

Wave Optics Module

User's Guide



Wave Optics Module User's Guide

© 1998–2013 COMSOL

Protected by U.S. Patents 7,519,518; 7,596,474; and 7,623,991. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/sla) and may be used or copied only under the terms of the license agreement.

COMSOL, COMSOL Multiphysics, Capture the Concept, COMSOL Desktop, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/tn.

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. CM023501

C o n t e n t s

Chapter 1: Introduction

About the Wave Optics Module	2
About the Wave Optics Module	2
What Problems Can You Solve?	3
The Wave Optics Module Physics Guide	4
Selecting the Study Type	5
The Wave Optics Module Modeling Process	6
Where Do I Access the Documentation and Model Library?	9
Overview of the User's Guide	12

Chapter 2: Wave Optics Modeling

Preparing for Wave Optics Modeling	14
Simplifying Geometries	15
2D Models	15
3D Models	16
Using Efficient Boundary Conditions	17
Applying Electromagnetic Sources	18
Meshing and Solving.	18
An Example — A Directional Coupler	20
Introduction	20
Model Definition	21
Results and Discussion.	22
Reference	26
Modeling Instructions	26

Periodic Boundary Conditions	41
Scattered Field Formulation	42
Modeling with Far-Field Calculations	43
Far-Field Support in the Electromagnetic Waves, Frequency Domain	
User Interface	43
The Far Field Plots	44
Maxwell's Equations	46
Introduction to Maxwell's Equations	46
Constitutive Relations	47
Potentials.	48
Electromagnetic Energy	49
Material Properties	50
Boundary and Interface Conditions	52
Phasors	52
Special Calculations	54
S-Parameter Calculations.	54
Far-Field Calculations Theory	57
References	58
S-Parameters and Ports	59
S-Parameters in Terms of Electric Field	59
S-Parameter Calculations: Ports	60
S-Parameter Variables	60
Port Sweeps and Touchstone Export	60
Lossy Eigenvalue Calculations	61
Eigenfrequency Analysis	61
Mode Analysis	63

Chapter 3: The Optics Branch

The Electromagnetic Waves, Frequency Domain User Interface	68
Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Frequency Domain Interface	70
Wave Equation, Electric	72
Initial Values.	77
External Current Density.	77
Far-Field Domain	78
Far-Field Calculation	78
Perfect Electric Conductor	79
Perfect Magnetic Conductor	81
Port.	82
Circular Port Reference Axis	86
Diffraction Order	87
Periodic Port Reference Point	88
Electric Field	89
Magnetic Field	90
Scattering Boundary Condition	90
Impedance Boundary Condition	92
Surface Current	93
Transition Boundary Condition	94
Periodic Condition	95
Magnetic Current	96
Edge Current	97
Electric Point Dipole	97
Magnetic Point Dipole	98
Line Current (Out-of-Plane)	98
The Electromagnetic Waves, Transient User Interface	100
Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Transient User Interface.	102
Wave Equation, Electric	104

Initial Values.	106
The Electromagnetic Waves, Time Explicit User Interface	107
Domain, Boundary, and Pair Nodes for the Electromagnetic Waves, Time Explicit User Interface.	109
Wave Equations	109
Initial Values.	111
Electric Current Density	112
Magnetic Current Density	113
Electric Field	113
Perfect Electric Conductor	114
Magnetic Field	114
Perfect Magnetic Conductor	115
Surface Current Density	115
Low Reflecting Boundary.	116
Flux/Source	117
The Electromagnetic Waves, Beam Envelopes User Interface	118
Domain, Boundary, Edge, and Point Nodes for the Electromagnetic Waves, Beam Envelopes Interface	121
Wave Equation, Beam Envelopes	123
Initial Values.	124
Electric Field	124
Magnetic Field	125
Scattering Boundary Condition	126
Surface Current	127
Theory for the Electromagnetic Waves User Interfaces	129
Introduction to the User Interface Equations	129
Frequency Domain Equation	129
Time Domain Equation	136
Vector Elements	137
Eigenfrequency Calculations.	138
Theory for the Electromagnetic Waves, Time Explicit User Interface	139
The Equations	139
In-plane E Field or In-plane H Field	143

Fluxes as Dirichlet Boundary Conditions 144

Chapter 4: Glossary

Glossary of Terms **148**

Introduction

This guide describes the Wave Optics Module, an optional add-on package for COMSOL Multiphysics® designed to assist you to set-up and solve electromagnetic wave problems at optical frequencies.

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.

In this chapter:

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

About the Wave Optics Module

These topics are included in this section:

- [About the Wave Optics Module](#)
- [What Problems Can You Solve?](#)
- [The Wave Optics Module Physics Guide](#)
- [Selecting the Study Type](#)
- [The Wave Optics Module Modeling Process](#)
- [Where Do I Access the Documentation and Model Library?](#)

About the Wave Optics Module

The Wave Optics Module extends the functionality of the physics user interfaces of the base package for COMSOL Multiphysics. The details of the physics user interfaces and study types for the Wave Optics Module are listed in the table. The functionality of the COMSOL Multiphysics base package is given in the *COMSOL Multiphysics Reference Manual*.



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](#).
-

The Wave Optics Module solves problems in the field of electromagnetic waves at optical frequencies (corresponding to wavelengths in the nano- to micrometer range). 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 simulations and design of optical applications in virtually all areas where you find electromagnetic waves, such as:

- Optical fibers
- Photonic waveguides

- Photonic crystals
- Nonlinear optics
- Laser resonator design
- 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.

What Problems Can You Solve?







The Wave Optics Module allows you to make high-frequency electromagnetic wave simulations. It distinguishes itself from the AC/DC Module, in that the AC/DC Module targets quasi-static simulations, where the size of the computational domain is small compared to the wavelength.

Both the RF and the Wave Optics Module can handle high-frequency electromagnetic wave simulations. However, with the Wave Optics Module you can do time-harmonic simulations of domains that are much larger than the wavelength. This situation is typical for optical phenomena, components and systems. Due to the relatively weak coupling between waves in optical materials, the interaction lengths are often much larger than the wavelength. This applies to linear couplers, like directional couplers and fiber Bragg gratings, and nonlinear phenomena, like second harmonic generation, self-phase modulation, *etc.* With the Wave Optics Module, these kinds of problems are directly addressable, without huge computer memory requirements.

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 Wave Optics 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 Wave Optics 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.

PHYSICS USER INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
 Optics				
 Wave Optics				
Electromagnetic Waves, Beam Envelopes		ewbe	3D, 2D, 2D axisymmetric	boundary mode analysis; eigenfrequency; frequency domain; frequency-domain modal
Electromagnetic Waves, Frequency Domain		ewfd	3D, 2D, 2D axisymmetric	boundary mode analysis; eigenfrequency; frequency domain; frequency-domain modal; mode analysis (2D and 2D axisymmetric models only)
Electromagnetic Waves, Time Explicit		teew	3D, 2D, 2D axisymmetric	time dependent
Electromagnetic Waves, Transient		ewt	3D, 2D, 2D axisymmetric	eigenfrequency; time dependent; time dependent modal

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 and Electromagnetic Waves, Time Explicit interfaces). 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 and the Electromagnetic Waves, Beam Envelopes interfaces 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.

For nonlinear problems you can apply a Frequency Domain study after a linearization of the problem, which assumes that the distortion of the sinusoidal signal is small. You can also couple waves at different frequencies, for example in applications like second harmonic generation, by coupling several interfaces, defined for the different frequencies, using weak expression coupling terms.

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.







COMPARING THE ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN AND THE ELECTROMAGNETIC WAVES, BEAM ENVELOPES INTERFACES

Both the Electromagnetic Waves, Frequency Domain and the Electromagnetic Waves, Beam Envelopes interfaces solve the time-harmonic Maxwell's equations. For the Frequency Domain interface, the dependent variable is the total electric field. Since the electric field has a spatial variation on the scale of a wavelength, the maximum mesh element size must be a fraction of a wavelength. If this mesh requirement is fulfilled, the Frequency Domain interface is very flexible for solving both propagation and scattering problems.

For many optical applications the propagation length is much longer than the wavelength. For instance, a typical optical wavelength is 1 μm , but the propagation length can easily be on the milli- to centimeter scale. To apply the Frequency Domain interface to this kind of problems, requires a large amount of available memory. However, many problems are such that the electric field can be factored into a slowly varying amplitude factor and a rapidly varying phase factor. The Electromagnetic Waves, Beam Envelopes interface is based on this assumption. Thus, this interface assumes a prescribed rapidly varying phase factor and solves for the slowly varying amplitude factor. Thereby it can be used for solving problems extending over domains that are a large number of wavelengths long, without requiring the use of large amounts of memory.

The Wave Optics 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 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.


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.




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
Interface Identifier	Predefined Physics Variables Variable Naming Convention and Scope Viewing Node Names, Identifiers, Types, and Tags
Material Type	Materials

SECTION	CROSS REFERENCE
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 *Wave Optics Module User's Guide* gets you started with modeling using COMSOL Multiphysics®. The information in this guide is specific to this module. Instructions on 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.

MODELING WITH THE WAVE OPTICS MODULE

The [Wave Optics 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 Wave Optics Modeling](#), [Simplifying Geometries](#), and [Scattered Field Formulation](#).

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

OPTICS

The [Optics Branch](#) chapter describes these interfaces. The underlying theory is also included at the end of the chapter.

- [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, Beam Envelopes User Interface](#), which analyzes frequency domain electromagnetic waves, and uses time-harmonic and eigenfrequency studies, boundary mode analysis and frequency domain modal.
- [The Electromagnetic Waves, Transient User Interface](#), which supports the time dependent study type.
- [The Electromagnetic Waves, Time Explicit User Interface](#), which solves a transient wave equation for both the electric and magnetic fields.

Wave Optics Modeling

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

In this chapter:

- [Preparing for Wave Optics Modeling](#)
- [Simplifying Geometries](#)
- [An Example — A Directional Coupler](#)
- [Periodic Boundary Conditions](#)
- [Scattered Field Formulation](#)
- [Modeling with Far-Field Calculations](#)
- [Maxwell's Equations](#)
- [Special Calculations](#)
- [S-Parameters and Ports](#)
- [Lossy Eigenvalue Calculations](#)
- [Electromagnetic Quantities](#)

Preparing for Wave Optics 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 assume that the electric field has a slowly varying amplitude?
- 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](#)
- [3D Models](#)
- [Using Efficient Boundary Conditions](#)
- [Applying Electromagnetic Sources](#)
- [Meshing and Solving](#)

2D Models

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

CARTESIAN COORDINATES

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

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

particular dimension. The second approach is when there is a problem where the influence of the finite extension in the third dimension can be neglected.

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).



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](#), [The Electromagnetic Waves, Transient User Interface](#), or [The Electromagnetic Waves, Beam Envelopes 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.
- Sometimes the electric field can be decomposed into a product of a slowly varying amplitude function and a prescribed rapidly varying phase function. In this case it is advantageous to reformulate the equations and solve for the slowly varying amplitude function. Thereby the mesh only need to resolve the slowly varying function, and not the prescribed rapidly varying phase function.

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. Notice that this limitation does not apply if it is possible to factor out the rapid field variation that occurs on a wavelength scale (see [3D Models](#)).

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](#)
-

An Example — A Directional Coupler

Introduction

Directional couplers are used for coupling a light wave from one waveguide to another waveguide. By controlling the refractive index in the two waveguides, for instance by heating or current injection, it is possible to control the amount of coupling between the waveguides.

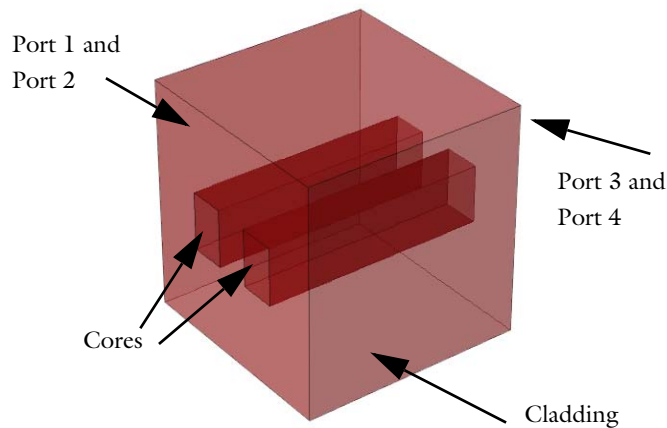


Figure 2-1: Schematic drawing of the waveguide structure. The structure consists of the two waveguide cores and the surrounding cladding. Port 1 and 2 are used for exciting the waveguides and Port 3 and 4 absorb the waves.

Light that propagates through a dielectric waveguide has most of the power concentrated within the central core of the waveguide. Outside the waveguide core, in the cladding, the electric field decays exponentially with the distance from the core. However, if you put another waveguide core close to the first waveguide (see [Figure 2-1](#)), that second waveguide perturbs the mode of the first waveguide (and vice versa). Thus, instead of having two modes with the same effective index, one localized in the first waveguide and the second mode in the second waveguide, the modes and their respective effective indexes split and you get a symmetric supermode (see [Figure 2-2](#) and [Figure 2-4](#) below), with an effective index that is slightly larger than

the effective index of the unperturbed waveguide mode, and an antisymmetric supermode (see [Figure 2-3](#) and [Figure 2-5](#)), with an effective index that is slightly lower than the effective index of the unperturbed waveguide mode.

Since the supermodes are the solution to the wave equation, if you excite one of them, it propagates unperturbed through the waveguide. However, if you excite both the symmetric and the antisymmetric mode, that have different propagation constants, there is a beating between these two waves. Thus, the power fluctuates back and forth between the two waveguides, as the waves propagate through the waveguide structure. You can adjust the length of the waveguide structure to get coupling from one waveguide to the other waveguide. By adjusting the phase difference between the fields of the two supermodes, you can decide which waveguide that initially is to be excited.

Model Definition

The directional coupler, as shown in [Figure 2-1](#), consists of two waveguide cores embedded in a cladding material. The cladding material is GaAs, with ion-implanted GaAs for the waveguide cores. The structure is modeled after [Ref. 1](#).

The core cross-section is square, with a side length of 3 μm . The two waveguides are separated 3 μm . The length of the waveguide structure is 2 mm. Thus, given the tiny cross-section, compared to the length, it is advantageous to use a view that don't preserve the aspect ratio for the geometry.

For this kind of problem, where the propagation length is much longer than the wavelength, [The Electromagnetic Waves, Beam Envelopes User Interface](#) is particularly suitable, as the mesh does not need to resolve the wave on a wavelength scale, but rather the beating between the two waves.

The model is setup to factor out the fast phase variation that occurs in synchronism with the first mode. Mathematically, we write the total electric field as the sum of the electric fields of the two modes,

$$\begin{aligned}\mathbf{E}(\mathbf{r}) &= \mathbf{E}_1 \exp(-j\beta_1 x) + \mathbf{E}_2 \exp(-j\beta_2 x) \\ &= [\mathbf{E}_1 + \mathbf{E}_2 \exp(-j(\beta_2 - \beta_1)x)] \exp(-j\beta_1 x)\end{aligned}$$

The expression within the square parentheses is what is solved for. It has a beat length L defined by

$$(\beta_2 - \beta_1)L = 2\pi$$

or

$$L = \frac{2\pi}{\beta_2 - \beta_1}$$

In the simulation, this beat length must be well resolved. Since the waveguide length is half of the beat length and the waveguide length is discretized into 20 subdivisions, the beat length is very well resolved in the model.

The model uses two numeric ports per input and exit boundary (see [Figure 2-1](#)). The two ports define the lowest symmetric and antisymmetric modes of the waveguide structure.

Results and Discussion

[Figure 2-2](#) to [Figure 2-5](#) shows the results of the initial boundary mode analysis. The first two modes (those with the largest effective mode index) are both symmetric. [Figure 2-2](#) shows the first mode. This mode has the transverse polarization component along the z -direction. The second mode, shown in [Figure 2-4](#), has transverse

polarization along the y-direction.

Effective mode index=3.471672 Surface: Boundary mode electric field, z component (V/m)

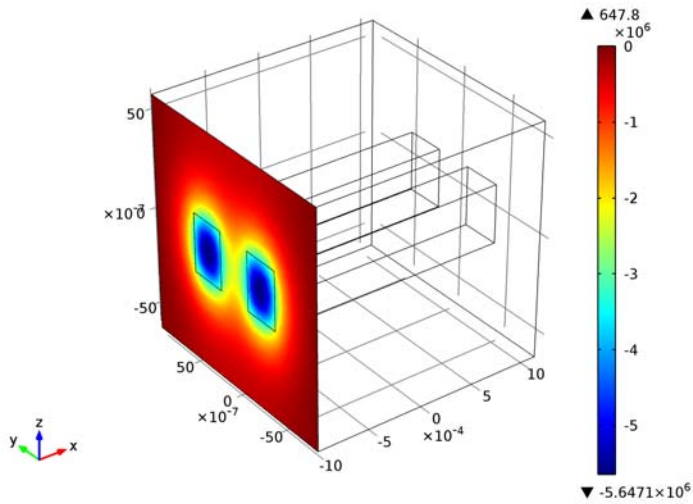


Figure 2-2: The symmetric mode for z-polarization. Notice that the returned solution can also show the electric field as positive values in the peaks at the cores.

Effective mode index=3.471422 Surface: Boundary mode electric field, z component (V/m)

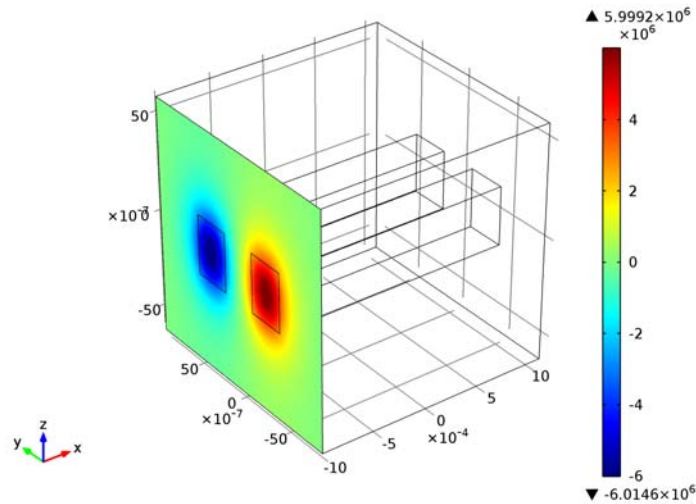


Figure 2-3: The antisymmetric mode for z-polarization.

Figure 2-3 and Figure 2-5 show the antisymmetric modes. Those have effective indexes that are slightly smaller than those of the symmetric modes. Figure 2-3 shows

the mode for z -polarization and Figure 2-5 shows the mode for y -polarization.

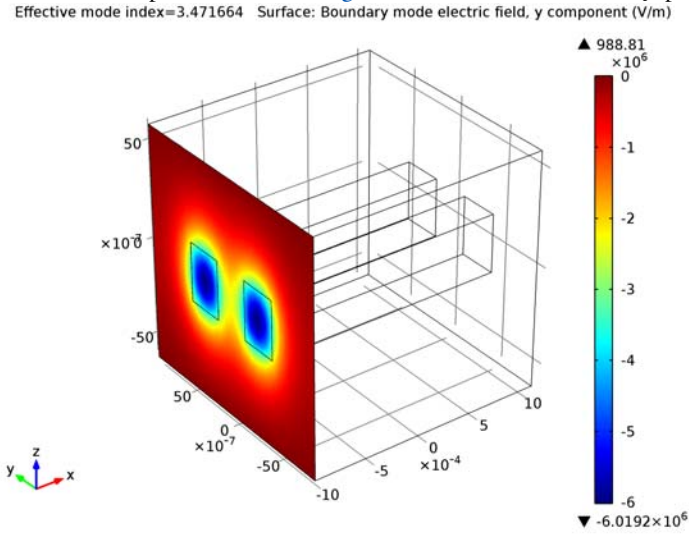


Figure 2-4: The symmetric mode for y -polarization. Notice that the returned solution can also show the electric field as positive values in the peaks at the cores.

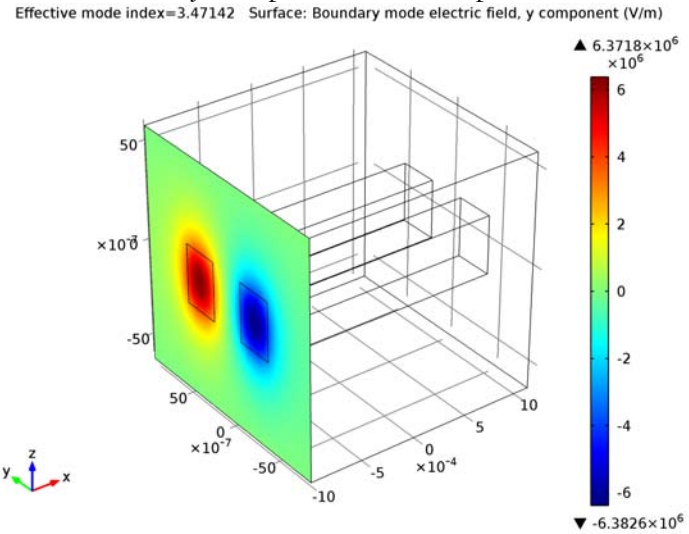


Figure 2-5: The antisymmetric mode for y -polarization.

Figure 2-6 shows how the electric field increases in the receiving waveguide and decreases in the exciting waveguide. If the waveguide had been longer, the waves would switch back and forth between the waveguides.

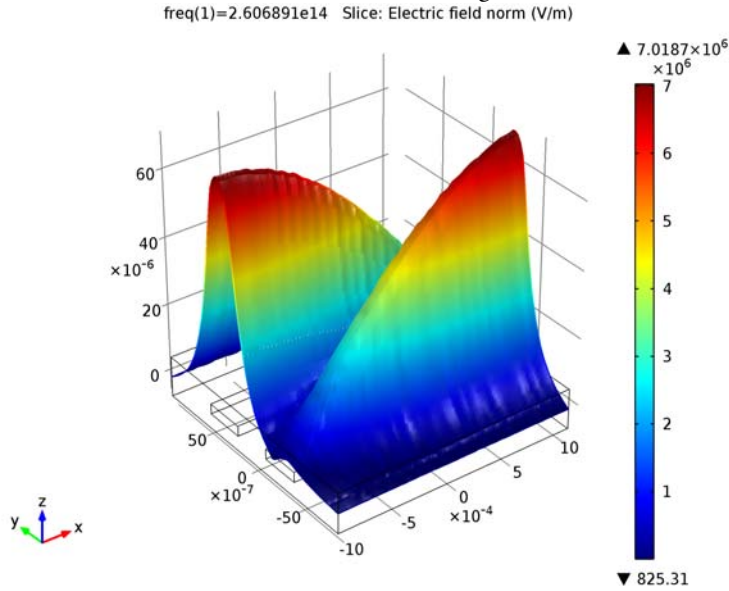


Figure 2-6: Excitation of the symmetric and the antisymmetric modes. The wave couples from the input waveguide to the output waveguide. Notice your result may show that the wave is excited in the other waveguide core, if your mode fields have different signs than what is displayed in Figure 2-2 to Figure 2-5.

Figure 2-7 show the result, when there is a π phase difference between the fields of the exciting ports. In this case, the superposition of the two modes results in excitation of the other waveguides (as compared to the case in Figure 2-6).

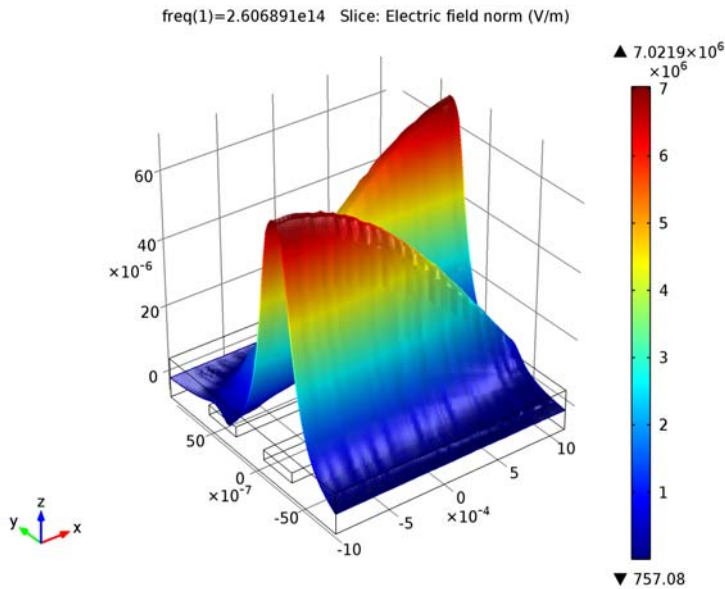


Figure 2-7: The same excitation conditions as in Figure 2-6, except that there is a phase difference between the two ports of π radians.

Reference

1. S. Somekh, E. Garmire, A. Yariv, H.L. Garvin, and R.G. Hunsperger, “Channel Optical Waveguides and Directional Couplers in GaAs-Imbedded and Ridged,” Applied Optics, vol. 13, no. 2, 1974, pp. 327–30.



Model Library path: **Wave_Optics_Module/Waveguides_and_Couplers/directional_coupler**

Modeling Instructions

MODEL WIZARD

- 1 Go to the **Model Wizard** window.
- 2 Click **Next**.

- 3 In the **Add physics** tree, select **Optics>Wave Optics>Electromagnetic Waves, Beam Envelopes (ewbe)**.
- 4 Click **Next**.
- 5 Find the **Studies** subsection. In the tree, select **Preset Studies>Boundary Mode Analysis**.
- 6 Click **Finish**.

GLOBAL DEFINITIONS

First, define a set of parameters for creating the geometry and defining the material parameters.

Parameters

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
- 2 In the **Parameters** settings window, locate the **Parameters** section.
- 3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
wl	1.15[um]	Wavelength
f0	c_const/wl	Frequency
a	3[um]	Side of waveguide cross-section
d	3[um]	Distance between the waveguides
len	2.1[mm]	Waveguide length
width	6*a	Width of calculation domain
height	4*a	Height of calculation domain
nc1	3.47	Refractive index of GaAs
dn	0.005	Refractive index increase in waveguide core
nco	nc1+dn	Refractive index in waveguide core

GEOMETRY I

Create the calculation domain.

Block 1

- 1 In the **Model Builder** window, under **Model I** right-click **Geometry I** and choose **Block**.
- 2 In the **Block** settings window, locate the **Size and Shape** section.
- 3 In the **Width** edit field, type len.
- 4 In the **Depth** edit field, type width.
- 5 In the **Height** edit field, type height.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

Now add the first embedded waveguide.

Block 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type 1e .

4 In the **Depth** edit field, type a .

5 In the **Height** edit field, type a .

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **y** edit field, type $-d$.

Add the second waveguide, by duplicating the first waveguide and modifying the position.

Block 3

1 Right-click **Model 1>Geometry 1>Block 2** and choose **Duplicate**.

2 In the **Block** settings window, locate the **Position** section.

3 In the **y** edit field, type d .

4 Click the **Build All** button.

DEFINITIONS

Since the geometry is so long and narrow, don't preserve the aspect ratio in the view.

1 In the **Model Builder** window, expand the **Model 1>Definitions** node.

Camera

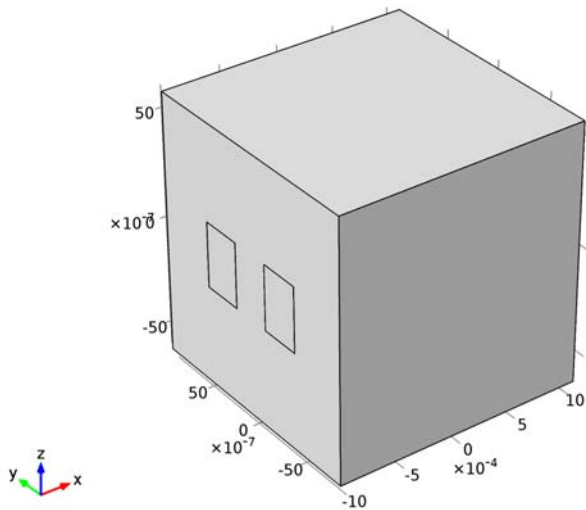
1 In the **Model Builder** window, expand the **Model 1>Definitions>View 1** node, then click **Camera**.

2 In the **Camera** settings window, locate the **Camera** section.

3 Clear the **Preserve aspect ratio** check box.

4 Click the **Go to View 1** button on the Graphics toolbar.

5 Click the **Zoom Extents** button on the Graphics toolbar.



MATERIALS

Now, add materials for the cladding and the core of the waveguides.

Material 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.
- 2 In the **Material** settings window, locate the **Material Contents** section.
- 3 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Refractive index	n	nc1

- 4 Right-click **Model 1**>**Materials**>**Material 1** and choose **Rename**.
- 5 Go to the **Rename Material** dialog box and type GaAs cladding in the **New name** edit field.
- 6 Click **OK**.

Material 2

- 1 Right-click **Materials** and choose **Material**.
- 2 Select Domains 2 and 3 only.

- 3 In the **Material** settings window, locate the **Material Contents** section.
- 4 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Refractive index	n	nc0

- 5 Right-click **Model 1 > Materials > Material 2** and choose **Rename**.
- 6 Go to the **Rename Material** dialog box and type **Implanted GaAs core** in the **New name** edit field.
- 7 Click **OK**.

ELECTROMAGNETIC WAVES, BEAM ENVELOPES

Since there will be no reflected waves in this model, it is best to select unidirectional propagation.

- 1 In the **Electromagnetic Waves, Beam Envelopes** settings window, locate the **Wave Vectors** section.
- 2 From the **Number of directions** list, choose **Unidirectional**.
- 3 In the **Model Builder** window, click **Electromagnetic Waves, Beam Envelopes**.
- 4 In the **Electromagnetic Waves, Beam Envelopes** settings window, locate the **Wave Vectors** section.
- 5 In the **\mathbf{k}_1** table, enter the following settings:

ewbe.beta_1	x
0	y
0	z

This sets the wave vector to be that of the lowest waveguide mode.

Add two numeric ports per port boundary. The first two ports excite the waveguides.

Port 1

- 1 Right-click **Electromagnetic Waves, Beam Envelopes** and choose **Port**.
- 2 Select Boundaries 1, 5, and 10 only.
- 3 In the **Port** settings window, locate the **Port Properties** section.
- 4 From the **Type of port** list, choose **Numeric**.
- 5 From the **Wave excitation at this port** list, choose **On**.

Now duplicate the first port and rename it.

Port 2

- 1 Right-click **Model 1>Electromagnetic Waves, Beam Envelopes>Port 1** and choose **Duplicate**.
- 2 In the **Port** settings window, locate the **Port Properties** section.
- 3 In the **Port name** edit field, type 2.

Next create the ports at the other end of the waveguides.

Port 3

- 1 In the **Model Builder** window, right-click **Electromagnetic Waves, Beam Envelopes** and choose **Port**.
- 2 Select Boundaries 16–18 only.
- 3 In the **Port** settings window, locate the **Port Properties** section.
- 4 From the **Type of port** list, choose **Numeric**.

Duplicate this port and give it a new unique name.

Port 4

- 1 Right-click **Model 1>Electromagnetic Waves, Beam Envelopes>Port 3** and choose **Duplicate**.
- 2 In the **Port** settings window, locate the **Port Properties** section.
- 3 In the **Port name** edit field, type 4.

MESH 1

Define a triangular mesh on the input boundary and then sweep that mesh along the waveguides.

Free Triangular 1

- 1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Free Triangular**.
- 2 Select Boundaries 1, 5, and 10 only.

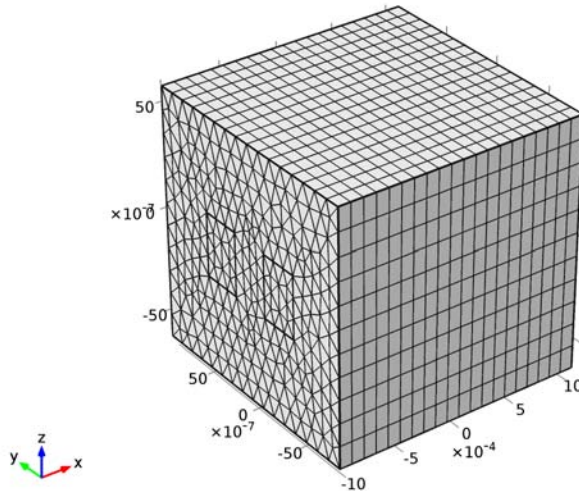
Size 1

- 1 Right-click **Model 1>Mesh 1>Free Triangular 1** and choose **Size**.
Set the maximum mesh element size to be one wavelength, which will be enough to resolve the modes.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.

- 4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 5 In the associated edit field, type w_1 .
- 6 Select the **Minimum element size** check box.
- 7 In the associated edit field, type $w_1/2$.
Sweep the mesh along the waveguides. Twenty elements along the waveguide will be sufficient to resolve the mode-coupling that will occur.
- 8 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Size

- 1 In the **Model Builder** window, under **Model 1 > Mesh 1** click **Size**.
- 2 In the **Size** settings window, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type $1en/20$.
- 5 Click the **Build All** button.



STUDY 1

Don't generate the default plots.

- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Study** settings window, locate the **Study Settings** section.
- 3 Clear the **Generate default plots** check box.

Step 1: Boundary Mode Analysis

Now analyze the four lowest modes. The first two modes will be symmetric. Since the waveguide cross-section is square, there will be one mode polarized in the z -direction and one mode polarized in the y -direction. Mode three and four will be antisymmetric, one polarized in the z -direction and the other in the y -direction.

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Boundary Mode Analysis**.
- 2 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 3 In the **Desired number of modes** edit field, type 4.
Search for the modes with effective index close to that of the waveguide cores.
- 4 In the **Search for modes around** edit field, type n_{co} .
- 5 In the **Mode analysis frequency** edit field, type f_0 .
Compute only the boundary mode analysis step.
- 6 Right-click **Study 1**>**Step 1: Boundary Mode Analysis** and choose **Compute Selected Step**.

RESULTS

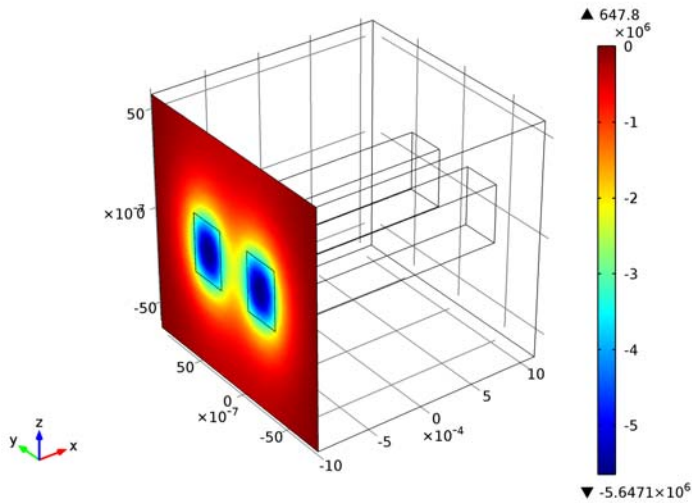
Create a 3D surface plot to view the different modes.

3D Plot Group 1

- 1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.
- 2 Right-click **3D Plot Group 1** and choose **Surface**.
First look at the modes polarized in the z -direction.
- 3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Beam Envelopes**>**Boundary mode analysis**>**Boundary mode electric field**>**Boundary mode electric field, z component (ewbe.tEbmlz)**.
- 4 In the **Model Builder** window, click **3D Plot Group 1**.
- 5 In the **3D Plot Group** settings window, locate the **Data** section.
- 6 From the **Effective mode index** list, choose the largest effective index.

- 7 Click the **Plot** button. This plot (same as [Figure 2-2](#)) shows the symmetric mode polarized in the z -direction. Notice that you might get positive electric field values in the peaks located at the waveguide cores.

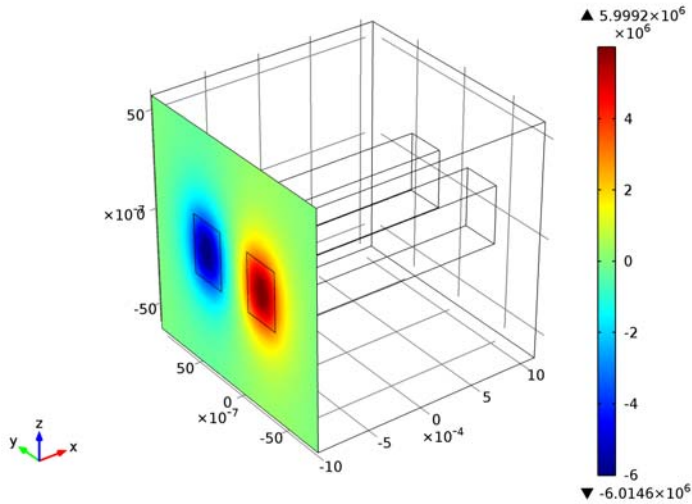
Effective mode index=3.471672 Surface: Boundary mode electric field, z component (V/m)



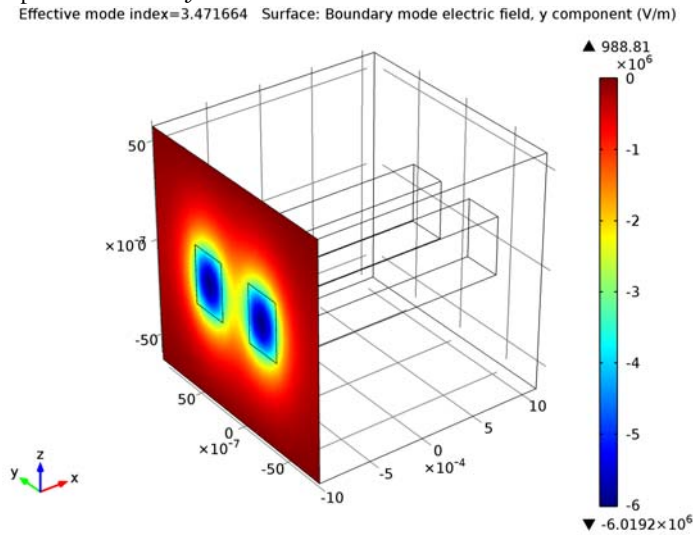
- 8 From the **Effective mode index** list, choose the third largest effective index.

- 9 Click the **Plot** button. This plot (same as [Figure 2-3](#)) shows the anti-symmetric mode polarized in the z -direction.

Effective mode index=3.471422 Surface: Boundary mode electric field, z component (V/m)

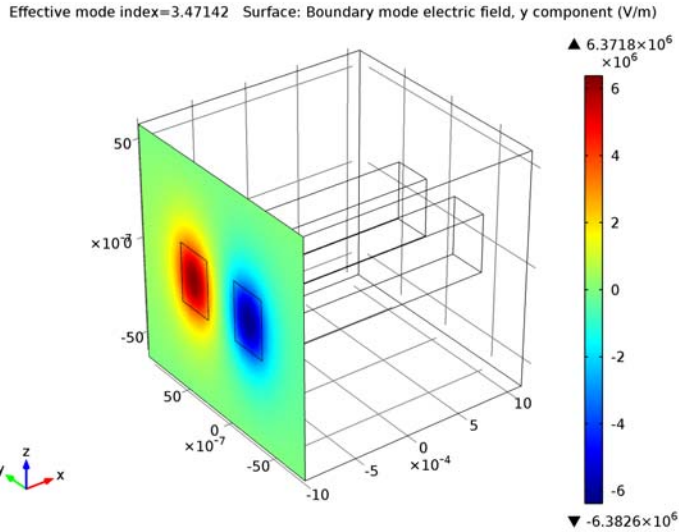


- 10 In the **Model Builder** window, under **Results>3D Plot Group 1** click **Surface 1**.
- 11 In the **Surface** settings window, locate the **Expression** section.
- 12 In the **Expression** edit field, type $ewbe.tEb_{m1y}$.
- 13 In the **Model Builder** window, click **3D Plot Group 1**.
- 14 In the **3D Plot Group** settings window, locate the **Data** section.
- 15 From the **Effective mode index** list, choose the second largest effective index.
- 16 Click the **Plot** button. This plot (same as [Figure 2-4](#)) shows the symmetric mode polarized in the y -direction.



- 17 From the **Effective mode index** list, choose the smallest effective index.

- 18 Click the **Plot** button. This plot (same as Figure 2-5) shows the anti-symmetric mode polarized in the y-direction.



Derived Values

You will need to copy the effective indexes for the different modes and use them in the boundary mode analyses for the different ports.

- 1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.
- 2 In the **Global Evaluation** settings window, locate the **Expression** section.
- 3 In the **Expression** edit field, type `ewbe.beta_1`.
- 4 Click the **Evaluate** button.
Copy all information in the table to the clipboard. Then paste that information in to your favorite text editor, so you easily can enter the values later in the boundary mode analysis steps.
- 5 In the **Table** window, click **Full Precision**.
- 6 In the **Table** window, click **Copy Table and Headers to Clipboard**. Table 2-1 lists the effective mode indices for the different modes.

TABLE 2-1: THE EFFECTIVE INDICES FOR THE MODES

POLARIZATION	MODE TYPE	EFFECTIVE MODE INDEX
z	Symmetric	3.4716717443092047
y	Symmetric	3.471663542438744

TABLE 2-1: THE EFFECTIVE INDICES FOR THE MODES

POLARIZATION	MODE TYPE	EFFECTIVE MODE INDEX
z	Antisymmetric	3.4714219480792177
y	Antisymmetric	3.471420178631897

STUDY 1

Step 1: Boundary Mode Analysis

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Boundary Mode Analysis**.
- 2 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 3 In the **Desired number of modes** edit field, type 1.
- 4 In the **Search for modes around** edit field, type 3.4716717443092047, by selecting the value in your text editor and then copying and pasting it here. This should be the largest effective index. The last figures could be different from what is written here.

Step 3: Boundary Mode Analysis 1

- 1 Right-click **Study 1 > Step 1: Boundary Mode Analysis** and choose **Duplicate**.
- 2 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 3 In the **Search for modes around** edit field, type 3.4714219480792177, by selecting the value in your text editor and then copying and pasting it here. This should be the third largest effective index. The last figures could be different from what is written here.
- 4 In the **Port name** edit field, type 2.

Step 4: Boundary Mode Analysis 2

- 1 Select the two boundary mode analyses, **Step 1: Boundary Mode Analysis** and **Step 3: Boundary Mode Analysis 3**.
- 2 In the **Model Builder** window, right-click **Step 1: Boundary Mode Analysis** and choose **Duplicate**.
- 3 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 4 In the **Port name** edit field, type 3.

Step 5: Boundary Mode Analysis 3

- 1 In the **Model Builder** window, under **Study 1** click **Step 5: Boundary Mode Analysis 3**.
- 2 In the **Boundary Mode Analysis** settings window, locate the **Study Settings** section.
- 3 In the **Port name** edit field, type 4.

Step 2: Frequency Domain

- 1 In the **Model Builder** window, under **Study 1** click **Step 2: Frequency Domain**.
- 2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
- 3 In the **Frequencies** edit field, type f_0 .
Finally, move **Step2: Frequency Domain** to be the last study step.
- 4 Right-click **Study 1>Step 2: Frequency Domain** and choose **Move Down**.
- 5 Right-click **Study 1>Step 2: Frequency Domain** and choose **Move Down**.
- 6 Right-click **Study 1>Step 2: Frequency Domain** and choose **Move Down**.
- 7 Right-click **Study 1** and choose **Compute**.

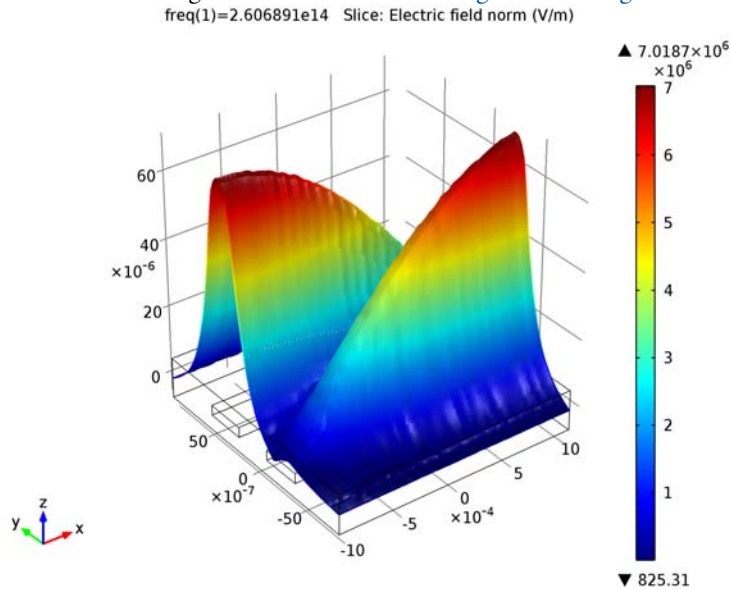
RESULTS

3D Plot Group 1

Remove the surface plot and replace it with a slice plot of the norm of the electric field.

- 1 In the **Model Builder** window, under **Results>3D Plot Group 1** right-click **Surface 1** and choose **Delete**. Click **Yes** to confirm.
- 2 Right-click **3D Plot Group 1** and choose **Slice**.
- 3 In the **Slice** settings window, locate the **Plane Data** section.
- 4 From the **Plane** list, choose **xy-planes**.
- 5 In the **Planes** edit field, type 1.
- 6 Right-click **Results>3D Plot Group 1>Slice 1** and choose **Deformation**.
- 7 In the **Deformation** settings window, locate the **Expression** section.
- 8 In the **z component** edit field, type $ewbe.normE$.
- 9 Click the **Plot** button.
- 10 Click the **Go to View 1** button on the Graphics toolbar.
- 11 Click the **Zoom Extents** button on the Graphics toolbar. The plot (same as [Figure 2-6](#)) shows how the light couples from the excited waveguide to the unexcited one. Notice that your graph might show that the other waveguide is excited than what is shown below. This can happen if your solution returned modes

with different signs for the mode fields in Figure 2-2 to Figure 2-5.



ELECTROMAGNETIC WAVES, BEAM ENVELOPES

Port 2

To excite the other waveguide, set the phase difference between the exciting ports to π .

- 1 In the **Model Builder** window, under **Model 1 > Electromagnetic Waves, Beam Envelopes** click **Port 2**.
- 2 In the **Port** settings window, locate the **Port Properties** section.
- 3 In the Θ_{in} edit field, type π .

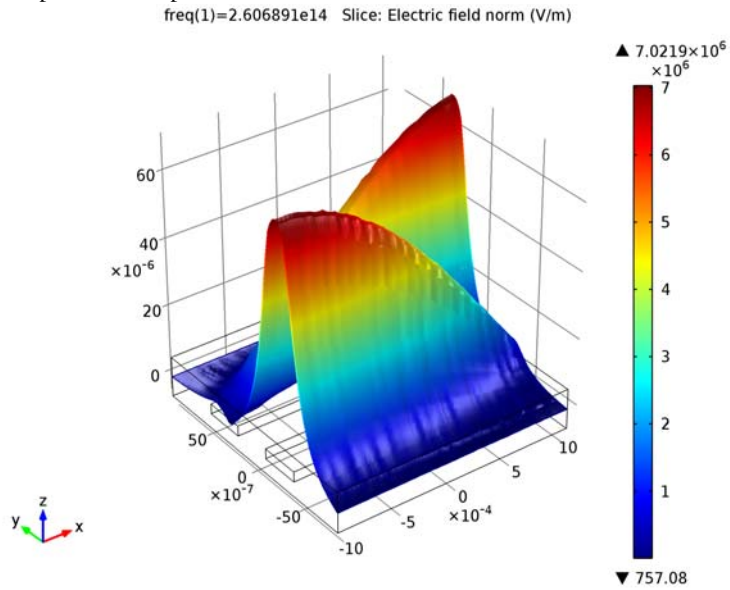
STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

3D Plot Group 1

Now the other waveguide is excited and the coupling occurs in reverse direction, compared to the previous case.



Periodic Boundary Conditions

The Wave Optics 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 **Wave_Optics_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.

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](#)



[Optical Scattering Off of a Gold Nanosphere](#): Model Library path
`Wave_Optics_Module/Optical_Scattering/scattering_nanosphere`

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 E_{farx} , E_{fary} , and E_{farz} . If, on the other hand, the geometry is axisymmetric with the independent variables r , ϕ , and z , the generated variables get the names $E_{far\phi}$, $E_{far\phi}$, and E_{farz} . 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 $ewfd.E_{farz}$ 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

$$E_{farx}(dx, dy, dz)$$

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

$$E_{farx}(\sin(\theta) \cdot \cos(\phi), \sin(\theta) \cdot \sin(\phi), \cos(\theta))$$

where the variables θ and ϕ 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 $normE_{far}$ and $normdB_{far}$, 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, $normE_{far}$ and $normdB_{far}$, or components of the far field variable E_{far}). 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.



3D model example with a Polar Plot Group [Optical Scattering Off of a Gold Nanosphere](#): Model Library path **Wave_Optics_Module/Optical_Scattering/scattering_nanosphere**.



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

Maxwell's Equations

In this section:

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

Introduction to Maxwell's Equations

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

- Electric field intensity \mathbf{E}
- Electric displacement or electric flux density \mathbf{D}
- Magnetic field intensity \mathbf{H}
- Magnetic flux density \mathbf{B}
- Current density \mathbf{J}
- Electric charge density ρ

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

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

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

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

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

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

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

GENERALIZED CONSTITUTIVE RELATIONS

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

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

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

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

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

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

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

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

Potentials

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

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

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

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

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

Electromagnetic Energy

The electric and magnetic energies are defined as

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Material Properties

Until now, there has only been a formal introduction of the constitutive relations. These seemingly simple relations can be quite complicated at times. There are four main groups of materials where they require some 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 Wave OpticsModule 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.

Boundary and Interface Conditions

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

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

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

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

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

INTERFACE BETWEEN A DIELECTRIC AND A PERFECT CONDUCTOR

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

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

Phasors

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

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

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

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

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

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

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



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

For example, all plot functions visualize

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

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

Special Calculations

In this section:

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

S-Parameter Calculations

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

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

and ϵ is the permittivity. The power flow then becomes

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

TEM Waves

For TEM waves it holds that

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

where Z_{TEM} is the wave impedance

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

The power flow then becomes

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

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

Far-Field Calculations Theory

The far 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:

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

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

References

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

S-Parameters and Ports

In this section:

- [S-Parameters in Terms of Electric Field](#)
- [S-Parameter Calculations: Ports](#)
- [S-Parameter Variables](#)
- [Port Sweeps and Touchstone Export](#)

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

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

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

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



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

S-Parameter Calculations: Ports

The Optics 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.



See [Port](#) for instructions to set up a model.

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 S_{11} and S_{21} . S_{11} is the S-parameter for the reflected wave and S_{21} is the S-parameter for the transmitted wave. For convenience, two variables for the S-parameters on a dB scale, $S_{11\text{dB}}$ and $S_{21\text{dB}}$, are also defined using the following relation:

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

The model and physics interface names also appear in front of the variable names so they 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, Frequency Domain interface cycles through the ports, computes the entire S-matrix and exports it to a Touchstone file.

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](#)

Eigenfrequency Analysis

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

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

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

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

VARIABLES AFFECTED BY EIGENFREQUENCY ANALYSIS

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

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

NONLINEAR EIGENFREQUENCY PROBLEMS

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

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

These situations 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

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

where $(-\lambda) = -\delta + j\omega$. When the solver starts to solve the eigenfrequency problem it linearizes the entire formulation with respect to the eigenvalue around a certain linearization point. By default this linearization point is 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 Wave Optics 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

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

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

VARIABLES INFLUENCED BY MODE ANALYSIS

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

NAME	EXPRESSION	CAN BE COMPLEX	DESCRIPTION
beta	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



For an example of Boundary Mode Analysis, see the model [Directional Coupler](#): Model Library path **Wave_Optics_Module/Waveguides_and_Couplers/directional_coupler**.



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

Electromagnetic Quantities

Table 2-2 shows the symbol and SI unit for most of the physical quantities that are included with this module.


TABLE 2-2: 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 2-2: 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



The Optics Branch

This chapter reviews the physics interfaces in the Wave Optics Module, which are under the **Optics** branch () in the **Model Wizard**.

In this chapter:

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

The Electromagnetic Waves, Frequency Domain User Interface

The **Electromagnetic Waves, Frequency Domain (ewfd)** interface () found under the **Wave Optics** 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 `ewfd`.

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 50Ω .

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](#)

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

- External Current Density
- Far-Field Calculation
- Far-Field Domain
- Initial Values
- 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
- 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

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

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

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

where c_0 is the speed of light in vacuum.

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

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

The wave equation is thereby rewritten as

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

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

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

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

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

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

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

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

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

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

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**, **Refractive index** (the default), **Loss tangent**, **Dielectric loss**, **Drude-Lorentz dispersion model**, **Debye dispersion model**, or **Sellmeier dispersion model**.

Relative Permittivity

When **Relative permittivity** is selected, the default **Relative permittivity** ϵ_r 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.

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

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

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

When **Drude-Lorentz dispersion model** is selected, the default **Relative permittivity, high frequency** ϵ_∞ (dimensionless) takes its value **From material**. 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

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

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

When **Debye dispersion model** is selected, the default **Relative permittivity, high frequency** ϵ_{∞} (dimensionless) takes its value **From material**. 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)**.

Sellmeier Dispersion Model

The **Sellmeier dispersion model** is often used for characterizing the refractive index of optical glasses. The model is given by

$$n^2(\lambda) = 1 + \sum_k \frac{B_k \lambda^2}{\lambda^2 - C_k}$$

where the coefficients B_k and C_k determine the dispersion properties.

When **Sellmeier dispersion model** is selected, in the table, enter values or expressions in the columns for **B** and **C (m²)**.

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 **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: $\Omega\cdot\text{m}$) are taken **From material**. Select **User defined** to enter other values or expressions for any of these variables.



For an example using the **Drude-Lorentz dispersion model**, see [Nanorods](#): Model Library path **Wave_Optics_Module/Optical_Scattering/nanorods**.

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** 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/m²).

Far-Field Domain

To set up a far-field calculation, add a **Far-Field Domain** node and specify the far-field domains in its settings window. Use **Far-Field Calculation** subnodes (one is added by default) to specify all other settings needed to define the far-field calculation. 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.



Modeling with Far-Field Calculations

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).

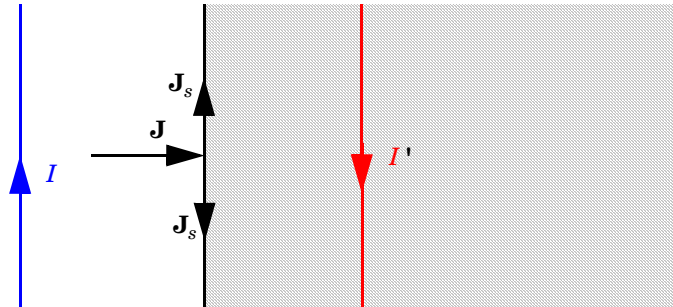
Perfect Electric Conductor

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](#)

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.

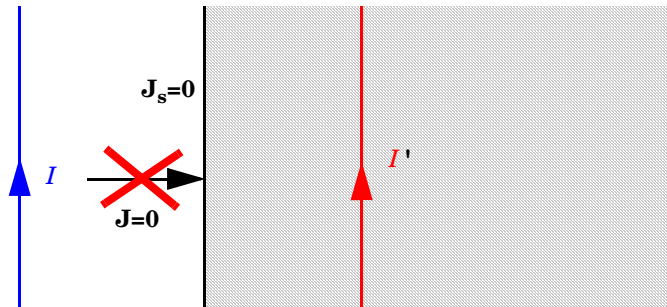


Figure 3-1: The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut. The shaded (high impedance) region is not part of the model but nevertheless carries effective mirror images of the sources. Note also that 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.



Optical Scattering Off of a Gold Nanosphere: Model Library path
Wave_Optics_Module/Optical_Scattering/scattering_nanosphere

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 TE₁₀ mode, or 11 for a TM₁₁ 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 TE₁₁ mode, or 01 for a TM₀₁ 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

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

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

where \mathbf{k} is the wave vector, $\mathbf{k}_{\text{parallel}}$ is the projection of \mathbf{k} onto the port, \mathbf{k}_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](#)



[Plasmonic Wire Grating](#): Model Library path

Wave_Optics_Module/Gratings_and_Metamaterials/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_m (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

Wave_Optics_Module/Gratings_and_Metamaterials/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

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

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

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

POINT SELECTION

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

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](#) 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

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

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

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

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.

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

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

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

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}_{dir} (dimensionless).

- If **Cylindrical wave** is selected, also enter coordinates for the **Source point** \mathbf{r}_0 (SI unit: m) and **Source axis direction** \mathbf{r}_{axis} (dimensionless).
- If **Spherical wave** is selected, enter coordinates for the **Source point** \mathbf{r}_0 (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.

Impedance Boundary Condition

The Impedance Boundary Condition

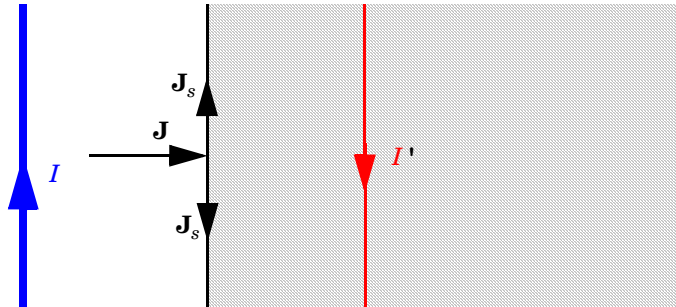
$$\sqrt{\frac{\mu_0 \mu_r}{\epsilon_c}} \mathbf{n} \times \mathbf{H} + \mathbf{E} - (\mathbf{n} \cdot \mathbf{E}) \mathbf{n} = (\mathbf{n} \cdot \mathbf{E}_s) \mathbf{n} - \mathbf{E}_s$$

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

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

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

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



The 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).

Surface Current

The **Surface Current** boundary condition

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

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

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:

$$\begin{aligned}\mathbf{J}_{s1} &= \frac{(Z_S \mathbf{E}_{t1} - Z_T \mathbf{E}_{t2})}{Z_S^2 - Z_T^2} \\ \mathbf{J}_{s2} &= \frac{(Z_S \mathbf{E}_{t2} - Z_T \mathbf{E}_{t1})}{Z_S^2 - Z_T^2} \\ Z_S &= \frac{-j\omega\mu}{k} \frac{1}{\tan(kd)} \\ Z_T &= \frac{-j\omega\mu}{k} \frac{1}{\sin(kd)} \\ k &= \omega\sqrt{(\epsilon + (\sigma/(j\omega)))\mu}\end{aligned}$$

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).

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
Wave_Optics_Module/Verification_Models/fresnel_equations
- [Plasmonic Wire Grating](#): Model Library path
Wave_Optics_Module/Gratings_and_Metamaterials/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 $I \cdot a$.

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: m^2A).
- If **Dipole moment** is selected, enter coordinates for the **Magnetic dipole moment \mathbf{m}** (SI unit: m^2A).

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).



Fabry-Perot Cavity: Model Library path

Wave_Optics_Module/Verification_Models/fabry_perot

The Electromagnetic Waves, Transient User Interface

The **Electromagnetic Waves, Transient (ewt)** user interface () , found under the **Wave Optics** 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 `ewt`.

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](#)
-

*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):

- [Magnetic Field](#)
- [Perfect Electric Conductor](#)
- [Perfect Magnetic Conductor](#)
- [Periodic Condition](#)
- [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 \(Out-of-Plane\) \(2D and 2D axisymmetric models\)](#)
- [Magnetic Point Dipole \(2D and 3D models\)](#)
- [Perfect Electric Conductor](#)
- [Perfect Magnetic 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.

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**, **Refractive index** (the default), **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.

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: C/m²).

Remanent Electric Displacement

If **Remanent electric displacement** is selected, enter coordinates for the **Remanent electric displacement \mathbf{D}_r** (SI unit: C/m²). Then select **User defined** or **From Material** as above for the **Relative permittivity ϵ_r** .

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.

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}_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.

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 Electromagnetic Waves, Time Explicit User Interface

The **Electromagnetic Waves, Time Explicit (teew)** user interface () , found under the **Wave Optics** 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 `teew`.

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 **E** and the magnetic field **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 E** and for the **Magnetic field vector 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

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

with the constitutive relations $\mathbf{B} = \mu_0 \mu_r \mathbf{H}$ and $\mathbf{D} = \epsilon_0 \epsilon_r \mathbf{E}$, which reads

$$\begin{aligned}\epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} + \sigma \mathbf{E} &= \mathbf{0} \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \times \mathbf{E} &= \mathbf{0}\end{aligned}$$

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:

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

where

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

and N_p is the basis function and i_m the polynomial order for coefficient m . α (default value: 36), η_c (default value: 1), and s (default value: 3) are the filter parameters that you specify in the corresponding text fields. The damping is derived from 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, $-\log(\epsilon)$, which is approximately 36. η_c should be between 0 and 1, where $\eta_c = 0$ means maximum filtering, and $\eta_c = 1$ means no filtering, even if filtering is active.

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

$$\epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} + \sigma \mathbf{E} = -\mathbf{J}_e$$

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/m²).

Magnetic Current Density

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

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

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/m²).

Electric Field

The **Electric Field** boundary condition

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

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

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).

The Electromagnetic Waves, Beam Envelopes User Interface

The **Electromagnetic Waves, Beam Envelopes (ewbe)** user interface () found under the **Wave Optics** branch () in the **Model Wizard**, solves the electric field based time-harmonic wave equation. The electric field is defined as the product of a slowly varying envelope function and a rapidly varying phase function. The phase function has a prescribed variation, so it is only the envelope function that is solved for. It is advantageous to use this interface for problems where the envelope function has a spatial variation on length scales much longer than the wavelength.

The physics interface supports the study types Frequency domain, Eigenfrequency, 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, Beam Envelopes, Perfect Electric Conductor, and Initial Values**.

Right-click the **Electromagnetic Waves, Beam Envelopes** node to add other features that implement, for example, boundary conditions. The following sections provide information about all feature nodes in the 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 `ewbe`.

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**—**Three-component vector** (the default), **Out-of-plane vector**, or **In-plane vector**. Select:

- **Three-component vector** to solve using a full three-component vector for the electric field envelope(s) \mathbf{E}_1 (and \mathbf{E}_2).
- **Out-of-plane vector** to solve for the electric field envelope 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 envelope vector components in the modeling plane assuming that there is no electric field perpendicular to the plane.

WAVE VECTORS

Select the **Number of directions**—**Bidirectional** (the default) or **Unidirectional**.

In the tables, enter values or expressions for the **Wave vector, first wave** \mathbf{k}_1 (SI unit: rad/m) and, if **Bidirectional** is selected, for **Wave vector, second wave** \mathbf{k}_2 (SI unit: rad/m).

When **Unidirectional** is selected, the electric field is expressed as

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r}) \exp(-j\mathbf{k}_1 \cdot \mathbf{r}),$$

where \mathbf{E}_1 is the electric field envelope that is solved for and $\exp(-j\mathbf{k}_1 \cdot \mathbf{r})$ is the prescribed rapidly varying phase function.

The solution for the electric field envelope \mathbf{E}_1 is as exact as the solution for the total electric field \mathbf{E} , as is done for the [The Electromagnetic Waves, Frequency Domain User Interface](#). The advantage is that the mesh only need to resolve the spatial variation of the field envelope \mathbf{E}_1 and not the rapid variation of the phase factor. On the other hand, for problems involving reflections and scattering there is a rapid spatial variation also for the field envelope. Then there is a no advantage of using the **Unidirectional** formulation.

When **Bidirectional** is selected, the electric field is expressed as

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r})\exp(-j\mathbf{k}_1 \cdot \mathbf{r}) + \mathbf{E}_2(\mathbf{r})\exp(-j\mathbf{k}_2 \cdot \mathbf{r}),$$

where \mathbf{E}_2 and $\exp(-j\mathbf{k}_2 \cdot \mathbf{r})$ are the electric field envelope and the prescribed phase function for the second wave.

The **Bidirectional** formulation is good to use when there are boundaries reflecting the wave in another direction than that of the incident wave. The direction for the reflected beam is typically in the opposite direction to the incident beam. The boundary conditions at these internal and/or external boundaries couple the electric field envelopes \mathbf{E}_1 and \mathbf{E}_2 .

Notice, however, that there is no coupling between \mathbf{E}_1 and \mathbf{E}_2 within domains, unless weak expressions are explicitly added to the domains in the Model Builder. For more information about how to add weak domain expressions, see [Show More Physics Options](#).

	For 2D and 3D, the default value for \mathbf{k}_1 represents a wave vector pointing in the x -direction.
	The default value for \mathbf{k}_2 represents the wave vector for a plane wave reflected from a plane normal to the x -direction. Thus, the x -component is negated, whereas the other components are the same as for wave vector of the incident wave.
	For 2D axisymmetry, the default value for \mathbf{k}_1 represents a wave vector pointing in the z -direction, whereas \mathbf{k}_2 points in the opposite direction to \mathbf{k}_1 .

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 **Parameter format (value pairs)**—**Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

Select an option from the **If file exists** list—**Overwrite** (the default) or **Create new**.

Enter a **Reference impedance**, **Touchstone file export**. The default is 50 ohm.

DISCRETIZATION

To display this section, click the **Show** button () and select **Discretization**. Select **Linear**, **Quadratic** (the default), or **Cubic** for the **Electric field envelope, first wave** and **Electric field envelope, second wave**. The second wave is applicable if the Wave Vectors are bidirectional. 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 envelope, first wave E1** and its components (in the **Electric field envelope components, first wave** fields).
- **Electric field envelope, second wave E2** and its components (in the **Electric field envelope components, second wave** fields). The second wave is applicable if the Wave Vectors are bidirectional.

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, and Point Nodes for the Electromagnetic Waves, Beam Envelopes Interface](#)
-

Domain, Boundary, Edge, and Point Nodes for the Electromagnetic Waves, Beam Envelopes Interface

The [Electromagnetic Waves, Beam Envelopes User Interface](#) has these domain, boundary, edge, point, and pair nodes and subnodes available.

DOMAIN

- Initial Values
- Wave Equation, Electric

BOUNDARY CONDITIONS

With no surface currents present, the following boundary conditions for the electric and magnetic fields need to be fulfilled

$$\begin{aligned}\mathbf{n}_{II} \times (\mathbf{E}_I - \mathbf{E}_{II}) &= \mathbf{0} \\ \mathbf{n}_{II} \times (\mathbf{H}_I - \mathbf{H}_{II}) &= \mathbf{0}'\end{aligned}$$

where the roman numerals denote the fields and normals on the two sides of the boundary.

For the unidirectional formulation, the electric field is given by the product of the electric field envelope \mathbf{E}_1 and the phase function (see [Wave Vectors](#)). Because \mathbf{E}_1 is being solved for and the phase function is continuous across boundaries, 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 for the unidirectional formulation

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

and is therefore also fulfilled.

For the bidirectional formulation the transverse electric and magnetic field envelopes are not necessarily continuous across boundaries. Thus, the continuity of the transverse electric and magnetic fields are enforced using weak expressions and constraints applied on the boundary.

The following boundary conditions are available for this interface:

- Diffraction Order
- Electric Field
- Magnetic Field
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Condition
- Port
- Scattering Boundary Condition
- Surface Current

Of the boundary conditions above, the following features are described for the [The Electromagnetic Waves, Frequency Domain User Interface](#):

- [Diffraction Order](#)
- [Periodic Condition](#)
- [Perfect Electric Conductor](#)
- [Port](#)
- [Perfect Magnetic Conductor](#)

Wave Equation, Beam Envelopes

The **Wave Equation, Beam Envelopes** node is the main node for the Electromagnetic Waves, Beam Envelopes interface. The electric field is factorized into the product

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r}) \exp(-j\mathbf{k}_1 \cdot \mathbf{r}),$$

for Wave Vectors set to unidirectional. Inserting this electric field formulation into the Maxwell's equations results in the following wave equation for the envelope function

$$(\nabla - j\mathbf{k}_1) \times ((\nabla - j\mathbf{k}_1) \times \mathbf{E}_1) - k^2 \mathbf{E}_1 = \mathbf{0}$$

for time-harmonic and eigenfrequency problems. The wave number k is defined by

$$k = k_0 n,$$

where n is the refractive index and the wave number of free space k_0 is defined as

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

Here c_0 is the speed of light in vacuum.

When Wave Vectors are set to bidirectional, the electric field is defined as the sum of two fields

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r}) \exp(-j\mathbf{k}_1 \cdot \mathbf{r}) + \mathbf{E}_2(\mathbf{r}) \exp(-j\mathbf{k}_2 \cdot \mathbf{r}).$$

The second field adds an additional wave equation to solve

$$(\nabla - j\mathbf{k}_2) \times (\mu_r^{-1} (\nabla - j\mathbf{k}_2) \times \mathbf{E}_2) - k^2 \mathbf{E}_2 = \mathbf{0}.$$

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

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



The settings for the Wave Equation, Beam Envelopes feature node are the same as [Wave Equation, Electric](#).

Initial Values

The **Initial Values** node adds initial values for the electric field envelopes for the first and second waves, which 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 envelope, first wave **E1**** and **Electric field envelope, second wave **E2**** (SI unit: V/m). The default values are 0 V/m. The second wave is applicable if the Wave Vectors are bidirectional.

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](#) 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, first wave \mathbf{E}_{01}** (SI unit: V/m). When Wave Vectors is set to bidirectional, also set the **Electric field, second wave \mathbf{E}_{02}** .

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.

MAGNETIC FIELD

Enter the value or expression for the components of the **Magnetic field, first wave \mathbf{H}_{01}** (SI unit: A/m). When Wave Vectors are set to bidirectional, also set the **Magnetic field, second wave \mathbf{H}_{02}** .

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

$$\begin{aligned}\mathbf{E} &= \mathbf{E}_{sc}e^{-jk(\mathbf{n} \cdot \mathbf{r})} + \mathbf{E}_0e^{-jk(\mathbf{k} \cdot \mathbf{r})} && \text{Plane scattered wave} \\ \mathbf{E} &= \mathbf{E}_{sc}\frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{\sqrt{r}} + \mathbf{E}_0e^{-jk(\mathbf{k} \cdot \mathbf{r})} && \text{Cylindrical scattered wave} \\ \mathbf{E} &= \mathbf{E}_{sc}\frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{r_s} + \mathbf{E}_0e^{-jk(\mathbf{k} \cdot \mathbf{r})} && \text{Spherical scattered wave}\end{aligned}$$

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. When Wave Vectors are set to unidirectional, the direction \mathbf{k} is provided automatically from the wave vector \mathbf{k}_1 specified for the interface. When Wave Vectors are set to bidirectional, the user selects whether the direction \mathbf{k} is provided from the wave vector for the first wave \mathbf{k}_1 or the wave vector for the second wave \mathbf{k}_2 .



The boundary is only perfectly transparent for scattered (outgoing) waves of the selected type at normal incidence to the boundary. That is, a plane wave at oblique incidence is partially reflected and so is a cylindrical wave or spherical wave unless the wave fronts are parallel to the boundary.

- For cylindrical waves, 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.
- When Wave Vectors are set to bidirectional, specify which wave the specified input field is associated with.

If the problem is solved for the eigenfrequency or 2D axisymmetric geometry, the boundary condition does not include the incident wave.

$$\begin{aligned} \mathbf{E}_{\text{sc}} &= \mathbf{E}_{\text{sc}} e^{-jk(\mathbf{n} \cdot \mathbf{r})} && \text{Plane scattered wave} \\ \mathbf{E}_{\text{sc}} &= \mathbf{E}_{\text{sc}} \frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{\sqrt{r}} && \text{Cylindrical scattered wave} \\ \mathbf{E}_{\text{sc}} &= \mathbf{E}_{\text{sc}} \frac{e^{-jk(\mathbf{n} \cdot \mathbf{r})}}{r_s} && \text{Spherical scattered wave} \end{aligned}$$

BOUNDARY SELECTION

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

SCATTERING BOUNDARY CONDITION

When Wave Vectors is set to bidirectional, you specify with **Input wave** which wave the specified input electric field is associated with.

Select **Incident field** to specify whether the input wave is specified by the electric field (**Wave given by E field**) or the magnetic field (**Wave given by H field**).

Specify the **Incident electric field** \mathbf{E}_0 (SI unit: V/m) or **Incident magnetic field** \mathbf{H}_0 (SI unit: A/m), depending on the setting of **Incident field**.

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

- If **Cylindrical wave** is selected, also enter coordinates for the **Source point** \mathbf{r}_0 (SI unit: m) and **Source axis direction** \mathbf{r}_{axis} (dimensionless).
- If **Spherical wave** is selected, enter coordinates for the **Source point** \mathbf{r}_0 (SI unit: m).

Surface Current

The **Surface Current** boundary condition

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

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

and avoids unexpected results when a current density with a component normal to the surface is given.

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, first wave** \mathbf{J}_{s01} (SI unit: A/m). When Wave Vectors are set to bidirectional, also specify the expression for the **Surface current density, second wave** \mathbf{J}_{s02} .

Theory for the Electromagnetic Waves User Interfaces

The Electromagnetic Waves, Frequency Domain User Interface, The Electromagnetic Waves, Beam Envelopes 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](#)

Introduction to the User Interface Equations

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

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

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

$$\begin{aligned}\nabla \times \mathbf{H} &= \sigma \mathbf{E} + \frac{\partial \epsilon \mathbf{E}}{\partial t} \\ \nabla \times \mathbf{E} &= -\mu \frac{\partial \mathbf{H}}{\partial t}\end{aligned}$$

Frequency Domain Equation

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

$$\begin{aligned}\mathbf{E}(x, y, z, t) &= \mathbf{E}(x, y, z)e^{j\omega t} \\ \mathbf{H}(x, y, z, t) &= \mathbf{H}(x, y, z)e^{j\omega t}\end{aligned}$$

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

$$\begin{aligned}\nabla \times (\mu^{-1} \nabla \times \mathbf{E}) - \omega^2 \epsilon_c \mathbf{E} &= \mathbf{0} \\ \nabla \times (\epsilon_c^{-1} \nabla \times \mathbf{H}) - \omega^2 \mu \mathbf{H} &= \mathbf{0}\end{aligned}$$

The first of these, based on the electric field is used in [The Electromagnetic Waves, Frequency Domain User Interface](#).

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

$$\nabla \times (\nabla \times \mathbf{E}) - k_0^2 n^2 \mathbf{E} = \mathbf{0} \quad (3-1)$$

The wave number in vacuum k_0 is defined by

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

where c_0 is the speed of light in vacuum.

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

For the [The Electromagnetic Waves, Beam Envelopes User Interface](#) the electric field is written as a product of an envelope function \mathbf{E}_1 and a rapidly varying phase factor with a prescribed wave vector \mathbf{k}_1 ,

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r}) \exp(-j\mathbf{k}_1 \cdot \mathbf{r}).$$

When inserting this expression into [Equation 3-1](#), the following wave equation for the electric field envelope \mathbf{E}_1 is obtained

$$(\nabla - j\mathbf{k}_1) \times ((\nabla - j\mathbf{k}_1) \times \mathbf{E}_1) - k_0^2 n^2 \mathbf{E}_1 = \mathbf{0}. \quad (3-2)$$

It is assumed that the envelope function \mathbf{E}_1 has a much slower spatial variation than the exponential phase factor. Thus, the mesh can be much coarser when solving

Equation 3-2 than when solving Equation 3-1. Thereby it is possible do simulation on domains that are much larger than the wavelength. Notice, however, that the assumption of a slowly varying envelope function is never implemented in Equation 3-2. Thus, the solution of Equation 3-2 is as exact as the solution of Equation 3-1.

EIGENFREQUENCY ANALYSIS

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

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

MODE ANALYSIS AND BOUNDARY MODE ANALYSIS

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

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

The spatial parameter, $\alpha = \delta_z + j\beta = -\lambda$, can have a real part and an imaginary part. The propagation constant is equal to the imaginary part, and the real part, δ_z , represents the damping along the propagation direction. When solving for all three electric field components the allowed anisotropy of the optionally complex relative permittivity and relative permeability is limited to:

$$\epsilon_{rc} = \begin{bmatrix} \epsilon_{rxx} & \epsilon_{rxy} & 0 \\ \epsilon_{ryx} & \epsilon_{ryy} & 0 \\ 0 & 0 & \epsilon_{rzz} \end{bmatrix} \quad \mu_r = \begin{bmatrix} \mu_{rxx} & \mu_{rxy} & 0 \\ \mu_{ryx} & \mu_{ryy} & 0 \\ 0 & 0 & \mu_{rzz} \end{bmatrix}$$



Limiting the electric field component solved for to the out-of-plane component for TE modes, requires that the medium is homogeneous, that is, μ and ϵ are constant. When solving for the in-plane electric field components for TM modes, μ 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

PROPAGATING WAVES IN 2D



In 2D, different polarizations can be chosen by selecting to solve for a subset of the 3D vector components. When selecting all three components, the 3D equation applies with the addition that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field.

In 2D, the electric field varies with the out-of-plane wave number k_z as (this functionality is not available for the [The Electromagnetic Waves, Beam Envelopes User Interface](#))

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

The wave equation is thereby rewritten as

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

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

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

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

and the wave equation is expressed as

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

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

In-plane Hybrid-Mode Waves

Solving for all three components in 2D is referred to as “hybrid-mode waves”. The equation is formally the same as in 3D with the addition that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field

In-plane TM Waves

The TM waves polarization has only one magnetic field component in the z direction, and the electric field lies in the modeling plane. Thus the time-harmonic fields can be obtained by solving for the in-plane electric field components only. The equation is formally the same as in 3D, the only difference being that the out-of-plane electric field component is zero everywhere and that out-of-plane spatial derivatives are evaluated for the prescribed out-of-plane wave vector dependence of the electric field.

In-plane TE Waves

As the field propagates in the modeling xy -plane a TE wave has only one 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 E_z ,

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

where

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

To be able to write the fields in this form, it is also required that ε_r , σ , and μ_r are 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 E_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 $\epsilon_{r\varphi\varphi}$ 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*,

$$\epsilon_c = \epsilon - j\frac{\sigma}{\omega}$$

The conductivity gives rise to ohmic losses in the medium.

A more general approach is to use a complex permittivity,

$$\epsilon_c = \epsilon_0(\epsilon' - j\epsilon'')$$

where ϵ' is the real part of ϵ_r , and all losses are given by ϵ'' . This dielectric loss model can be combined with a finite conductivity resulting in:

$$\epsilon_c = \epsilon_0\left(\epsilon' - j\left(\frac{\sigma}{\omega\epsilon_0} + \epsilon''\right)\right)$$

The complex permittivity may also be introduced as a loss tangent:

$$\epsilon_c = \epsilon_0\epsilon'(1 - j\tan\delta)$$



When specifying losses through a loss tangent, conductivity is not allowed as an input.

For the [The Electromagnetic Waves, Beam Envelopes User Interface](#), the refractive index is the default electric displacement field model instead of the permittivity. In materials where μ_r is 1, the relation between the complex refractive index

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

and the complex relative permittivity is

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

that is

$$\begin{aligned}\epsilon'_r &= n^2 - \kappa^2 \\ \epsilon''_r &= 2n\kappa\end{aligned}$$

The inverse relations are

$$\begin{aligned}n^2 &= \frac{1}{2}(\epsilon'_r + \sqrt{\epsilon'^2_r + \epsilon''^2_r}) \\ \kappa^2 &= \frac{1}{2}(-\epsilon'_r + \sqrt{\epsilon'^2_r + \epsilon''^2_r})\end{aligned}$$

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.

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

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 :

$$\mu_0 \sigma \frac{\partial A_z}{\partial t} + \mu_0 \epsilon_0 \frac{\partial}{\partial t} \left(\epsilon_r \frac{\partial A_z}{\partial t} \right) + \nabla \cdot (\mu_r^{-1} (\nabla A_z)) = 0$$

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

$$\mu_0 \epsilon_0 \frac{\partial}{\partial t} \left(n^2 \frac{\partial A_z}{\partial t} \right) + \nabla \cdot (\nabla A_z) = 0$$

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

Axisymmetric Hybrid-Mode Waves

Solving for all three components in 2D is referred to as “hybrid-mode waves”. The equation form is formally the same as in 3D with the addition that spatial derivatives with respect to ϕ are set to zero.

Axisymmetric TM Waves

TM waves have a magnetic field with only a ϕ component and thus an electric field and a magnetic vector potential with components in the rz -plane only. The equation is formally the same as in 3D, the only difference being that the ϕ component is zero everywhere and that spatial derivatives with respect to ϕ are set to zero.

Axisymmetric TE Waves

A TE wave has only a vector potential component in the ϕ direction, and the magnetic field lies in the modeling plane. Given these constraints, the 3D equation can be simplified to a scalar equation for A_ϕ . To write the fields in this form, it is also required that ϵ_r and μ_r are non diagonal only in the rz -plane. μ_r denotes a 2-by-2 tensor, and $\epsilon_{r\phi\phi}$ and $\sigma_{\phi\phi}$ 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 \mathbf{H} field and the \mathbf{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.

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 \mathbf{E}
- Electric displacement or electric flux density \mathbf{D}
- Magnetic field intensity \mathbf{H}
- Magnetic flux density \mathbf{B}
- Current density \mathbf{J}
- Electric charge density ρ

For general time-varying fields, the differential form of Maxwell's equations can be written as

$$\begin{aligned}\nabla \times \mathbf{H} &= \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \\ \nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t} \\ \nabla \cdot \mathbf{D} &= \rho \\ \nabla \cdot \mathbf{B} &= 0\end{aligned}\tag{3-3}$$

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

$$\begin{aligned}\mathbf{D} &= \varepsilon_0 \mathbf{E} + \mathbf{P} \\ \mathbf{B} &= \mu_0 (\mathbf{H} + \mathbf{M}) \\ \mathbf{J} &= \sigma \mathbf{E}\end{aligned}\tag{3-4}$$

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 3-3](#) (previously divided by μ_0), and insert it into the time derivative of the first row in [Equation 3-3](#)

$$-\nabla \times \left(\frac{1}{\mu_0} \nabla \times \mathbf{E} + \frac{\partial \mathbf{M}}{\partial t} \right) = \sigma \frac{\partial \mathbf{E}}{\partial t} + \varepsilon_0 \frac{\partial^2 \mathbf{E}}{\partial t^2} + \frac{\partial^2 \mathbf{P}}{\partial t^2}$$

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

$$\partial \mathbf{P} / \partial \mathbf{E} = \varepsilon_0 \chi_e \quad \text{and} \quad \mathbf{P} = \varepsilon_0 \chi_e \mathbf{E}$$

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 3-4 can be written as

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

Here, $\epsilon = \epsilon_0 \epsilon_r$ and $\mu = \mu_0 \mu_r$ are the permittivity and permeability of the material. The relative permittivity ϵ_r and the relative permeability μ_r are usually scalar properties but these can be second-rank symmetric (Hermitian) tensors for a general anisotropic material.

For general time-varying fields, Maxwell's equations in linear materials described in Equation 3-3 can be simplified to Maxwell-Ampere's law and Faraday's law:

$$\begin{aligned} \nabla \times \mathbf{H} &= \sigma \mathbf{E} + \epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} \\ \nabla \times \mathbf{E} &= -\mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} \end{aligned} \quad (3-5)$$

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

$$\nabla \times \mathbf{u} = \nabla \cdot \begin{bmatrix} 0 & u_3 & -u_2 \\ -u_3 & 0 & u_1 \\ u_2 & -u_1 & 0 \end{bmatrix} \quad (3-6)$$

where the divergence is applied on each row of the flux $\Gamma(\mathbf{u})$.

Maxwell's equations in 3D

$$\begin{aligned} \epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} - \nabla \times \mathbf{H} &= -\sigma \mathbf{E} \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \times \mathbf{E} &= \mathbf{0} \end{aligned}$$

are then accommodated to the Wave Form PDE as

$$\begin{aligned} d_E \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= \mathbf{f} \\ d_H \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= \mathbf{0} \end{aligned}$$

with the “mass” coefficients

$$d_E = \epsilon_0 \epsilon_r \quad \text{and} \quad d_H = \mu_0 \mu_r$$

the “flux” terms

$$\Gamma_E(\mathbf{H}) = - \begin{bmatrix} 0 & h_3 & -h_2 \\ -h_3 & 0 & h_1 \\ h_2 & -h_1 & 0 \end{bmatrix} \quad \text{and} \quad \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 & -e_2 \\ -e_3 & 0 & e_1 \\ e_2 & -e_1 & 0 \end{bmatrix}$$

and the “source” term $\mathbf{f} = -\sigma \mathbf{E}$.

THE LAX-FRIEDRICHS FLUX PARAMETERS



When using SI units (or other) for the electromagnetic fields and material properties, the Lax-Friedrichs Flux Parameter 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

$$\begin{aligned}\varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma \cdot \mathbf{E} \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= 0\end{aligned}\tag{3-7}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} 0 & -h_3 \\ h_3 & 0 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} e_2 & -e_1 \end{bmatrix}\tag{3-8}$$

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

$$\begin{aligned}\varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma \mathbf{E} \\ \mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= 0\end{aligned}\tag{3-9}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} -h_2 & h_1 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 \\ -e_3 & 0 \end{bmatrix}\tag{3-10}$$

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

$$\epsilon_0 \epsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) = -\sigma \mathbf{E}$$

$$\mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) = \mathbf{0}$$

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 $\mathbf{n} \cdot \Gamma_E(\mathbf{H}) = \mathbf{0}$.
- 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 $\mathbf{n} \times \mathbf{E} = Z\mathbf{H}$.

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, please 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.

magnetic dipole A small circular loop carrying a current. The magnetic dipole moment is $\mathbf{m} = IA\mathbf{e}$, where I is the current carried by the loop, A its area, and \mathbf{e} a unit vector along the central axis of the loop.

Maxwell's equations A set of equations, written in differential or integral form, stating the relationships between the fundamental electromagnetic quantities.

Nedelec's edge element See *vector element*.

perfect electric conductor (PEC) A material with high electrical conductivity, modeled as a boundary where the electric field is zero.

perfect magnetic conductor A material with high permeability, modeled as a boundary where the magnetic field is zero.

phasor A complex function of space representing a sinusoidally varying quantity.

quasi-static approximation The electromagnetic fields are assumed to vary slowly, so that the retardation effects can be neglected. This approximation is valid when the geometry under study is considerably smaller than the wavelength.

surface current density Current density defined on the surface. The component normal to the surface is zero. The unit is A/m.

vector element A finite element often used for electromagnetic vector fields. The tangential component of the vector field at the mesh edges is used as a degree of freedom. Also called *Nedelec's edge element* or just *edge element*.

I n d e x

- 2D
 - wave equations 73
- 2D axisymmetry
 - wave equations 73
- 2D modeling techniques 15–16
- 3D modeling techniques 16
- A**
 - advanced settings 7
 - anisotropic materials 50
 - antiperiodicity, periodic boundaries and 41
 - applying electromagnetic sources 18
 - axisymmetric models 16
 - axisymmetric waves theory 133, 137
- B**
 - Bloch-Floquet periodicity 95
 - boundary conditions
 - nonlinear eigenfrequency problems and 62
 - perfect electric conductor 80
 - perfect magnetic conductor 81
 - periodic 41
 - theory 52
 - using efficiently 17
 - boundary nodes
 - electromagnetic waves, beam envelopes 121
 - electromagnetic waves, frequency domain interface 70
 - electromagnetic waves, time explicit 109
 - electromagnetic waves, transient 102
 - boundary selection 8
- C**
 - calculating
 - S-parameters 60
 - Cartesian coordinates 15
 - circular port reference axis (node) 86
 - complex permittivity, electric losses and 134
 - complex relative permeability, magnetic losses and 135
 - consistent stabilization settings 8
 - constitutive relations 140
 - constitutive relations, theory 47–48
 - constraint settings 8
 - continuity, periodic boundaries and 41
 - coordinate system selection 8
 - curl-curl formulation 140
 - cylindrical coordinates 16
 - cylindrical waves 91, 126
- D**
 - Debye dispersion model 76
 - dielectric medium theory 52
 - diffraction order (node) 87
 - discretization 7
 - dispersive materials 50
 - documentation 9
 - domain nodes
 - electromagnetic waves, beam envelopes 121
 - electromagnetic waves, frequency domain interface 70
 - electromagnetic waves, time explicit 109
 - domain selection 8
 - Drude-Lorentz dispersion model 75
- E**
 - E (PMC) symmetry 43
 - edge current (node) 97
 - edge nodes
 - electromagnetic waves, beam envelopes 121
 - edge selection 8
 - eigenfrequency analysis 61

- eigenfrequency calculations theory 138
 - eigenfrequency study 131
 - eigenvalue (node) 62
 - electric current density (node) 112
 - electric field (node) 89, 113, 124
 - electric losses theory 134
 - electric point dipole (node) 97
 - electric scalar potential 48
 - electric susceptibility 141
 - electrical conductivity 47
 - electromagnetic energy theory 49
 - electromagnetic quantities 65
 - electromagnetic sources, applying 18
 - electromagnetic waves, beam envelopes
 - interface 118
 - electromagnetic waves, frequency domain interface 68
 - theory 129
 - electromagnetic waves, time explicit interface 107
 - theory 139
 - electromagnetic waves, transient interface 100
 - theory 129
 - emailing COMSOL 11
 - equation view 7
 - expanding sections 7
 - exponential filter, for wave problems 110
 - external current density (node) 77
- F**
- far field variables 44
 - Faraday's law 139
 - far-field calculation (node) 78
 - far-field calculations 43, 57
 - far-field domain (node) 78
 - far-field variables 43
 - file, Touchstone 70, 121
 - Floquet periodicity 41, 95
 - flux/source (node) 117
- free-space variables 72, 123
 - frequency domain equation 129
- G**
- Gauss' law 139
 - geometric entity selection 8
- H**
- H (PEC) symmetry 43
 - hide button 7
 - hybrid-mode waves
 - in-plane 133, 136–137
 - perpendicular 131
- I**
- impedance boundary condition (node) 92
 - inconsistent stabilization settings 8
 - inhomogeneous materials 50
 - initial values (node)
 - electromagnetic waves, beam envelopes 124
 - electromagnetic waves, frequency domain interface 77
 - electromagnetic waves, transient 106
 - in-plane TE waves theory 133, 136
 - in-plane TM waves 133
 - inports 83
 - Internet resources 9
- K**
- knowledge base, COMSOL 11
- L**
- line current (out-of-plane) (node) 98
 - linearization point 62
 - listener ports 83
 - losses, electric 134
 - losses, magnetic 135
 - low-reflecting boundary (node) 116
- M**
- magnetic current (node) 96
 - magnetic current density (node) 113
 - magnetic field (node) 90, 114, 125
 - magnetic losses theory 135
 - magnetic point dipole (node) 98
 - magnetic scalar potential 49

- magnetic susceptibility 48, 141
- material properties 50–51
- Maxwell's equations 46
- Maxwell-Ampere's law 139
- mesh resolution 18
- mode analysis 63, 131
- Model Library 10
- Model Library examples
 - diffraction order 88
 - Drude-Lorentz dispersion model 77
 - far-field calculations 43, 45
 - line current (out-of-plane) 99
 - perfect magnetic conductor 82
 - periodic boundary condition 96
 - periodic boundary conditions 41
 - port 86
- modeling tips 14
- MPH-files 10
- N** nonlinear materials 50
- numeric modes 83
- O** override and contribution 7–8
- P** pair nodes
 - electromagnetic waves, beam envelopes 121
 - pair selection 9
- PEC. see perfect electric conductor
- perfect conductors theory 52
- perfect electric conductor (node) 114
 - boundaries 79
- perfect magnetic conductor (node) 81, 115
- periodic boundary conditions 41
- periodic condition (node) 95
- periodic port reference point (node) 88
- permeability
 - anisotropic 131
- permeability of vacuum 47
- permittivity
 - anisotropic 131
- permittivity of vacuum 47
- phasors theory 52
- PMC. see perfect magnetic conductor
- point nodes
 - electromagnetic waves, beam envelopes 121
- point selection 8
- polarization, 2D and 2D axisymmetry 16
- port (node) 82
- port boundary conditions 60
- potentials theory 48
- Poynting's theorem 49
- Q** quality factor (Q-factor) 61, 131
- R** refractive index 73
- refractive index theory 134
- relative electric field 42
- relative permeability 48
- relative permittivity 48
- S** scattered fields, definition 42
- scattering boundary condition (node) 90, 126
- scattering parameters. see S-parameters
- selecting
 - mesh resolution 18
 - solver sequences 19
 - study types 3, 5
- Sellmeier dispersion model 76
- settings windows 7
- show button 7
- SI units 65
- skin effect, meshes and 18
- solver sequences, selecting 19
- space dimensions 4, 15
- S-parameter calculations
 - electric field and 59

- port node and 82
 - theory 54
 - spherical waves 91, 126
 - stabilization settings 8
 - study types 3
 - boundary mode analysis 83
 - eigenfrequency 61, 131
 - frequency domain 129
 - mode analysis 63, 131
 - surface charge density 52
 - surface current (node) 93, 127
 - surface current density (node) 115
 - symbols for electromagnetic quantities 65
 - symmetry in E (PMC) or H (PEC) 43
 - symmetry planes, far-field calculations 43
 - symmetry, axial 16
- T**
- TE axisymmetric waves theory 133, 137
 - TE waves theory 56
 - technical support, COMSOL 11
 - TEM waves theory 56
 - theory
 - constitutive relations 47–48
 - dielectrics and perfect conductors 52
 - electric and magnetic potentials 48
 - electromagnetic energy 49
 - electromagnetic waves 129
 - electromagnetic waves, time explicit
 - interface 139
 - far-field calculations 57
 - Maxwell equations 46
 - phasors 52
 - S-parameters 54
 - surface charges 52
 - time domain equation, theory 136
 - TM waves
 - axisymmetric 129
 - TM waves theory 56
- Touchstone file 70, 121
- transition boundary condition (node) 94
- TW axisymmetric waves theory 133, 137
- U**
- units, SI 65
 - user community, COMSOL 11
- V**
- variables
 - eigenfrequency analysis and 61
 - far-field 43
 - for far fields 44
 - mode analysis 63, 132
 - S-parameters 60
 - vector elements theory 137
- W**
- wave equation, beam envelopes (node) 123
 - wave equation, electric (node) 72, 104
 - wave equations (node) 109
 - wave excitation 83
 - wave impedance theory 56
 - wave number, free-space 72, 123
 - wavelength, meshes and 19
 - weak constraint settings 8
 - web sites, COMSOL 11