ means maximum filtering, and $\eta_c = 1$ means no filtering, even if filtering is active.

Reference

1. Jan S. Hesthaven and Tim Warburton, *Nodal Discontinuous Galerkin Methods—Algorithms, Analysis, and Applications*, Springer, 2008.

Initial Values

The **Initial Values** node adds the initial values for the **Electric field** and **Magnetic field** variables that serve as an initial condition for the transient simulation.

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

If there is more than one type of domain, each with different initial values defined, it may be necessary to remove these domains from the selection. These are then defined in an additional **Initial Values** node.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

INITIAL VALUES

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

Electric Current Density

The **Electric Current Density** node adds an external current density to the specified domains, which appears on the right-hand side of Ampere's law

$$\varepsilon_0\varepsilon_r\frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} + \sigma\mathbf{E} = -\mathbf{J}_e$$

DOMAIN SELECTION

From the **Selection** list, choose the domains to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

ELECTRIC CURRENT DENSITY

Based on space dimension, enter the coordinates (**x**, **y**, and **z** for 3D models for example) of the **Electric current density** $\mathbf{J}_e$ (SI unit: A/m^2).

Magnetic Current Density

The **Magnetic Current Density** node adds an external current density to the specified domains, which appears on the right-hand side of Faraday's law

$$\mu_0\mu_r\frac{\partial \mathbf{H}}{\partial t} + \nabla \times \mathbf{E} = -\mathbf{J}_m$$

DOMAIN SELECTION

From the **Selection** list, choose the domains to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

MAGNETIC CURRENT DENSITY

Based on space dimension, enter the coordinates (**x**, **y**, and **z** for 3D models for example) of the **Magnetic current density** $\mathbf{J}_m$ (SI unit: V/m^2).

Electric Field

The **Electric Field** boundary condition

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

specifies the tangential component of the electric field. The commonly used special case of zero tangential electric field (perfect electric conductor) is described in the next section.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

ELECTRIC FIELD

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

Perfect Electric Conductor

The **Perfect Electric Conductor** boundary condition

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

is a special case of the electric field boundary condition that sets the tangential component of the electric field to zero. It is used for the modeling of a lossless metallic surface, for example a ground plane or as a symmetry type boundary condition.

It imposes symmetry for magnetic fields and antisymmetry for electric fields and electric currents. It supports induced electric surface currents and thus any prescribed or induced electric currents (volume, surface or edge currents) flowing into a perfect electric conductor boundary is automatically balanced by induced surface currents.

BOUNDARY SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific boundaries or select **All boundaries** as required.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

Magnetic Field

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

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

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

MAGNETIC FIELD

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

Perfect Magnetic Conductor

The **Perfect Magnetic Conductor** boundary condition

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

is a special case of the surface current boundary condition that sets the tangential component of the magnetic field and thus also the surface current density to zero. On external boundaries, this can be interpreted as a "high surface impedance" boundary condition or used as a symmetry type boundary condition. It imposes symmetry for electric fields and electric currents. Electric currents (volume, surface, or edge currents) are not allowed to flow into a perfect magnetic conductor boundary as that would violate current conservation. On interior boundaries, the perfect magnetic conductor boundary condition literally sets the tangential magnetic field to zero which in addition to setting the surface current density to zero also makes the tangential electric field discontinuous.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

Surface Current Density

The **Surface Current Density** boundary condition

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

specifies a surface current density at both exterior and interior boundaries. The current density is specified as a three-dimensional vector, but because it needs to flow along the boundary surface, COMSOL Multiphysics projects it onto the boundary surface and neglects its normal component. This makes it easier to specify the current density and avoids unexpected results when a current density with a component normal to the surface is given.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

SURFACE CURRENT

Enter values or expressions for the components of the **Surface current** $\mathbf{J}_{s0}$ (SI unit: A/m). The defaults are 0 A/m for all vector components.

Low Reflecting Boundary

The **Low-Reflecting Boundary** condition

$$\mathbf{n} \times \mathbf{E} = Z_0 \mathbf{H}$$

specifies the tangential component of both electric and magnetic fields.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

IMPEDANCE

Enter the value or expression for the medium **Impedance** Z_0 (SI unit: Ω). By default, the Z_0 uses the value of the vacuum's impedance. Then choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the material characteristics and enter values or expressions in the field or matrix.

Flux/Source

The **Flux/Source** boundary condition

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

specifies the tangential component of both electric and magnetic fields.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries to define.

MATERIAL TYPE

Select a **Material type**—**Non-solid** (the default), **Solid**, or **From material**.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

BOUNDARY FLUX/SOURCE

Enter values or expressions for the components of the tangential **Electric field** $\mathbf{E}_0$ (SI unit: V/m) and the tangential **Magnetic field** $\mathbf{H}_0$ (SI unit: A/m).

Theory for the Electromagnetic Waves User Interfaces

The Electromagnetic Waves, Frequency Domain User Interface and The Electromagnetic Waves, Transient User Interface theory is described in this section:

- Introduction to the User Interface Equations
- Frequency Domain Equation
- Time Domain Equation
- Vector Elements
- Eigenfrequency Calculations
- Effective Material Properties in Porous Media and Mixtures
- Effective Conductivity in Porous Media and Mixtures
- Effective Relative Permittivity in Porous Media and Mixtures
- Effective Relative Permeability in Porous Media and Mixtures
- Archie's Law Theory
- Reference for Archie's Law

Introduction to the User Interface Equations

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

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

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

Frequency Domain Equation

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

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

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

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

The first of these, based on the electric field is used in The Electromagnetic Waves, Frequency Domain User Interface.

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

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

The wave number in vacuum k_0 is defined by

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

where c_0 is the speed of light in vacuum.

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

EIGENFREQUENCY ANALYSIS

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

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

MODE ANALYSIS AND BOUNDARY MODE ANALYSIS

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

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

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. When solving for all three electric field components the allowed anisotropy of the optionally complex relative permittivity and relative permeability is limited to:

$$\varepsilon_{rc} = \begin{bmatrix} \varepsilon_{rxx} & \varepsilon_{rxy} & 0 \\ \varepsilon_{ryx} & \varepsilon_{ryy} & 0 \\ 0 & 0 & \varepsilon_{rzz} \end{bmatrix} \qquad \mu_r = \begin{bmatrix} \mu_{rxx} & \mu_{rxy} & 0 \\ \mu_{ryx} & \mu_{ryy} & 0 \\ 0 & 0 & \mu_{rzz} \end{bmatrix}$$

Limiting the electric field component solved for to the out-of-plane component for TE modes, requires that the medium is homogeneous, that is, μ and ε are constant. When solving for the in-plane electric field components for TM modes, μ may vary but ε must be constant. It is strongly recommended to use the most general approach, that is solving for all three components which is sometimes referred to as "perpendicular hybrid-mode waves".

Variables Influenced by Mode Analysis

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

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

In 2D, different polarizations can be chosen by selecting to solve for a subset of the 3D vector components. When selecting all three components, the 3D equation applies with the addition that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field.

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

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

The wave equation is thereby rewritten as

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

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

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

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

and the wave equation is expressed as

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

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

In-plane Hybrid-Mode Waves

Solving for all three components in 2D is referred to as “hybrid-mode waves”. The equation is formally the same as in 3D with the addition that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field

In-plane TM Waves

The TM waves polarization has only one magnetic field component in the z direction, and the electric field lies in the modeling plane. Thus the time-harmonic fields can be obtained by solving for the in-plane electric field components only. The equation is formally the same as in 3D, the only difference being that the out-of-plane electric field

component is zero everywhere and that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field.

In-plane TE Waves

As the field propagates in the modeling xy-plane a TE wave has only one non zero electric field component, namely in the z direction. The magnetic field lies in the modeling plane. Thus the time-harmonic fields can be simplified to a scalar equation for $\boldsymbol{E}_z$,

$$-\nabla \cdot (\tilde{\mu}_r \nabla \boldsymbol{E}_z) - \varepsilon_{rzz} k_0^2 \boldsymbol{E}_z = 0$$

where

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

To be able to write the fields in this form, it is also required that ε_r, σ, and μ_r are non diagonal only in the xy-plane. μ_r denotes a 2-by-2 tensor, and ε_{rzz} and σ_{zz} are the relative permittivity and conductivity in the z direction.

Axisymmetric Hybrid-Mode Waves

Solving for all three components in 2D is referred to as "hybrid-mode waves". The equation is formally the same as in 3D with the addition that spatial derivatives with respect to φ are evaluated for the prescribed azimuthal mode number dependence of the electric field.

Axisymmetric TM Waves

A TM wave has a magnetic field with only a φ component and thus an electric field with components in the rz-plane only. The equation is formally the same as in 3D, the only difference being that the φ component is zero everywhere and that spatial derivatives with respect to φ are evaluated for the prescribed azimuthal mode number dependence of the electric field.

Axisymmetric TE Waves

A TE wave has only an electric field component in the φ direction, and the magnetic field lies in the modeling plane. Given these constraints, the 3D equation can be simplified to a scalar equation for $\boldsymbol{E}_\varphi$. To write the fields in this form, it is also required that ε_r and μ_r are non diagonal 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.

INTRODUCING LOSSES IN THE FREQUENCY DOMAIN

Electric Losses

The frequency domain equations allow for several ways of introducing electric losses. Finite conductivity results in a *complex permittivity*,

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

The conductivity gives rise to ohmic losses in the medium.

A more general approach is to use a complex permittivity,

$$\varepsilon_c = \varepsilon_0(\varepsilon' - j\varepsilon'')$$

where ε' is the real part of ε_r, and all losses are given by ε''. This dielectric loss model can be combined with a finite conductivity resulting in:

$$\varepsilon_c = \varepsilon_0\left(\varepsilon' - j\left(\frac{\sigma}{\omega\varepsilon_0} + \varepsilon''\right)\right)$$

The complex permittivity may also be introduced as a loss tangent:

$$\varepsilon_c = \varepsilon_0\varepsilon'(1 - j\tan\delta)$$

When specifying losses through a loss tangent, conductivity is not allowed as an input.

In optics and photonics applications, the refractive index is often used instead of the permittivity. In materials where μ_r is 1, the relation between the complex refractive index

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

and the complex relative permittivity is

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

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. When specifying the refractive index, conductivity is not allowed as an input.

In the physics and optics literature, the time harmonic form is often written with a minus sign (and "i" instead of "j"):

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

This makes an important difference in how loss is represented by complex material coefficients like permittivity and refractive index, that is, by having a positive imaginary part rather than a negative one. Therefore, material data taken from the literature may have to be conjugated before using in a model.

Magnetic Losses

The frequency domain equations allow for magnetic losses to be introduced as a *complex relative permeability*.

$$\mu_r = (\mu' - j\mu'')$$

The complex relative permeability may be combined with any electric loss model except refractive index.

Time Domain Equation

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

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

This is the equation used by The Electromagnetic Waves, Transient User Interface. It is suitable for the simulation of non-sinusoidal waveforms or non linear media.

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

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

WAVES IN 2D

In 2D, different polarizations can be chosen by selecting to solve for a subset of the 3D vector components. When selecting all three components, the 3D equation applies with the addition that out-of-plane spatial derivatives are set to zero.

In-plane Hybrid-Mode Waves

Solving for all three components in 2D is referred to as "hybrid-mode waves". The equation form is formally the same as in 3D with the addition that out-of-plane spatial derivatives are set to zero.

In-plane TM Waves

The TM waves polarization has only one magnetic field component in the z direction, and thus the electric field and vector potential lie in the modeling plane. Hence it is obtained by solving only for the in-plane vector potential components. The equation is formally the same as in 3D, the only difference being that the out-of-plane vector potential component is zero everywhere and that out-of-plane spatial derivatives are set to zero.

In-plane TE Waves

As the field propagates in the modeling xy-plane a TE wave has only one non zero vector potential component, namely in the z direction. The magnetic field lies in the modeling plane. Thus the equation in the time domain can be simplified to a scalar equation for A_z:

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

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

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

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

Axisymmetric Hybrid-Mode Waves

Solving for all three components in 2D is referred to as "hybrid-mode waves". The equation form is formally the same as in 3D with the addition that spatial derivatives with respect to φ are set to zero.

Axisymmetric TM Waves

TM waves have a magnetic field with only a φ component and thus an electric field and a magnetic vector potential with components in the *rz*-plane only. The equation is formally the same as in 3D, the only difference being that the φ component is zero everywhere and that spatial derivatives with respect to φ are set to zero.

Axisymmetric TE Waves

A TE wave has only a vector potential component in the φ direction, and the magnetic field lies in the modeling plane. Given these constraints, the 3D equation can be simplified to a scalar equation for A_φ. To write the fields in this form, it is also required that ε_r and μ_r are non diagonal 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.

Vector Elements

Whenever solving for more than a single vector component, it is not possible to use Lagrange elements for electromagnetic wave modeling. The reason is that they force the fields to be continuous everywhere. This implies that the interface conditions, which specify that the normal components of the electric and magnetic fields are discontinuous across interior boundaries between media with different permittivity and permeability, cannot be fulfilled. To overcome this problem, the Electromagnetic Waves, Frequency Domain physics interface uses *vector elements*, which do not have this limitation.

The solution obtained when using vector elements also better fulfills the divergence conditions $\nabla\cdot\mathbf{D} = 0$ and $\nabla\cdot\mathbf{B} = 0$ than when using Lagrange elements.

Eigenfrequency Calculations

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

- Nonlinear eigenvalue problems appear for impedance boundary conditions with nonzero conductivity and for *scattering boundary conditions* adjacent to domains with nonzero conductivity. Such problems have to be treated specially.
- Some of the boundary conditions, such as the surface current condition and the electric field condition, can specify a source in the eigenvalue problem. These conditions are available as a general tool to specify arbitrary expressions between the **H** field and the **E** field. Avoid specifying solution-independent sources for these conditions because the eigenvalue solver ignores them anyway.

Using the default parameters for the eigenfrequency study, it might find a large number of false eigenfrequencies, which are almost zero. This is a known consequence of using vector elements. To avoid these eigenfrequencies, change the parameters for the eigenvalue solver in the **Study settings**. Adjust the settings so that the solver searches for eigenfrequencies closer to the lowest eigenfrequency than to zero.

Effective Material Properties in Porous Media and Mixtures

One way of dealing with porous media or mixtures of solids in electromagnetic models is to replace them with an homogenized medium. The electric and magnetic properties of this medium are computed from the properties of each phase by means of an averaging formula.

There are several possible approaches to compute an average material property starting from the material properties and the volume fraction of each material.

The following sections illustrate the different formulae available to compute the *Effective electrical conductivity*, the *Effective relative permittivity* and the *Effective relative permeability* of a homogenized medium. In the following, volume fractions of the materials are indicated with θ_i, where i is the material index, and they are assumed to be fractional (between 0 and 1). Up to five different materials can be specified as phases of the mixture. Typically, their volume fractions should add up to 1.

Effective Conductivity in Porous Media and Mixtures

Three methods are available to compute the averaged electrical conductivity of the mixture.

VOLUME AVERAGE, CONDUCTIVITY

If the electric conductivities of the two materials are not so different from each other, a simple form of averaging can be used, such as a volume average:

$$\sigma = \sum_{i=1}^{n} \theta_i \sigma_i = \theta_1 \sigma_1 + \theta_2 \sigma_2 + \ldots$$

where σ_i is the conductivity of the material i. This is equivalent to a "parallel" system of resistivities.

If the conductivities are defined by second order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RESISTIVITY

A similar expression for the effective conductivity can be used, which mimics a "series" connection of resistivities. Equivalently, the effective conductivity is obtained from

$$\frac{1}{\sigma} = \sum_{i=0}^{n} \frac{\theta_i}{\sigma_i} = \frac{\theta_1}{\sigma_1} + \frac{\theta_2}{\sigma_2} + \ldots$$

If the conductivities are defined by second order tensors, the inverse of the tensors are used.

POWER LAW

A power law gives the following expression for the equivalent conductivity:

$$\sigma = \prod_{i=0}^{n} \sigma_i^{\theta_i} = \sigma_1^{\theta_1}\sigma_2^{\theta_2}\ldots$$

The effective conductivity calculated by Volume Average, Conductivity is the upper bound, the effective conductivity calculated by Volume Average, Resistivity is the lower bound, and the Power Law average is somewhere in between these two.

Effective Relative Permittivity in Porous Media and Mixtures

Three methods are available to compute the averaged electrical conductivity of the mixture.

VOLUME AVERAGE, PERMITTIVITY

If the relative permittivity of the two materials is not so different from each other, the effective relative permittivity ε_r is calculated by simple volume average:

$$\varepsilon = \sum_{i=1}^{n} \theta_i \varepsilon_i = \theta_1\varepsilon_1 + \theta_2\varepsilon_2 + \ldots$$

where ε_i is the relative permeability of the material i.

If the permittivity is defined by second order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RECIPROCAL PERMITTIVITY

The second method is the volume average of the inverse of the permittivities:

$$\frac{1}{\varepsilon} = \sum_{i=0}^{n} \frac{\theta_i}{\varepsilon_i} = \frac{\theta_1}{\varepsilon_1} + \frac{\theta_2}{\varepsilon_2} + \dots$$

If the permittivity is defined by a second order tensor, the inverse of the tensor is used.

POWER LAW

A power law gives the following expression for the equivalent permittivity:

$$\varepsilon = \prod_{i=0}^{n} \varepsilon_i^{\theta_i} = \varepsilon_1^{\theta_1} \varepsilon_2^{\theta_2} \dots$$

The effective permeability calculated by Volume Average, Permittivity is the upper bound, the effective permeability calculated by Volume Average, Reciprocal Permittivity is the lower bound, and the Power Law average gives a value somewhere in between these two.

Effective Relative Permeability in Porous Media and Mixtures

Three methods are available to compute the averaged electrical conductivity of the mixture.

VOLUME AVERAGE, PERMEABILITY

If the relative permeability of the two materials is not so different from each other, the effective relative permeability μ_r is calculated by simple volume average:

$$\mu = \sum_{i=1}^{n} \theta_i \mu_i = \theta_1 \mu_1 + \theta_2 \mu_2 + \dots$$

where μ_i is the relative permeability of the material i.

If the permeability is defined by second order tensors (such as for anisotropic materials), the volume average is applied element by element.

VOLUME AVERAGE, RECIPROCAL PERMEABILITY

The second method is the volume average of the inverse of the permeabilities:

$$\frac{1}{\mu} = \sum_{i=0}^{n} \frac{\theta_i}{\mu_i} = \frac{\theta_1}{\mu_1} + \frac{\theta_2}{\mu_2} + \dots$$

If the permeability is defined by a second order tensor, the inverse of the tensor is used.

POWER LAW

A power law gives the following expression for the equivalent permeability:

$$\mu = \prod_{i=0}^{n} \mu_i^{\theta_i} = \mu_1^{\theta_1} \mu_2^{\theta_2} \dots$$

The effective permeability calculated by Volume Average, Permeability is the upper bound, the effective permeability calculated by Volume Average, Reciprocal Permeability is the lower bound, and the Power Law average gives a value somewhere in between these two.

Archie's Law Theory

The electrical conductivity of the materials composing saturated rocks and soils can vary over many orders of magnitude. For instance, in the petroleum reservoirs, normal sea water (or brine) has a typical conductivity of around 3 S/m, whereas hydrocarbons are typically much more resistive and have conductivities in the range 0.1 – 0.01 S/m.

The porous rocks and sediments may have even lower conductivities. In variably saturated soils, the conductivity of air is roughly ten orders of magnitude lower that the ground water. A simple volume average (of either conductivity or resistivity) in rocks or soils might give different results compared to experimental data.

Since most crustal rocks, sedimentary rocks, and soils are formed by non-conducting materials, Archie (Ref. 2) assumed that electric current are mainly caused by ion fluxes trough the pore network. Originally, Archie's law is an empirical law for the effective conductivity of a fully-saturated rock or soil, but it can be extended to variably saturated porous media.

Archie's law relates the effective conductivity to the fluid conductivity σ_L, fluid saturation s_L and porosity ε_p:

$$\sigma = s_L^n \varepsilon_p^m \sigma_L$$

here, m is the cementation exponent, a parameters that describes the connectivity of the pores. The cementation exponent normally varies between 1.3 and 2.5 for most sedimentary rocks, and it is close to 2 for sandstones. The lower limit $m = 1$ represents a volume average of the conductivities of a fully saturated, insulating (zero conductivity) porous matrix, and a conducting fluid. The saturation coefficient n is normally close to 2.

The ratio $F = \sigma_L/\sigma$ is called the *formation factor*.

Archie's Law does not take care of the relative permittivity of either fluids or solids, so the effective relative permittivity of the porous medium is normally consider as $\varepsilon_r = 1$.

Reference for Archie's Law

2. G.E. Archie, "The Electric Resistivity as an Aid in Determining Some Reservoir Characteristics," *Trans. Am. Inst. Metal. Eng.* 146, 54–62, 1942.

Theory for the Transmission Line User Interface

The Transmission Line User Interface theory is described in this section.

- Introduction to Transmission Line Theory
- Theory for the Transmission Line Boundary Conditions

Introduction to Transmission Line Theory

Figure 4-2 is an illustration of a transmission line of length L. The distributed resistance R, inductance L, conductance G, and capacitance C, characterize the properties of the transmission line.

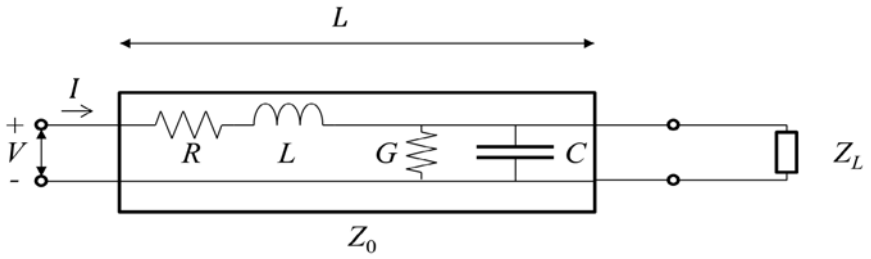


Figure 4-2: Schematic of a transmission line with a load impedance.

The distribution of the electric potential V and the current I describes the propagation of the signal wave along the line. The following equations relate the current and the electric potential

$$\frac{\partial V}{\partial x} = -(R + j\omega L)I \tag{4-1}$$

$$\frac{\partial I}{\partial x} = -(G + j\omega C)V \tag{4-2}$$

Equation 4-1 and Equation 4-2 can be combined to the second-order partial differential equation

$$\frac{\partial^2 V}{\partial x^2} = \gamma^2 V \tag{4-3}$$

where

$$\gamma = \sqrt{(R + j\omega L)(G + j\omega C)} = \alpha + j\beta$$

Here γ, α, and β are called the complex propagation constant, the attenuation constant, and the (real) propagation constant, respectively.

The attenuation constant, α, is zero if R and G are zero.

The solution to Equation 4-3 represents a forward- and a backward-propagating wave

$$V(x) = V_+ e^{-\gamma x} + V_- e^{\gamma x} \tag{4-4}$$

By inserting Equation 4-4 in Equation 4-1 you get the current distribution

$$I(x) = \frac{\gamma}{R + j\omega L}(V_+ e^{-\gamma x} - V_- e^{\gamma x})$$

If only a forward-propagating wave is present in the transmission line (no reflections), dividing the voltage by the current gives the characteristic impedance of the transmission line

$$Z_0 = \frac{V}{I} = \frac{R + j\omega L}{\gamma} = \sqrt{\frac{R + j\omega L}{G + j\omega C}}$$

To make sure that the current is conserved across internal boundaries, COMSOL Multiphysics solves the following wave equation (instead of Equation 4-3)

$$\frac{\partial}{\partial x}\left(\frac{1}{R + j\omega L}\frac{\partial V}{\partial x}\right) - (G + j\omega C)V = 0 \tag{4-5}$$

Theory for the Transmission Line Boundary Conditions

The Transmission Line User Interface has these boundary conditions:

$$V_1 = V_2 \tag{4-6}$$

and

$$I_1 = I_2 \tag{4-7}$$

In Equation 4-6 and Equation 4-7, the indices 1 and 2 denote the domains on the two sides of the boundary. The currents flowing out of a boundary are given by

$$I_i = -\frac{\mathbf{n}_i \cdot \nabla V_i}{R_i + j\omega L_i},\ i = 1, 2$$

where $\mathbf{n}_i$ are the normals pointing out of the domain.

Because V is solved for, the electric potential is always continuous, and thus Equation 4-6 is automatically fulfilled. Equation 4-7 is equivalent to the natural boundary condition

$$\frac{1}{R_2 + j\omega L_2}\frac{\partial V}{\partial x}\bigg|_2 - \frac{1}{R_1 + j\omega L_1}\frac{\partial V}{\partial x}\bigg|_1 = 0$$

which is fulfilled with the wave-equation formulation in Equation 4-5.

When the transmission line is terminated by a load impedance, as Figure 4-2 shows, the current through the load impedance is given by

$$I(L) = \frac{V(L)}{Z_L} \tag{4-8}$$

Inserting Equation 4-1into Equation 4-8, results in the Terminating Impedance boundary condition

$$\frac{1}{R + j\omega L}\frac{\partial V}{\partial x} + \frac{V}{Z_L} = 0 \tag{4-9}$$

If the arbitrary load impedance Z_L is replaced by the characteristic impedance of the transmission line Z_0 you get the Absorbing Boundary condition. By inserting the voltage, defined in Equation 4-4, in Equation 4-9 you can verify that the boundary condition doesn't allow any reflected wave (that is, V is zero).

The Open Circuit boundary condition is obtain by letting the load impedance become infinitely large, that is, no current flows through the load impedance.

On the other hand, the Short Circuit boundary condition specifies that the voltage at the load is zero. In COMSOL Multiphysics this is implemented as a constraint on the electric potential.

To excite the transmission line, you use the Incoming Wave boundary condition. Referring to the left (input) end of the transmission line in Figure 4-2, the forward propagating wave has a voltage amplitude of V_0. Thus, the total voltage at this boundary is given by

$$V(0) = V = V_0 + V_-$$

Thereby, the current can be written as

$$I(0) = -\frac{1}{R + j\omega L}\frac{\partial V}{\partial x}\bigg|_{x=0} = \frac{1}{Z_0}(V_0 - V_-) = \frac{2V_0 - V}{Z_0}$$

resulting in the boundary condition

$$-\frac{1}{R + j\omega L}\frac{\partial V}{\partial x} + \frac{V - 2V_0}{Z_0} = 0$$

For the Lumped Port boundary condition, the port current (positive when entering the transmission line) defines the boundary condition as

$$-\frac{1}{R + j\omega L}\frac{\partial V}{\partial x} - I_{\text{port}} = 0$$

where the port current I_{port} is given by

$$I_{\text{port}} = \frac{2V_0 - V}{Z_0}$$

for a **Cable** lumped port (see the Lumped Port section for a description of the lumped port settings).

For a **Current**-controlled lumped port, you provide I_{port} as an input parameter, whereas it is part of an electrical circuit equation for a **Circuit**-based lumped port.

Theory for the Electromagnetic Waves, Time Explicit User Interface

The Electromagnetic Waves, Time Explicit User Interface theory is described in this section:

- The Equations
- In-plane E Field or In-plane H Field
- Fluxes as Dirichlet Boundary Conditions

The Equations

Maxwell's equations are a set of equations, written in differential or integral form, stating the relationships between the fundamental electromagnetic quantities. These quantities are the:

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

For general time-varying fields, the differential form of Maxwell's equations can be written as

$$\begin{aligned} \nabla \times \mathbf{H} &= \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \\ \nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t} \\ \nabla \cdot \mathbf{D} &= \rho \\ \nabla \cdot \mathbf{B} &= 0 \end{aligned} \tag{4-10}$$

The first two equations are also called Maxwell-Ampere's law and Faraday's law, respectively. Equation three and four are two forms of Gauss' law, the electric and magnetic form, respectively.

CONSTITUTIVE RELATIONS

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

$$\begin{aligned} \mathbf{D} &= \varepsilon_0 \mathbf{E} + \mathbf{P} \\ \mathbf{B} &= \mu_0(\mathbf{H} + \mathbf{M}) \\ \mathbf{J} &= \sigma \mathbf{E} \end{aligned} \tag{4-11}$$

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

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

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 might have a nonzero $\mathbf{P}$ also when there is no electric field present.

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

To get a wave equation for the $\mathbf{E}$ field, for example, take the curl of the second equation in Equation 4-10 (previously divided by μ_0), and insert it into the time derivative of the first row in Equation 4-10

$$-\nabla \times \left(\frac{1}{\mu_0} \nabla \times \mathbf{E} + \frac{\partial \mathbf{M}}{\partial t} \right) = \sigma \frac{\partial \mathbf{E}}{\partial t} + \varepsilon_0 \frac{\partial^2 \mathbf{E}}{\partial t^2} + \frac{\partial^2 \mathbf{P}}{\partial t^2}$$

this is referred as curl-curl formulation in the literature (second order time derivatives and second order space derivatives).

LINEAR MATERIALS

In the simplest case linear materials, the polarization is directly proportional to the electric field, that is

$$\partial \mathbf{P} / \partial \mathbf{E} = \varepsilon_0 \chi_e \text{ and } \mathbf{P} = \varepsilon_0 \chi_e \mathbf{E}$$

where χ_e is the electric susceptibility (which can be a scalar or a second-rank tensor). Similarly, the magnetization is directly proportional to the magnetic field, or

$$\partial \mathbf{M}/\partial \mathbf{H} = \chi_m \text{ and } \mathbf{M} = \chi_m \mathbf{H}$$

where χ_m is the magnetic susceptibility.

As a consequence, for linear materials, the constitutive relations in Equation 4-11 can be written as

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

Here, $\varepsilon = \varepsilon_0 \varepsilon_r$ and $\mu = \mu_0 \mu_r$ are the permittivity and permeability of the material. The relative permittivity ε_r and the relative permeability μ_r are usually scalar properties but these can be second-rank symmetric (Hermitian) tensors for a general anisotropic material.

For general time-varying fields, Maxwell's equations in linear materials described in Equation 4-10 can be simplified to Maxwell-Ampere's law and Faraday's law:

$$\nabla \times \mathbf{H} = \sigma \mathbf{E} + \varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t}$$
$$\nabla \times \mathbf{E} = -\mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} \qquad (4\text{-}12)$$

The electric conductivity σ can also be a scalar or a second rank tensor. Another important assumption is that the relative permittivity ε_r, the relative permeability μ_r and the electric conductivity σ might change with position and orientation (inhomogeneous or anisotropic materials) but not with time.

FIRST ORDER IMPLEMENTATION OF MAXWELL EQUATIONS

In order to accommodate Maxwell's equations in the coefficients for the Wave Form PDE interface in the form

$$d_a \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \Gamma(\mathbf{u}) = \mathbf{f}$$

the curl of a vector is written in divergence form as

$$\nabla \times \mathbf{u} = \nabla \cdot \begin{bmatrix} 0 & u_3 & -u_2 \\ -u_3 & 0 & u_1 \\ u_2 & -u_1 & 0 \end{bmatrix} \tag{4-13}$$

where the divergence is applied on each row of the flux $\Gamma(\mathbf{u})$.

Maxwell's equations in 3D

$$\varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} = -\sigma \mathbf{E}$$
$$\mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \times \mathbf{E} = \mathbf{0}$$

are then accommodated to the Wave Form PDE as

$$d_E \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) = \mathbf{f}$$
$$d_H \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) = \mathbf{0}$$

with the "mass" coefficients

$$d_E = \varepsilon_0 \varepsilon_r \text{ and } d_H = \mu_0 \mu_r$$

the "flux" terms

$$\Gamma_E(\mathbf{H}) = -\begin{bmatrix} 0 & h_3 & -h_2 \\ -h_3 & 0 & h_1 \\ h_2 & -h_1 & 0 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 & -e_2 \\ -e_3 & 0 & e_1 \\ e_2 & -e_1 & 0 \end{bmatrix}$$

and the "source" term $\mathbf{f} = -\sigma \mathbf{E}$.

THE LAX-FRIEDRICHS FLUX PARAMETERS

When using SI units (or other) for the electromagnetic fields and material properties, the Lax-Friedrichs Flux Parameter are not dimensionless, and must have units of $\tau_E = 1/(2Z)$ for Ampere's law, and $\tau_H = Z/2$ for Faraday's law, where Z is the impedance of the medium.

In-plane E Field or In-plane H Field

In the general case, in 2D and 2D axisymmetric, solving for three variables for each field is still required. The "in-plane H" or "in-plane E" assumption simplifies the problem to only three dependent variables.

TM WAVES IN 2D

For TM waves in 2D, solve for an in-plane electric field vector and one out-of-plane variable for the magnetic field. Maxwell's equations then read

$$\begin{aligned} \varepsilon_0\varepsilon_r\frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma \cdot \mathbf{E} \\ \mu_0\mu_r\frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= 0 \end{aligned} \tag{4-14}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} 0 & -h_3 \\ h_3 & 0 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} e_2 & -e_1 \end{bmatrix} \tag{4-15}$$

The divergence on $\Gamma_E(\mathbf{H})$ is applied row-wise. The conductivity and permittivity tensors σ and ε_r represent in-plane material properties, while the relative permeability μ_r is an out-of-plane scalar property.

The default Lax-Friedrichs flux parameters are $\tau_E = 1/(2Z)$ for Ampere law, and the scalar $\tau_H = Z/2$ for Faraday's law, where Z is the impedance of a vacuum.

TE WAVES IN 2D

For TE waves in 2D, solve for an in-plane magnetic field vector and one out-of-plane variable for the electric field. Maxwell's equations then read

$$\begin{aligned} \varepsilon_0\varepsilon_r\frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma\mathbf{E} \\ \mu_0\mu_r\frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= 0 \end{aligned} \tag{4-16}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} -h_2 & h_1 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 \\ -e_3 & 0 \end{bmatrix} \tag{4-17}$$

The divergence of $\Gamma_H(\mathbf{E})$ is applied row-wise. The tensor of relative permeability μ_r represents in-plane material properties, while the relative permittivity ε_r and conductivity σ are out-of-plane scalar properties.

The default Lax-Friedrichs flux parameters are $\tau_E = 1/(2Z)$ for Ampere law, and two scalar $\tau_H = Z/2$ for Faraday's law, where Z is the impedance of a vacuum.

Fluxes as Dirichlet Boundary Conditions

Consider Maxwell's equations in 3D

$$\varepsilon_0\varepsilon_r\frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) = -\sigma\mathbf{E}$$

$$\mu_0\mu_r\frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) = \mathbf{0}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} 0 & -h_3 & h_2 \\ h_3 & 0 & -h_1 \\ -h_2 & h_1 & 0 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 & -e_2 \\ -e_3 & 0 & e_1 \\ e_2 & -e_1 & 0 \end{bmatrix}$$

and the divergence on $\Gamma_E(\mathbf{H})$ and $\Gamma_H(\mathbf{E})$ applied row-wise.

For Ampere's law, the normal to the flux term on exterior boundaries reads

$$\mathbf{n} \cdot \Gamma_E(\mathbf{H}) = -\mathbf{n} \times \mathbf{H}$$

and for Faraday's law

$$\mathbf{n} \cdot \Gamma_H(\mathbf{E}) = \mathbf{n} \times \mathbf{E}$$

which means that normal fluxes on external boundaries can only prescribe tangential components for the fields.

BOUNDARY CONDITIONS

The boundary conditions for outer boundaries are computed from the normal fluxes $\mathbf{n} \cdot \Gamma_H(\mathbf{E})$ and $\mathbf{n} \cdot \Gamma_E(\mathbf{H})$.

- Perfect electric conductor $\mathbf{n} \times \mathbf{E} = \mathbf{0}$, or zero tangential components for $\mathbf{E}$, is obtained by setting $\mathbf{n} \cdot \Gamma_H(\mathbf{E}) = \mathbf{0}$.

- Perfect magnetic conductor $\mathbf{n} \times \mathbf{H} = \mathbf{0}$, or zero tangential components for $\mathbf{H}$, is obtained by prescribing $\mathbf{n} \cdot \boldsymbol{\Gamma}_E(\mathbf{H}) = \mathbf{0}$.
- Electric field $\mathbf{n} \times \mathbf{E} = \mathbf{n} \times \mathbf{E}_0$, or $\mathbf{n} \cdot \Gamma_H(\mathbf{E}) = \mathbf{n} \times \mathbf{E}_0$.
- Magnetic field $\mathbf{n} \times \mathbf{H} = \mathbf{n} \times \mathbf{H}_0$, or $-\mathbf{n} \cdot \Gamma_E(\mathbf{H}) = \mathbf{n} \times \mathbf{H}_0$.
- For external boundaries, the surface currents BC means $\mathbf{n} \times \mathbf{H} = \mathbf{J}_s$, or $-\mathbf{n} \cdot \Gamma_E(\mathbf{H}) = \mathbf{J}_s$.

ABSORBING BOUNDARY CONDITION

A simple absorbing boundary can be implemented by setting $\mathbf{n} \times \mathbf{E} = Z\mathbf{H}$.

5

The ACDC Branch

This chapter summarizes the functionality of the electrical circuit interface found under the **AC/DC** branch ( ) in the **Model Wizard**.

In this chapter:

The Electrical Circuit User Interface

The **Electrical Circuit (cir)** interface ( ), found under the **AC/DC** branch ( ) in the **Model Wizard**, has the equations for modeling electrical circuits with or without connections to a distributed fields model, solving for the voltages, currents, and charges associated with the circuit elements.

When this interface is added, it adds a default **Ground Node** feature and associates that with node zero in the electrical circuit.

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

INTERFACE IDENTIFIER

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

The default identifier (for the first interface in the model) is `cir`.

- Theory for the Electrical Circuit User Interface
- Connecting to Electrical Circuits

The following are available from the context menu for this interface:

- Ground Node
- Resistor
- Capacitor
- Inductor
- Voltage Source
- Current Source
- Voltage-Controlled Voltage Source
- Voltage-Controlled Current Source
- Current-Controlled Voltage Source
- Current-Controlled Current Source
- Subcircuit Definition
- Subcircuit Instance
- NPN BJT
- n-Channel MOSFET
- Diode
- External I vs. U
- External U vs. I
- External I-Terminal
- SPICE Circuit Import

Ground Node

The **Ground Node** ( ) feature adds a ground node with the default node number zero to the electrical circuit. This is the default node in the Electrical Circuit interface. More ground nodes can be added but those must have unique node numbers and are by default given higher node numbers.

GROUND CONNECTION

Set the **Node name** for the ground node in the circuit. The convention is to use 0 (zero) for the ground node. If adding more ground nodes. each must have a unique node name (number).

Resistor

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

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the resistor. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

Enter the **Resistance** of the resistor.

Capacitor

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

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the capacitor. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

Enter the **Capacitance** of the capacitor.

Inductor

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

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the inductor. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

Enter the **Inductance** of the inductor.

Voltage Source

The **Voltage Source** ( ) feature connects a voltage source between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

Enter the **Source type** that should be adapted to the selected study type. It can be **DC-source**, **AC-source**, or a time-dependent **Sine source** or **Pulse source**. Depending on the choice of source, also specify the following parameters:

- For a DC-source, the **Voltage** V_{src} (default value: 1 V).
- For an AC-source: the **Voltage** V_{src} (default value: 1 V) and the **Phase** Θ (default value: 0 rad).
- For a sine source: the **Voltage** V_{src} (default value: 1 V), the **Offset** V_{off} (default value: 0 V), the **Frequency** (default value: 1 kHz), and the **Phase** Θ (default value: 0 rad).
- For a pulse source: the **Voltage** V_{src} (default value: 1 V), the **Offset** V_{off} (default value: 0 V), the **Delay** t_d (default value: 0s), the **Rise time** t_r and **Fall time** t_f (default values: 0 s), the **Pulse width** p_w (default value: 1 µs), and the **Period** T_{per} (default value: 2 µs).

All values are peak values rather than RMS.

For the AC source, the frequency is a global input set by the solver so do not use the **Sine source** unless the model is time dependent.

Current Source

The **Current Source** () feature connects a current source between two nodes in the electrical circuit.

NODE CONNECTIONS

Set the two **Node names** for the connecting nodes for the current source. The first node represents the positive reference terminal from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

Enter the **Source type** that should be adapted to the selected study type. It can be **DC-source**, **AC-source**, or a time-dependent **Sine source** or **Pulse source**. Depending on the choice of source, also specify the following parameters:

- For a DC-source, the **Current** i_{src} (default value: 1 A).

- For an AC-source: the **Current i_{src}** (default value: 1 A) and the **Phase** Θ (default value: 0 rad).
- For a sine source: the **Current i_{src}** (default value: 1 A), the **Offset i_{off}** (default value: 0 A), the **Frequency** (default value: 1 kHz), and the **Phase** Θ (default value: 0 rad).
- For a pulse source: the **Current i_{src}** (default value: 1 A), the **Offset i_{off}** (default value: 0 A), the **Delay t_d** (default value: 0s), the **Rise time t_r** and **Fall time t_f** (default values: 0 s), the **Pulse width p_w** (default value: 1 µs), and the **Period T_{per}** (default value: 2 µs).

All values are peak values rather than RMS.

For the AC source, the frequency is a global input set by the solver so do not use the **Sine source** unless the model is time dependent.

Voltage-Controlled Voltage Source

The **Voltage-Controlled Voltage Source** ( ) feature connects a voltage-controlled voltage source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

NODE CONNECTIONS

Specify four **Node names**: the first pair for the connection nodes for the voltage source and the second pair defining the input control voltage. The first node in a pair represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

Enter the voltage **Gain**. The resulting voltage is this number multiplied by the control voltage.

Voltage-Controlled Current Source

The **Voltage-Controlled Current Source** ( ) feature connects a voltage-controlled current source between two nodes in the electrical circuit. A second pair of nodes define the input control voltage.

NODE CONNECTIONS

Specify four **Node names**: the first pair for the connection nodes for the current source and the second pair defining the input control voltage. The first node in a pair represents the positive voltage reference terminal or the one from which the current flows through the source to the second node. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

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

Current-Controlled Voltage Source

The **Current-Controlled Voltage Source** () feature connects a current-controlled voltage source between two nodes in the electrical circuit. The input control current is the one flowing through a named device that must be a two-pin device.

NODE CONNECTIONS

Set two **Node names** for the connection nodes for the voltage source. The first node in a pair represents the positive reference terminal. If the ground node is involved, the convention is to use 0 (zero) for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

Enter the voltage **Gain** and the **Device** (any two-pin device) name. The resulting voltage is this number multiplied by the control current through the named **Device** (any two-pin device). Thus it formally has the unit of resistance.

Current-Controlled Current Source

The **Current-Controlled Current Source** () feature connects a current-controlled current source between two nodes in the electrical circuit. The input control current is the one flowing through a named device that must be a two-pin device.

NODE CONNECTIONS

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

is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

DEVICE PARAMETERS

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

Subcircuit Definition

The **Subcircuit Definition** ( ) feature is used to define subcircuits. Right-click a **Subcircuit Definition** node to add all circuit nodes available except for the subcircuit definition feature itself. Also right-click to **Rename** the node.

SUBCIRCUIT PINS

Define the **Pin names** at which the subcircuit connects to the main circuit or to other subcircuits when referenced by a Subcircuit Instance node. The **Pin names** refer to circuit nodes in the subcircuit. The order in which the **Pin names** are defined is the order in which they are referenced by a Subcircuit Instance node.

Subcircuit Instance

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

NODE CONNECTIONS

Select the **Name of subcircuit link** from the list of defined subcircuits in the circuit model and the circuit **Node names** at which the subcircuit instance connects to the main circuit or to another subcircuit if used therein.

NPN BJT

The **NPN BJT** device model ( ) is a large-signal model for an NPN bipolar junction transistor (BJT). It is an advanced device model and no thorough description and motivation of the many input parameters are attempted here. Many device manufacturers provide model input parameters for this BJT model. For any particular make of BJT, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Specify three **Node names** for the connection nodes for the **NPN BJT** device. These represent the *collector*, *base*, and *emitter* nodes, respectively. If the ground node is involved, the convention is to use 0 (zero) for this but it is allowed to have more than one ground node provided it has been given a unique node name.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular BJT, the device manufacturer should be the primary source of information.

The interested reader is referred to Ref. 2 for more details on semiconductor modeling within circuits.

For an explanation of the **Model Parameters** see NPN Bipolar Transistor.

n-Channel MOSFET

The **n-Channel MOSFET** device model () is a large-signal model for an n-Channel MOS transistor (MOSFET). It is an advanced device model and no thorough description and motivation of the many input parameters are attempted here. Many device manufacturers provide model parameters for this MOSFET model. For any particular make of MOSFET, the device manufacturer should be the primary source of information.

NODE CONNECTIONS

Specify four **Node names** for the connection nodes for the **n-Channel MOSFET** device. These represent the *drain*, *gate*, *source*, and *bulk* nodes, respectively. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular MOSFET, the device manufacturer should be the primary source of information.

The interested reader is referred to Ref. 2 for more details on semiconductor modeling within circuits.

For an explanation of the **Model Parameters** see n-Channel MOS Transistor.

Diode

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

NODE CONNECTIONS

Specify two **Node names** for the positive and negative nodes for the **Diode** device. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

MODEL PARAMETERS

Specify the **Model Parameters**. Reasonable defaults are provided but for any particular diode, the device manufacturer should be the primary source of information.

For an explanation of the **Model Parameters** see Diode.

External I vs. U

The **External I vs. U** ( ) feature connects an arbitrary voltage measurement (for example, a circuit terminal or circuit port boundary or a coil domain from another physics) as a source between two nodes in the electrical circuit. The resulting circuit

current from the first node to the second node is typically coupled back as a prescribed current source in the context of the voltage measurement.

NODE CONNECTIONS

Specify the two **Node names** for the connecting nodes for the voltage source. The first node represents the positive reference terminal. If the ground node is involved, the convention is to use zero for this but it is allowed to have more than one ground node provided it has been given a unique node name.

EXTERNAL DEVICE

Enter the source of the **Voltage**. If circuit or current excited terminals or circuit ports are defined on boundaries or a multiturn coil domains is defined in other physics interfaces, these display as options in the **Voltage** list. Also select the **User defined** option and enter your own voltage variable, for example, using a suitable coupling operator. For inductive or electromagnetic wave propagation models, the voltage measurement must be performed as an integral of the electric field because the electric potential only does not capture induced EMF. Also the integration must be performed over a distance that is short compared to the local wavelength.

Except for when coupling to a circuit terminal or circuit port, the current flow variable must be manually coupled back in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention `cirn.IvsUm_i`, where `cirn` is the tag of the Electrical Circuit interface node and `IvsUm` is the tag of the **External I vs. U** node. The mentioned tags are typically displayed within curly braces `{}` in the model tree.

Model Couplings in the *COMSOL Multiphysics Reference Manual*

External U vs. I

The **External U vs. I** ( ) feature connects an arbitrary current measurement (for example, from another) as a source between two nodes in the electrical circuit. The

resulting circuit voltage between the first node and the second node is typically coupled back as a prescribed voltage source in the context of the current measurement.

NODE CONNECTIONS

Specify the two **Node names** for the connecting nodes for the current source. The current flows from the first node to the second node. If the ground node is involved, the convention is to use 0 (zero) for this but it is allowed to have more than one ground node provided it has been given a unique node name.

EXTERNAL DEVICE

Enter the source of the **Current**. Voltage excited terminals or lumped ports defined on boundaries in other interfaces are natural candidates but do not appear as options in the **Voltage** list because those do not have an accurate built-in current measurement variable. A **User defined** option must be selected and a current variable entered, for example, using a suitable coupling operator.

The voltage variable must be manually coupled back in the electrical circuit to the context of the current measurement. This applies also when coupling to a voltage excited terminal or lumped port. The name of this voltage variable follows the convention `cirn.UvsIm_v`, where `cirn` is the tag of the Electrical Circuit interface node and `UvsIm` is the tag of the **External U vs. I** node. The mentioned tags are typically displayed within curly braces `{}` in the model tree.

Model Couplings in the *COMSOL Multiphysics Reference Manual*

External I-Terminal

The **External I-Terminal** () feature connects an arbitrary voltage-to-ground measurement (for example, a circuit terminal boundary from another interface) as a voltage-to-ground assignment to a node in the electrical circuit. The resulting circuit current from the node is typically coupled back as a prescribed current source in the context of the voltage measurement. This node does not apply when coupling to inductive or electromagnetic wave propagation models because then voltage must be

defined as a line integral between two points rather than a single point measurement of electric potential. For such couplings, use the **External I vs. U** node instead.

NODE CONNECTIONS

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

EXTERNAL TERMINAL

Enter the source of the **Voltage**. If circuit- or current-excited terminals are defined on boundaries in other interfaces, these display as options in the **Voltage** list. Also select the **User defined** option and enter a voltage variable, for example, using a suitable coupling operator.

Except for when coupling to a circuit terminal, the current flow variable must be manually coupled back in the electrical circuit to the context of the voltage measurement. This applies also when coupling to a current excited terminal. The name of this current variable follows the convention `cirn.termIm_i`, where `cirn` is the tag of the Electrical Circuit interface node and `termIm` is the tag of the **External I-Terminal** node. The mentioned tags are typically displayed within curly braces `{}` in the model tree.

Model Couplings in the *COMSOL Multiphysics Reference Manual*

SPICE Circuit Import

Right-click the **Electrical Circuit** ( ) feature node to import an existing SPICE netlist (select **Import Spice Netlist**). A window opens—enter a file location or browse your directories to find one. The default file extension for a SPICE netlist is `.cir`. The SPICE circuit import translates the imported netlist into Electrical Circuit interface nodes so these define the subset of SPICE features that can be imported.

See SPICE Import for more details on the supported SPICE commands.

Theory for the Electrical Circuit User Interface

The Electrical Circuit User Interface theory is discussed in this section:

- Electric Circuit Modeling and the Semiconductor Device Models
- NPN Bipolar Transistor
- n-Channel MOS Transistor
- Diode
- SPICE Import
- References for the Electrical Circuit User Interface

Connecting to Electrical Circuits

Electric Circuit Modeling and the Semiconductor Device Models

Electrical circuit modeling capabilities are useful when simulating all sorts of electrical and electromechanical devices ranging from heaters and motors to advanced plasma reactors in the semiconductor industry. There are two fundamental ways that an electrical circuit model relates to a physical field model.

- The field model is used to get a better, more accurate description of a single device in the electrical circuit model.
- The electrical circuit is used to drive or terminate the device in the field model in such a way that it makes more sense to simulate both as a tightly coupled system.

The Electrical Circuit interface makes it is possible to add nodes representing circuit elements directly to the model tree in a COMSOL Multiphysics model. The circuit variables can then be connected to a physical device model to perform co-simulations of circuits and multiphysics. The model acts as a device connected to the circuit so that its behavior is analyzed in larger systems.

The fundamental equations solved by the electrical circuit interface are Kirchhoff's circuit laws, which in turn can be deduced from Maxwell's equations. The supported study types are Stationary, Frequency Domain, and Time Dependent.

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

NPN Bipolar Transistor

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

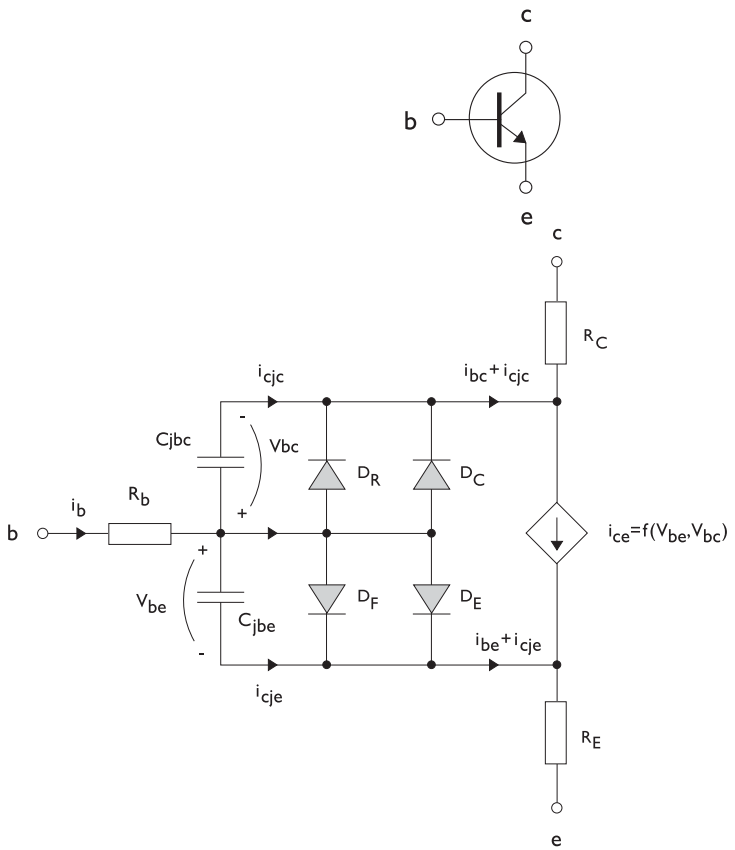


Figure 5-1: A circuit for the bipolar transistor.

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

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

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

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

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

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

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also two capacitances that use the same formula as the junction capacitance of the diode model. In the parameter names below, replace *x* with *C* for the base-collector capacitance and *E* for the base-emitter capacitance.

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

The model parameters are listed in the table below.

TABLE 5-1: BIPOLAR TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
B_F	100	Ideal forward current gain
B_R	1	Ideal reverse current gain
C_{JC}	0 F/m^2	Base-collector zero-bias depletion capacitance
C_{JE}	0 F/m^2	Base-emitter zero-bias depletion capacitance

TABLE 5-1: BIPOLAR TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
F_C	0.5	Breakdown current
I_{KF}	Inf (A/m^2)	Corner for forward high-current roll-off
I_{KR}	Inf (A/m^2)	Corner for reverse high-current roll-off
I_S	1e-15 A/m^2	Saturation current
I_{SC}	0 A/m^2	Base-collector leakage saturation current
I_{SE}	0 A/m^2	Base-emitter leakage saturation current
M_{JC}	1/3	Base-collector grading coefficient
M_{JE}	1/3	Base-emitter grading coefficient
N_C	2	Base-collector ideality factor
N_E	1.4	Base-emitter ideality factor
N_F	1	Forward ideality factor
N_R	1	Reverse ideality factor
R_B	0 Ωm^2	Base resistance
R_{BM}	0 Ωm^2	Minimum base resistance
R_C	0 Ωm^2	Collector resistance
R_E	0 Ωm^2	Emitter resistance
T_{NOM}	298.15 K	Device temperature
V_{AF}	Inf (V)	Forward Early voltage
V_{AR}	Inf (V)	Reverse Early voltage
V_{JC}	0.71 V	Base-collector built-in potential
V_{JE}	0.71 V	Base-emitter built-in potential

n-Channel MOS Transistor

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

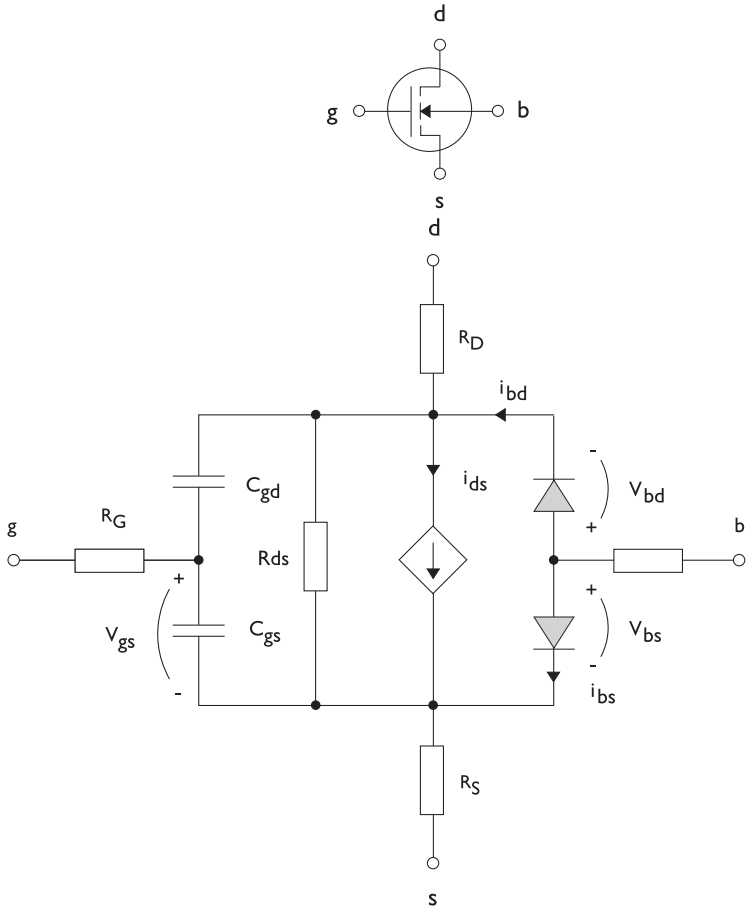


Figure 5-2: A circuit for the MOS transistor.

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

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

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

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

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

$$V_T = \frac{k_B T_{NOM}}{q}$$

There are also several capacitances between the terminals

$$C_{gd} = C_{gd0}W$$

$$C_{gs} = C_{gs0}W$$

$$C_{jbd} = C_{BD} \times \begin{cases} \left(1 - \frac{v_{bd}}{P_B}\right)^{-M_J} & v_{bx} < F_C P_B \\ (1-F_C)^{-1-M_J}\left(1 - F_C(1+M_J) + M_J\frac{v_{bx}}{P_B}\right) & v_{bx} \geq F_C P_B \end{cases}$$

The model parameters are as follows:

TABLE 5-2: MOS TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
C_{BD}	0 F/m	Bulk-drain zero-bias capacitance
C_{GDO}	0 F/m	Gate-drain overlap capacitance
C_{GSO}	0 F/m	Gate-source overlap capacitance
F_C	0.5	Capacitance factor
I_S	1e-13 A	Bulk junction saturation current
K_P	2e-5 A/V^2	Transconductance parameter
L	50e-6 m	Gate length
M_J	0.5	Bulk junction grading coefficient

TABLE 5-2: MOS TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
N	1	Bulk junction ideality factor
P_B	0.75 V	Bulk junction potential
R_B	0 Ω	Bulk resistance
R_D	0 Ω	Drain resistance
R_{DS}	Inf (Ω)	Drain-source resistance
R_G	0 Ω	Gate resistance
R_S	0 Ω	Source resistance
T_{NOM}	298.15 K	Device temperature
V_{TO}	0 V	Zero-bias threshold voltage
W	50e-6 m	Gate width
Γ (GAMMA)	0 $V^{0.5}$	Bulk threshold parameter
Φ (PHI)	0.5 V	Surface potential
Λ (LAMBDA)	0 1/V	Channel-length modulation

Diode

Figure 5-3 illustrates equivalent circuit for the diode.

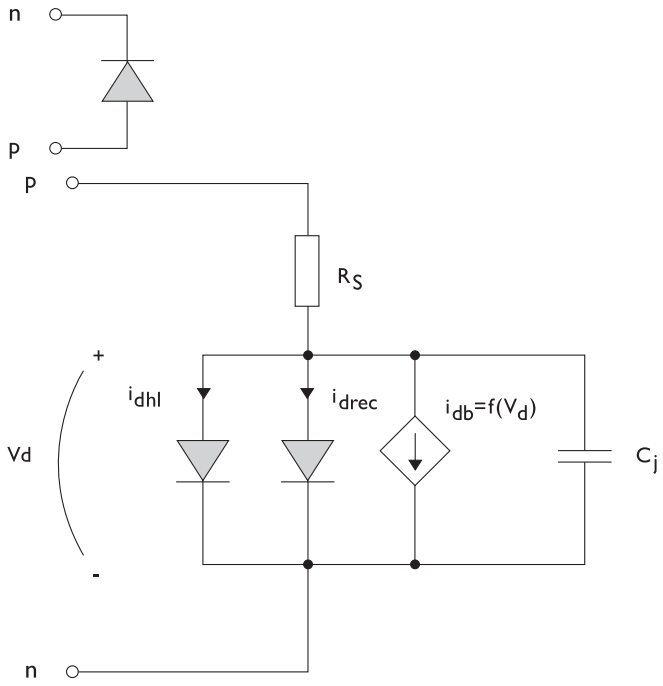


Figure 5-3: A circuit for the diode.

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

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

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

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

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

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

$$V_T = \frac{k_B T_{NOM}}{q}$$

where the following model parameters are required

TABLE 5-3: DIODE TRANSISTOR MODEL PARAMETERS

PARAMETER	DEFAULT	DESCRIPTION
B_V	Inf (V)	Reverse breakdown voltage
C_{J0}	0 F	Zero-bias junction capacitance
F_C	0.5	Forward-bias capacitance coefficient
I_{BV}	1e-09 A	Current at breakdown voltage
I_{KF}	Inf (A)	Corner for high-current roll-off
I_S	1e-13 A	Saturation current
M	0.5	Grading coefficient
N	1	Ideality factor
N_{BV}	1	Breakdown ideality factor
N_R	2	Recombination ideality factor
R_S	0 Ω	Series resistance
T_{NOM}	298.15 K	Device temperature
V_J	1.0 V	Junction potential

SPICE Import

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

The Electrical Circuit interface supports the following device models:

TABLE 5-4: SUPPORTED SPICE DEVICE MODELS

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

The interface also supports the `.subckt` statement, that will be represented in COMSOL by a Subcircuit Definition node, and the `.include` statement. SPICE commands are interpreted case-insensitively.

References for the Electrical Circuit User Interface

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

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

6

The Electromagnetic Heating Branch

This chapter describes The Microwave Heating User Interface, which is found under the **Heat Transfer>Electromagnetic Heating** branch ( ) in the **Model Wizard**. This interfaces combines the features of an Electromagnetic Waves interface from the RF Module with those of the Heat Transfer interface.

The Microwave Heating User Interface

The **Microwave Heating (mh)** user interface ( ), found under the **Heat Transfer>Electromagnetic Heating** branch ( ) in the **Model Wizard**, combines the features of an Electromagnetic Waves interface with those of the Heat Transfer interface. The predefined interaction adds the electromagnetic losses from the electromagnetic waves as a heat source. This interface is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale (adiabatic assumption). It is associated with two predefined study types:

- Frequency-Stationary—Time-harmonic electromagnetic waves and stationary heat transfer
- Frequency-Transient—Time-harmonic electromagnetic waves and transient heat transfer

When this interface is added, these default nodes are also added to the **Model Builder**—**Microwave Heating Model**, **Electromagnetic Heat Source**, **Boundary Electromagnetic Heat Source**, **Thermal Insulation**, **Perfect Electric Conductor**, and **Initial Values**.

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

Except where noted in this section, the Microwave Heating interface shares most of its settings windows with The Electromagnetic Waves, Frequency Domain User Interface (described in this guide) and The Heat Transfer User Interface (described in the *COMSOL Multiphysics Reference Manual*).

INTERFACE IDENTIFIER

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

The default identifier (for the first interface in the model) is `mh`.

DOMAIN SELECTION

The default setting is to include **All domains** in the model to define the dependent variables and the equations. To choose specific domains, select **Manual** from the **Selection** list.

PHYSICAL MODEL

Select the **Out-of-plane heat transfer** check box (2D models only) to include heat transfer out of the plane.

If your license includes the Heat Transfer Module, you can select additional options, which are described in the *Heat Transfer Module User's Guide*:

- **Surface-to-surface radiation** check box to include surface-to-surface radiation as part of the heat transfer. This adds a **Radiation Settings** section.
- **Radiation in participating media** check box to include radiation in participating media as part of the heat transfer. This adds a **Participating media Settings** section.
- **Heat Transfer in biological tissue** check box to enable the **Biological Tissue** node.

SETTINGS

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

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

PORT SWEEP SETTINGS

Enter a **Reference impedance** Z_{ref} (SI unit: Ω). The default is 50 Ω.

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

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

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

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

DEPENDENT VARIABLES

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

ADVANCED SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. By default the **Enable conversions between material and spatial frames** check box is selected. Normally these settings do not need to be changed.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**.

Select **Linear**, **Quadratic**, **Cubic**, **Quartic**, or (in 2D) **Quintic** for the **Temperature**, **Surface radiosity**, and **Electric field**.

Specify the **Value type when using splitting of complex variables**—**Real** or **Complex**.

- Show More Physics Options
- Microwave Heating Model
- Domain, Boundary, Edge, Point, and Pair Nodes for the Microwave Heating User Interface
- About Frames in Heat Transfer in the *COMSOL Multiphysics Reference Manual*

- Microwave Oven: Model Library path **RF_Module/Microwave_Heating/microwave_oven**
- RF Heating: Model Library path **RF_Module/Microwave_Heating/rf_heating**

Domain, Boundary, Edge, Point, and Pair Nodes for the Microwave Heating User Interface

Because The Microwave Heating User Interface is a multiphysics interface, almost every node is shared with, and described for, other interfaces. Below are links to the domain, boundary, edge, point, and pair nodes as indicated.

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

These nodes are described in this section:

- Microwave Heating Model
- Electromagnetic Heat Source
- Initial Values

These nodes (and subnodes) are described for the Electromagnetic Waves, Frequency Domain interface (listed in alphabetical order):

- Divergence Constraint
- Edge Current
- Electric Field
- Electric Point Dipole
- Far-Field Domain
- Impedance Boundary Condition
- Line Current (Out-of-Plane) (2D axisymmetric models)
- Lumped Port
- Magnetic Current
- Magnetic Field
- Magnetic Point Dipole
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Condition
- Port
- Scattering Boundary Condition
- Surface Current
- Transition Boundary Condition
- Wave Equation, Electric

These nodes are described for the Heat Transfer and Joule Heating interfaces in the *COMSOL Multiphysics Reference Manual* (listed in alphabetical order):

- Boundary Electromagnetic Heat Source
- Boundary Heat Source
- Heat Flux
- Heat Source
- Heat Transfer in Fluids
- Heat Transfer in Solids
- Line Heat Source
- Outflow
- Point Heat Source
- Surface-to-Ambient Radiation
- Symmetry
- Temperature
- Thermal Insulation
- Thin Thermally Resistive Layer

In the *COMSOL Multiphysics Reference Manual*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs

If you also have the Heat Transfer Module, there are several other feature nodes available and described in the *Heat Transfer Module User's Guide*.

The links to nodes described the *COMSOL Multiphysics Reference Manual* do not work in the PDF, only from the on line help.

Microwave Heating Model

The **Microwave Heating Model** node has settings to define the displacement field, magnetic field, conduction current, heat conduction, and thermodynamics.

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

MODEL INPUTS

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

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes.

DISPLACEMENT FIELD

Select a **Displacement field model**—**Relative permittivity**, **Refractive index**, **Loss tangent**, or **Dielectric loss**. Select:

- **Relative permittivity** to specify the relative permittivity or take it from the material.
- **Refractive index** to specify the real and imaginary parts of the refractive index or take them from the material. This assumes a relative permeability of unity and zero conductivity.

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

- **Loss tangent** to specify a loss tangent for dielectric losses or take it from the material. This assumes zero conductivity.
- **Dielectric loss** to specify the real and imaginary parts of the relative permittivity or take them from the material.

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

MAGNETIC FIELD

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

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

- By default the **Relative permeability** μ_r uses values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix.
- If **Magnetic losses** is selected, the default values for μ' and μ'' are taken **From material**. Select **User defined** to enter different values.

CONDUCTION CURRENT

By default, the **Electrical conductivity** σ (SI unit: S/m) uses values **From material**. If **User defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the current and enter values or expressions in the field or matrix.

If **Linearized resistivity** is selected, the default values for the **Reference temperature** T_{ref} (SI unit: K), **Resistivity temperature coefficient** α (SI unit: 1/K), and **Reference resistivity** ρ_0 (SI unit: Ω m) are taken **From material**. Select **User defined** to enter other values or expressions for any of these variables.

HEAT CONDUCTION

The default **Thermal conductivity** k (SI unit: W/(m·K)) uses values **From material**. If **User-defined** is selected, choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity and enter another value or expression.

THERMODYNAMICS

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

Electromagnetic Heat Source

The **Electromagnetic Heat Source** node represents the electromagnetic losses, Q_e (SI unit: W/m^3), as a heat source in the heat transfer part of the model. It is given by

$$Q_e = Q_{rh} + Q_{ml}$$

where the resistive losses are

$$Q_{rh} = \frac{1}{2}\text{Re}(\mathbf{J} \cdot \mathbf{E}^*)$$

and the magnetic losses are

$$Q_{\mathrm{ml}} = \frac{1}{2}\mathrm{Re}(i\omega \mathbf{B} \cdot \mathbf{H}^*)$$

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

Initial Values

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

DOMAIN SELECTION

For a default node, the setting inherits the selection from the parent node, and cannot be edited; that is, the selection is automatically selected and is the same as for the interface. When nodes are added from the context menu, you can select **Manual** from the **Selection** list to choose specific domains or select **All domains** as required.

INITIAL VALUES

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

7

Glossary

This Glossary of Terms contains finite element modeling terms in an electromagnetic waves context. For mathematical terms as well as geometry and CAD terms specific to the COMSOL Multiphysics® software and documentation, see the glossary in the *COMSOL Multiphysics Reference Manual*. For references to more information about a term, see the index.

Glossary of Terms

absorbing boundary A boundary that lets an electromagnetic wave propagate through the boundary without reflections.

anisotropy Variation of material properties with direction.

constitutive relation The relation between the **D** and **E** fields and between the **B** and **H** fields. These relations depend on the material properties.

cutoff frequency The lowest frequency for which a given mode can propagate through, for example, a waveguide or optical fiber.

edge element See *vector element*.

eigenmode A possible propagating mode of, for example, a waveguide or optical fiber.

electric dipole Two equal and opposite charges $+q$ and $-q$ separated a short distance d. The electric dipole moment is given by $\mathbf{p} = q\mathbf{d}$, where $\mathbf{d}$ is a vector going from $-q$ to $+q$.

gauge transformation A variable transformation of the electric and magnetic potentials that leaves Maxwell's equations invariant.

lumped port A type of port feature. Use the lumped port to excite the model with a voltage, current, or circuit input. The lumped port must be applied between two metallic objects, separated by much less than a wavelength.

magnetic dipole A small circular loop carrying a current. The magnetic dipole moment is $\mathbf{m} = IA\mathbf{e}$, where I is the current carried by the loop, A its area, and $\mathbf{e}$ a unit vector along the central axis of the loop.

Maxwell's equations A set of equations, written in differential or integral form, stating the relationships between the fundamental electromagnetic quantities.

Nedelec's edge element See *vector element*.

perfect electric conductor (PEC) A material with high electrical conductivity, modeled as a boundary where the electric field is zero.

perfect magnetic conductor A material with high permeability, modeled as a boundary where the magnetic field is zero.

phasor A complex function of space representing a sinusoidally varying quantity.

quasi-static approximation The electromagnetic fields are assumed to vary slowly, so that the retardation effects can be neglected. This approximation is valid when the geometry under study is considerably smaller than the wavelength.

surface current density Current density defined on the surface. The component normal to the surface is zero. The unit is A/m.

vector element A finite element often used for electromagnetic vector fields. The tangential component of the vector field at the mesh edges is used as a degree of freedom. Also called *Nedelec's edge element* or just *edge element*.

Index