Wave Optics Module

User's Guide

Wave Optics Module User's Guide

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries 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
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM023501

Contents

Chapter 1: Introduction

About the Wave Optics Module 10

Overview of the User's Guide 20

Chapter 2: Wave Optics Modeling

Preparing for Wave Optics Modeling 25

Simplifying Geometries 26

Periodic Boundary Conditions 31

Scattered Field Formulation 33

Modeling with Far-Field Calculations 35

Chapter 3: Wave Optics Interfaces

Chapter 4: Heat Transfer Interfaces

Chapter 5: Glossary

1

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 Interface Guide

• Common Physics Interface and Feature Settings and Nodes

• Selecting the Study Type

• The Wave Optics Module Modeling Process

• Where Do I Access the Documentation and Application Libraries?

🔍   Building a COMSOL Multiphysics Model in the *COMSOL Multiphysics Reference Manual*

About the Wave Optics Module

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

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 radiation 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 quasistatic 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, and so forth. 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 Interface Guide

The physics interfaces in this module form a complete set of simulation tools for electromagnetic wave simulations. In addition to the core physics interfaces included with the COMSOL Multiphysics license, the physics interfaces below are included with the Wave Optics Module and available in the indicated space dimension. All physics 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.

In the *COMSOL Multiphysics Reference Manual*:

- Studies and Solvers
- The Physics Interfaces
- Creating a New Model
- For a list of all the core physics interfaces included with a COMSOL Multiphysics license, see Physics Interface Guide.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Fluid-Structure Interaction				
Heat Transfer				
Electromagnetic Heating				

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Laser Heating[1]		—	3D, 2D, 2D axisymmetric	frequency-stationary; frequency-transient; frequency-stationary, one-way electromagnetic heating; frequency-transient, one-way electromagnetic heating
Optics				
Wave Optics				
Electromagnetic Waves, Beam Envelopes		ewbe	3D, 2D, 2D axisymmetric	adaptive frequency sweep; boundary mode analysis; eigenfrequency; frequency domain; frequency domain, modal; wavelength domain; frequency domain source sweep
Electromagnetic Waves, Boundary Elements		ebem	3D, 2D	frequency domain; wavelength domain
Electromagnetic Waves, Frequency Domain		ewfd	3D, 2D, 2D axisymmetric	adaptive frequency sweep; boundary mode analysis; eigenfrequency; frequency domain; frequency domain, modal; mode analysis (2D and 2D axisymmetric models only); wavelength domain; frequency domain source sweep
Electromagnetic Waves, Time Explicit		teew	3D, 2D, 2D axisymmetric	time dependent; time dependent with FFT
Electromagnetic Waves, Transient		ewt	3D, 2D, 2D axisymmetric	eigenfrequency; time dependent; time dependent, modal, time dependent with FFT

[1] This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.

Common Physics Interface and Feature Settings and Nodes

There are several common settings and sections available for the physics interfaces and feature nodes. Some of these sections also have similar settings or are implemented in the same way no matter the physics interface or feature being used. There are also some physics feature nodes that display in COMSOL Multiphysics.

In each module's documentation, only unique or extra information is included; standard information and procedures are centralized in the *COMSOL Multiphysics Reference Manual*.

> In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

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 or Wavelength Domain studies, available with the Electromagnetic Waves, Frequency Domain and the Electromagnetic Waves, Beam Envelopes interfaces, are 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 or wavelength domain simulations are wave-propagation problems.

For nonlinear problems you can apply a Frequency Domain or Wavelength 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 physics 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 can 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 millimeter 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 physics 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 when adding a physics interface.

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 physics interface to account for a phenomenon not previously described in a model.

Where Do I Access the Documentation and Application Libraries?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries 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 an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules, application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes the core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to the features in the COMSOL Desktop. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (?)). This is called *topic-based* (or *context) help*.

Win	To open the **Help** window: • In the **Model Builder**, **Application Builder**, or **Physics Builder** click a node or window and then press F1. • On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1. • From the **File** menu, click **Help** (?). • In the upper-right corner of the COMSOL Desktop, click the **Help** (?) button.

Mac **Linux**	To open the **Help** window: • In the **Model Builder** or **Physics Builder** click a node or window and then press F1. • On the main toolbar, click the **Help** (?) button. • From the main menu, select **Help>Help**.

Opening the Documentation Window

Win	To open the **Documentation** window: • Press Ctrl+F1. • From the **File** menu select **Help>Documentation** (▮).

	To open the **Documentation** window:
Mac	• Press Ctrl+F1.
Linux	• On the main toolbar, click the **Documentation** (▮) button.
	• From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

Each model or application includes documentation with the theoretical background and step-by-step instructions to create a model or application. The models and applications are available in COMSOL Multiphysics as MPH files that you can open for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the model or application and its properties, including options to open it or its associated PDF document.

	The Application Libraries Window in the *COMSOL Multiphysics Reference Manual*.

Opening the Application Libraries Window
To open the **Application Libraries** window (▦):

	• From the **Home** toolbar, **Windows** menu, click (▦) **Applications Libraries**.
Win	• From the **File** menu select **Application Libraries**.
	To include the latest versions of model examples, from the **File>Help** menu, select (▦) **Update COMSOL Application Library**.

	Select **Application Libraries** from the main **File>** or **Windows>** menus.
Mac	
Linux	To include the latest versions of model examples, from the **Help** menu select (▦) **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

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

COMSOL ACCESS AND TECHNICAL SUPPORT

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 a case number are sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/support/updates
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/community
Events	www.comsol.com/events
COMSOL Application Gallery	www.comsol.com/models
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

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 Application Libraries? this information can also be searched from the COMSOL Multiphysics software **Help** menu.

TABLE OF CONTENTS, GLOSSARY, AND INDEX

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

MODELING WITH THE WAVE OPTICS MODULE

The Wave Optics Modeling chapter familiarize you with the modeling procedures. A number of models available through the Wave Optics Module Applications Libraries 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 Wave Optics Interfaces chapter describes:

- The Electromagnetic Waves, Frequency Domain Interface, which analyzes frequency domain electromagnetic waves, and uses time-harmonic and eigenfrequency or eigenmode (2D only) studies, boundary mode analysis and frequency domain, modal.
- The Electromagnetic Waves, Transient Interface, which supports the time dependent study type.
- The Electromagnetic Waves, Time Explicit Interface, which solves a transient wave equation for both the electric and magnetic fields.

- The Electromagnetic Waves, Beam Envelopes Interface, which analyzes frequency domain electromagnetic waves, and uses time-harmonic and eigenfrequency studies, boundary mode analysis, and frequency domain, modal studies.

- The Electromagnetic Waves, Boundary Elements Interface, which analyzes time-harmonic electromagnetic waves using the boundary element method.

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

HEAT TRANSFER

The Laser Heating Interface is used to model electromagnetic heating for systems and devices where the electric field amplitude varies slowly on a wavelength scale. This multiphysics interface adds an Electromagnetic Waves, Beam Envelopes interface and a Heat Transfer in Solids interface.

2

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 in the Applications Libraries also illustrate the different aspects of the simulation process.

In this chapter:

- Preparing for Wave Optics Modeling
- Simplifying Geometries
- Periodic Boundary Conditions
- Scattered Field Formulation
- Modeling with Far-Field Calculations
- Maxwell's Equations
- Special Calculations
- S-Parameters and Ports
- Jones Vectors and Polarization Plots
- Lossy Eigenvalue Calculations
- Material Libraries
- Part Libraries

- Reduced Order Modeling
- Electromagnetic Quantities

Preparing for Wave Optics Modeling

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

This section is intended to help answer questions such as:

• Which space 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.

> The Physics Interfaces and Building a COMSOL Multiphysics 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 Interface, The Electromagnetic Waves, Transient Interface, or The Electromagnetic Waves, Beam Envelopes 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 can 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 quasistatic 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.

  Another option is to truncate the model, using The Electromagnetic Waves, Boundary Elements Interface for the infinite void domain.

- 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 can be required. This is especially important for 3D problems because they can require a large amount of memory.

In the *COMSOL Multiphysics Reference Manual:*

- Meshing
- Studies and Solvers

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. This can be done automatically by enabling the **Physics-controlled mesh Sequence Type** in the settings for the mesh or by manually setting up the correct mesh sequence

---

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 Nonlocal Couplings and Coupling Operators 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 *Band-Gap Analysis of a Photonic Crystal*: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/ bandgap_photonic_crystal**.

For an example of how to use the **Physics-controlled mesh**, see *Fresnel Equations*: Application Library path **Wave_Optics_Module/ Verification_Examples/fresnel_equations**.

In the *COMSOL Multiphysics Reference Manual*:

- Periodic Condition and Destination Selection
- Periodic Boundary Conditions

Scattered Field Formulation

For problems where a known background field is illuminating an object in free space it is possible to use the scattered field formulation. Since the equation of the background field is known it can be entered as a model input and does not need to be computed. Starting from the frequency-domain governing equation:

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

The total electric field, $\mathbf{E}$, can be decomposed into two components:

$$\mathbf{E} = \mathbf{E}_{total} = \mathbf{E}_{background} + \mathbf{E}_{relative}$$

The known background field becomes a source term and the scattered field formulation thus solves for the relative electric field. A linearly polarized plane wave background field, a paraxial-approximate Gaussian beam, or a user-defined background field can be specified. When solving the scattered field formulation the total, the background, and the relative electric fields are available. The relative field is the difference between the background field and the total field. It is the relative field that contributes to the far-field calculation. For more information about the Far-Field computation, see Far-Field Calculations Theory. The benefit to this approach is that if the background field is much larger in magnitude than the scattered field, the accuracy of the simulation improves if the relative field is solved for. Another advantage is that is becomes very easy to set up a perfectly matched layer surrounding the homogeneous medium modeling domain.

The drawback to this approach is that the relative field requires some careful interpretation. The relative electric field can conceptually be decomposed into:

$$\mathbf{E}_{relative} = \mathbf{E}_{scattered} + \mathbf{E}_{correction} + \mathbf{E}_{cancellation}$$

The $\mathbf{E}_{scattered}$ component is the scattered field from object. This is the field that is of interest in a scattering problem. However, the relative field may also consist of a component that represents a correction to the background field and a cancellation of the background field. The $\mathbf{E}_{correction}$ component can be nonzero when the background field does not exactly satisfy Maxwell's equations, such as when the paraxial Gaussian beam approximation is used for a tightly focused beam. For more information about the Gaussian beam theory, see Gaussian Beams as Background Fields and Input Fields. The $\mathbf{E}_{cancellation}$ component will be nonzero and equal to $-\mathbf{E}_{background}$ wherever the total field should be zero, such as in the interior of any

perfectly shielded objects, or behind a relatively large shielding object. Note that this decomposition is conceptual only, it is only the relative field that is available.

An alternative of using the scattered-field formulation, is to use ports with the **Activate slit condition on interior port** setting enabled. Then the domain can be excited by the interior port and the outgoing field can be absorbed by perfectly matched layers. For more information about the **Port** feature and the **Activate slit condition on interior port** setting, see Port Properties.

SCATTERED FIELDS SETTING

The scattered-field formulation is available for The Electromagnetic Waves, Frequency Domain 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.

> *Optical Scattering off a Gold Nanosphere*: Application Library path
> **Wave_Optics_Module/Optical_Scattering/scattering_nanosphere**

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 Interface
- The Radiation Pattern Plots

▥ *Optical Scattering off a Gold Nanosphere*: Application Library path
Wave_Optics_Module/Optical_Scattering/scattering_nanosphere

Far-Field Support in the Electromagnetic Waves, Frequency Domain 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.

The Far-Field Domain and the Far-Field Calculation nodes get their selections automatically, if the Perfectly Matched Layer (PML) feature has been defined before adding the Far-Field Domain feature.

For each variable name entered, the software generates functions and variables, which represent the vector components of the far electric field. The names of these variables

are constructed by appending the names of the independent variables to the name entered in the field.

For example, the name `Efar` is entered and the geometry is Cartesian with the independent variables `x`, `y`, and `z`, the generated variables get the names `Efarx`, `Efary`, and `Efarz`.

If, on the other hand, the geometry is axisymmetric with the independent variables `r`, `phi`, and `z`, the generated variables get the names `Efarr`, `Efarphi`, and `Efarz`. The

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 can vary, but typically look something like `ewfd.Efarz` and so forth.

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

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

The expression

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

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

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

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

Far-Field Calculations Theory

The Radiation Pattern Plots

The **Radiation Pattern** plots are available with this module to plot the value of a global variable (for example, the far field norm, `normEfar` and `normdBEfar`, or components of the far field variable `Efar`).

The variables are plotted for a selected number of angles on a unit circle (in 2D) or a unit sphere (in 3D). The angle interval and the number of angles can be manually specified. For 2D **Radiation Pattern** plots also the reference direction from which the angle is measured and the normal to the plane the far field is computed for can be specified. For 3D **Radiation Pattern** plots you also specify an expression for the surface color.

The main advantage with the **Radiation Pattern** 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 **Radiation Pattern** plots of far-field norm are automatically added to any model that uses far field calculation features.

TABLE 2-1: VARIABLES AND OPERATORS GENERATED BY FAR FIELD

	DESCRIPTION	NAME	AVAILABLE COMPONENT
Variables	Far-field norm	normEfar	2D, 2D Axisymmetric, 3D
	Far-field norm, dB	normdBEfar	2D, 2D Axisymmetric, 3D
	Far-field variable, x component	Efarx	2D, 2D Axisymmetric, 3D
	Far-field variable, y component	Efary	2D, 2D Axisymmetric, 3D
	Far-field variable, z component	Efarz	2D, 2D Axisymmetric, 3D
	Effective isotropic radiated power	EIRP	3D
	Effective isotropic radiated power, dB	EIRPdB	3D
	Far-field gain	gainEfar	3D
	Far-field gain, dB	gaindBEfar	3D
	Axial ratio	axialRatio	3D
	Axial ratio, dB	axialRatiodB	3D
	Bistatic radar cross section	bRCS3D	3D
	Far-field variable, phi component	Efarphi	3D

TABLE 2-1: VARIABLES AND OPERATORS GENERATED BY FAR FIELD

	DESCRIPTION	NAME	AVAILABLE COMPONENT
	Far-field variable, theta component	Efartheta	3D
	Far-field realized gain	rGainEfar	3D
	Far-field realized gain, dB	rGaindBEfar	3D
	Bistatic radar cross section per unit length	bRCS2D	2D
	Maximum directivity[1]	maxD	2D Axisymmetric, 3D
	Maximum directivity, dB[1]	maxDdB	2D Axisymmetric, 3D
	Maximum gain[1]	maxGain	2D Axisymmetric, 3D
	Maximum gain, dB[1]	maxGaindB	2D Axisymmetric, 3D
	Maximum realized gain[1]	maxRGain	2D Axisymmetric, 3D
	Maximum realized gain, dB[1]	maxRGaindB	2D Axisymmetric, 3D
Functions[4]	3D far-field norm[2]	norm3DEfar	2D Axisymmetric
	3D far-field norm, dB[2]	normdB3DEfar	2D Axisymmetric
	Uniform three dimensional array factor[3]	af3	2D Axisymmetric, 3D
	Uniform two dimensional array factor[3]	af2	2D

[1]See Directivity via Global Evaluation.

[2]See 3D Far-Field Norm Functions in 2D Axisymmetry.

[3]See Array factor operators.

[4]See Far-Field Analysis Using Functions and Operators.

FAR-FIELD ANALYSIS USING FUNCTIONS AND OPERATORS

The postprocessing far-field functions are available under
Component>Definitions>Functions. Below you find example models using these
functions and some links to more information.

▥	3D example with a Polar Plot Group *Optical Scattering off a Gold Nanosphere*: Application Library path **Wave_Optics_Module/ Optical_Scattering/scattering_nanosphere**

3D Far-Field Norm Functions in 2D Axisymmetry

The functions norm3DEfar and normdB3DEfar calculate the 3D far-field norms, based on field solutions in 2D axisymmetric geometry. These functions are available in these cases:

- Far-field analysis using circular port excitation with a positive azimuthal mode number
- Scattered field analysis excited by the predefined circularly polarized plane wave type

The function can be used in a 3D **Radiation Pattern** plot, where the input argument of the function must be same as the **Azimuth angle variable** in the **Evaluation** section in the settings window.

The suffix of a function name varies based on the circular port mode type, port mode number and azimuthal mode number in the physics interface. For example, when using azimuthal mode number 1 in the physics interface and transverse electric (TE) mode with mode number 2 in the port settings, the generated operator name is norm3DEfar_TE12.

When the function is used in a radiation pattern plot under a 1D or a polar plot group, the value of input argument defines the plotting plane regardless of the normal and reference direction in the evaluation section in the settings window. For example, norm3DEfar_TE12(0) evaluates the norm of the electric far field for the TE12 mode for 0-degree azimuthal angle. This is equivalent to plotting this variable on the *xz*-plane. Similarly, norm3DEfar_TE12(pi/2) is the evaluation at 90-degree azimuthal angle, which is equivalent to plotting the variable on the *yz*-plane.

The 3D far-field norm, the linear superposition of the positive and negative azimuthal modes scaled by 0.5, is

$$\sqrt{\left|E_r \cos m\phi\right|^2 + \left|E_\phi \sin m\phi\right|^2 + \left|E_z \cos m\phi\right|^2},$$

where ϕ is the azimuthal angle.

Array factor operators

The equation for the uniform three dimensional array factor operator **af3** is

$$\frac{\sin\left(\frac{n_x}{2}(2\pi d_x \sin\theta\cos\phi + \alpha_x)\right)}{\sin\left(\frac{2\pi d_x \sin\theta\cos\phi + \alpha_x}{2}\right)} \cdot \frac{\sin\left(\frac{n_y}{2}(2\pi s d_y \sin\theta\cos\phi + \alpha_y)\right)}{\sin\left(\frac{2\pi d_y \sin\theta\cos\phi + \alpha_y}{2}\right)}$$

$$\cdot \frac{\sin\left(\frac{n_z}{2}(2\pi d_z \sin\theta\cos\phi + \alpha_z)\right)}{\sin\left(\frac{2\pi d_z \sin\theta\cos\phi + \alpha_z}{2}\right)},$$

where θ is the elevation angle and ϕ is the azimuthal angle.

The uniform two dimensional array factor operator **af2** is simpler than the three dimensional version, as the third, the z-component factor, is unity.

The number of input arguments for the array factor operators depends on the dimension of model component, 2D, 2D Axisymmetric, or 3D.

TABLE 2-2: INPUT ARGUMENTS OF ARRAY FACTOR OPERATOR

ARGUMENT	DESCRIPTION	UNIT	COMPONENT
nx	Number of elements along x-axis	Dimensionless	2D, 2D Axisymmetric, 3D
ny	Number of elements along y-axis	Dimensionless	2D, 2D Axisymmetric, 3D
nz	Number of elements along z-axis	Dimensionless	2D Axisymmetric, 3D
dx	Distance between array elements along x-axis	Wavelength	2D, 2D Axisymmetric, 3D
dy	Distance between array elements along y-axis	Wavelength	2D, 2D Axisymmetric, 3D
dz	Distance between array elements along z-axis	Wavelength	2D Axisymmetric, 3D
alphax	Phase progression along x-axis	Radian	2D, 2D Axisymmetric, 3D
alphay	Phase progression along y-axis	Radian	2D, 2D Axisymmetric, 3D
alphaz	Phase progression along z-axis	Radian	2D Axisymmetric, 3D

ANTENNA ANALYSIS USING FAR-FIELD VARIABLES

The directional properties of a radiation pattern described by variables, generated from a far-field calculation feature, help to characterize the performance of antenna devices.

Directivity from a 3D Plot

While plotting a 3D radiation pattern, the maximum directivity can be calculated by evaluating the ratio between the radiation intensity and the average value of the

radiation intensity. Since the radiation intensity is a function of power, the square of the far-field norm has to be used in the **Directivity expression** in the **Radiation Pattern** settings window for the antenna directivity calculation. For other physics interfaces, such as in the Acoustics module, the expression is different.

Directivity via Global Evaluation

The maximum directivity can be computed through **Results>Derived Values>Global Evaluation**. This calculation is based on the maximum and averaged intensity values on the far-field calculation selection. It requires the selection for the far-field calculation feature to be spherical for 3D and circular for 2D axisymmetric model components, both centered at the origin.

Gain

The antenna realized gain is defined as

$$G_{\mathrm{realized}} = \frac{4\pi U}{P_{\mathrm{in}}} = \frac{|\mathtt{normEfar}|^2}{60 P_{\mathrm{in}}}$$

where U is the radiation intensity, $\mathrm{Re}(\overline{E}_{\mathrm{far}} \times \overline{H}^*_{\mathrm{far}})/2 = |\mathtt{normEfar}|^2/240\pi$, and P_{in} is the total input power.

The antenna gain is

$$G = \frac{|\mathtt{normEfar}|^2}{60 P_{\mathrm{delivered}}}$$

where the delivered power, $P_{\mathrm{delivered}}$ is $P_{\mathrm{in}}(1 - |S_{11}|^2)$. The gain is available only when the S-parameter calculation is valid, that is, for the single port excitation case.

Maxwell's Equations

In this section:

Maxwell's Equations

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

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

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

$$
\begin{aligned}
\nabla \cdot \mathbf{D} &= \rho & &\text{Gauss' law, electric} \\
\nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t} & &\text{Faraday's law} \\
\nabla \cdot \mathbf{B} &= 0 & &\text{Gauss' law, magnetic} \\
\nabla \times \mathbf{H} &= \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} & &\text{Maxwell-Ampère's law}
\end{aligned}
\tag{2-1}
$$

Another fundamental equation, derived from Maxwell's equations, is the *equation of continuity*:

$$\nabla \cdot \mathbf{J} = -\frac{\partial \rho}{\partial t} \qquad (2\text{-}2)$$

Constitutive Relations

To obtain a closed system, the equations include *constitutive relations* that describe the macroscopic properties of the medium. They are given as

$$\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P}$$
$$\mathbf{B} = \mu_0 (\mathbf{H} + \mathbf{M}) \qquad (2\text{-}3)$$
$$\mathbf{J} = \sigma \mathbf{E}$$

where ε_0 and μ_0 are the permittivity and permeability of vacuum, and σ is the electrical conductivity. The constants ε_0 and μ_0 are available in COMSOL Multiphysics as predefined physical constants. In the SI units system, ε_0 has an approximate value of $1/(36\pi)\cdot10^{-9}$ F/m; μ_0 has an approximate value of $4\pi\cdot10^{-7}$ H/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 $\mathbf{P}$ when there is no electric field present.

The *magnetization vector* $\mathbf{M}$ similarly describes how the material is magnetized when a magnetic field $\mathbf{H}$ is present. It can be interpreted as the volume density of *magnetic dipole* moments. $\mathbf{M}$ is generally a function of $\mathbf{H}$. Permanent magnets, for instance, have a nonzero $\mathbf{M}$ 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 are:

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

The parameter ε_r is the relative permittivity and μ_r is the relative permeability of the material. Usually these are scalar properties but can, in the general case, be 3-by-3

tensors when the material is anisotropic. The properties ε and μ (without subscripts) are the permittivity and permeability of the material, respectively.

GENERALIZED CONSTITUTIVE RELATIONS

For nonlinear materials, a generalized form of the constitutive relationships is useful. The relationship used for electric fields is $\mathbf{D} = \varepsilon_0\varepsilon_r\mathbf{E} + \mathbf{D}_r$, where $\mathbf{D}_r$ 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.

For some materials, there is a nonlinear relationship between $\mathbf{B}$ and $\mathbf{H}$ such that

$$\mathbf{B} = f(|\mathbf{H}|)$$

The relation defining the current density is generalized by introducing an externally generated current $\mathbf{J}_e$. The resulting constitutive relation is $\mathbf{J} = \sigma\mathbf{E} + \mathbf{J}_e$.

Boundary Conditions

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

$$
\begin{aligned}
\mathbf{n}_2 \cdot (\mathbf{D}_1 - \mathbf{D}_2) &= \rho_s \\
\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) &= \mathbf{0} \\
\mathbf{n}_2 \cdot (\mathbf{B}_1 - \mathbf{B}_2) &= 0 \\
\mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) &= \mathbf{J}_s
\end{aligned}
\tag{2-4}
$$

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 two.
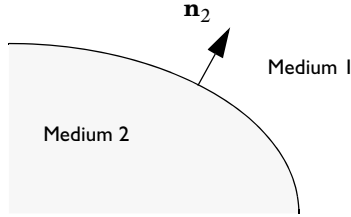
The boundary condition for the current density, derived from Equation 2-4, is expressed as

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

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. Assume that subscript 1 corresponds to a perfect conductor; then $\mathbf{D}_1 = \mathbf{0}$ and $\mathbf{E}_1 = \mathbf{0}$ in the relationships just given. If it is a time-varying case, then $\mathbf{B}_1 = \mathbf{0}$ and $\mathbf{H}_1 = \mathbf{0}$ as well, as a consequence of Maxwell's equations. The result is the following set of boundary conditions for the fields in the dielectric medium for the time-varying case:

$$\begin{aligned} -\mathbf{n}_2 \cdot \mathbf{D}_2 &= \rho_s \\ -\mathbf{n}_2 \times \mathbf{E}_2 &= 0 \\ -\mathbf{n}_2 \cdot \mathbf{B}_2 &= 0 \\ -\mathbf{n}_2 \times \mathbf{H}_2 &= \mathbf{J}_s \end{aligned} \tag{2-5}$$

Potentials

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

$$\mathbf{B} = \nabla \times \mathbf{A}$$

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

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

Electromagnetic Energy

The electric and magnetic energies are defined as

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Material Properties

Until now, there has only been a formal introduction of the constitutive relations. These seemingly simple relations can be quite complicated at times. There are four main groups of materials for which they require some consideration. A given material can belong to one or more of these groups.

INHOMOGENEOUS MATERIALS

Inhomogeneous materials are the least complicated. An inhomogeneous medium is one in which the constitutive parameters vary with the space coordinates so that different field properties prevail at different parts of the material structure.

ANISOTROPIC MATERIALS

For anisotropic materials, the field relationships at any point differ for different directions of propagation. This means that a 3-by-3 tensor is necessary to properly define the constitutive relationships. If this tensor is symmetric, the material is often referred to as *reciprocal*. In such cases, rotate the coordinate system such that a diagonal matrix results. 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). Anisotropic parameters are needed, for example, to

examine permittivity in crystals (Ref. 2) and when working with conductivity in solenoids.

NONLINEAR MATERIALS

Nonlinearity is the effect of variations in permittivity or permeability with the intensity of the electromagnetic field. Nonlinearity 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.

DISPERSIVE MATERIALS

Dispersion describes changes in a wave's velocity with wavelength. In the frequency domain, dispersion is expressed with a frequency dependence of the constitutive relations.

MATERIAL PROPERTIES AND THE MATERIAL BROWSER

All interfaces in the Wave Optics Module support the use of the COMSOL Multiphysics material libraries. The typical electromagnetic material properties that can be stored are:

- The electrical conductivity
- The relative permittivity
- The relative permeability

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 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.

For detailed information about Materials and Modeling Anisotropic Materials, including the Electromagnetic Models, see the *COMSOL Multiphysics Reference Manual*.

Phasors

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

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

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

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

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

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

If the fields satisfy a linear time-dependent equation, then the corresponding phasors must satisfy a similar equation in which the time derivatives are replaced by a factor $j\omega$. All time-harmonic equations are expressed as equations for the phasors. (The tilde is dropped from the variable denoting the phasor.)

The frequency domain formulation is only applicable for equations linear in the fields. In particular, it cannot be used with materials whose properties depend on the fields themselves (nonlinear materials).

!	When analyzing the solution of a time-harmonic equation, it is important to remember that the field that has been calculated is a phasor and not a physical field.
	For example, all plot functions visualize $\mathrm{Re}(\tilde{\mathbf{E}}(\mathbf{r}))$ by default, which is $\mathbf{E}$ at time $t = 0$. To obtain the solution at a given time, specify a phase factor in all results settings and in the corresponding functions.

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 $\mathbf{E}_1$, $\mathbf{E}_2$, $\mathbf{E}_3$, … 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 mode with subscript 1. The computed electric field $\mathbf{E}_c$ on the port consists of the excitation plus the reflected field. That is, on the port boundary where there is an incident wave, the computed field can be expanded in terms of the mode fields as

$$\mathbf{E}_c = \mathbf{E}_1 + \sum_{i=1} S_{i1}\mathbf{E}_i ,$$

whereas on all other port boundaries, the computed field is given by

$$\mathbf{E}_c = \sum_{i=1} S_{i1}\mathbf{E}_i$$

The S-parameter for the mode with index k is then given by multiplying with the conjugate of the mode field for mode k and integrating over the port boundary. Since the mode fields for the different modes are orthogonal, the following relations are obtained for the S-parameters

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

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

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

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

POWER FLOW NORMALIZATION

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

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

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

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

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

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

TE Waves
For TE waves it holds that

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

where Z_{TE} is the wave impedance

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

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

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

TM Waves
For TM waves it holds that

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

where Z_{TM} is the wave impedance

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

and ε is the permittivity. The power flow then becomes

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

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

TEM Waves
For TEM waves it holds that

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

where Z_{TEM} is the wave impedance

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

The power flow then becomes

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

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

Far-Field Calculations Theory

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

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

and in 2D it looks slightly different:

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

In both cases the integration is performed on a closed boundary. In the scattered field formulation, where the total electric field is the sum of the background field and the scattered field, the far field only gets contributions from the scattered field, since the contributions from the background field cancel out when integrated over all parts of the closed boundary.

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 toward point p.

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

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

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

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

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

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 Voltage and Electric Field
- S-Parameter Calculations
- S-Parameter Variables
- Port Sweeps/Manual Terminal Sweeps and Touchstone Export

S-Parameters in Terms of Voltage and 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/feeds, that is, there is no reflection directly at a port.

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

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

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

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

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

S-Parameter Calculations

The Optics interfaces have built-in support for S-parameter calculations. 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 S11 and S21. S11 is the S-parameter for the reflected wave and S21 is the S-parameter for the transmitted wave. For convenience, two variables for the S-parameters on a dB scale, S11dB and S21dB, are also defined using the following relation:

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

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

Port Sweeps/Manual Terminal Sweeps and Touchstone Export

The Port Sweep Settings section in the Electromagnetic Waves interface describes how to cycle through the ports, compute the entire S-matrix and export it to a Touchstone file.

The Frequency Domain Source Sweep study is another way of making efficient port sweeps. It is available as a preset study for the Electromagnetic Waves, Frequency Domain and Electromagnetic Waves, Beam Envelopes interfaces.

Exporting a Touchstone file can also be performed by right-clicking the **Export** node under **Results** and selecting **Touchstone** or by selecting **Touchstone** under **Data** in the **Results** ribbon toolbar (Windows users) or the **Results** context menu (Mac and Linux users).

Jones Vectors and Polarization Plots

In this section:

- Jones Vectors for Polarization Analysis
- Polarization Plots
- Jones Vector Variables

Jones Vectors for Polarization Analysis

Periodic ports and Diffraction order ports launch and absorb plane waves propagating in homogeneous domains (adjacent the port boundary). For a plane wave propagating with the wave vector $\mathbf{k}$, the polarization must be orthogonal to the wave vector. For each wave vector, there are two possible orthogonal polarizations. We can select one such set of orthogonal polarizations by first defining the out-of-plane polarization as the field in the direction $\mathbf{e}_1 = \mathbf{k} \times \mathbf{n}$, where $\mathbf{n}$ is the normal direction to the port. Then the in-plane polarization direction is defined as $\mathbf{e}_2 = \mathbf{e}_1 \times \mathbf{k}$.

Assuming now that $\mathbf{e}_1$ and $\mathbf{e}_2$ are normalized base vectors, the electric field can be expanded as

$$\mathbf{E} = E_1 \mathbf{e}_1 + E_2 \mathbf{e}_2,$$

where E_1 and E_2 are the elements of the Jones vector

$$\begin{pmatrix} E_1 \\ E_2 \end{pmatrix}.$$

> 📝 In COMSOL, the Jones vector elements are complex numbers and the Jones vectors are not normalized.

If the Jones vector elements have the same phase or a π phase difference, the Jones vector represents a linear polarization state. A phase difference of $\pm\pi/2$ between the

two Jones vector elements defines a circular polarization state. For other phase differences, the Jones vector represents elliptic polarization states.

🔍	For more details about the different types of periodic ports, see • Periodic • Diffraction Order • Orthogonal Polarization

Polarization Plots

Default Polarization plots are automatically generated for Periodic ports in 3D and in 2D, if the **Electric field components for** setting in the **Components** section for the physics interface is set to **Three-component vector**. The Polarization plot includes polarization ellipses for each diffraction order. The polarization ellipses are generated by plotting the in-plane Jones vector element versus the out-of-plane Jones vector element for a phase change of 2π.

❗	In the *COMSOL Multiphysics Reference Manual* you can find more information about the Polarization plot type.

▥	For an example model including a Polarization plot, see *Hexagonal Grating*: Application Library path **Wave_Optics_Module/ Gratings_and_Metamaterials/hexagonal_grating**.

Jones Vector Variables

This module automatically generates variables for the Jones vector elements. As for Polarization Plots, the variables are created for Periodic ports in 3D and in 2D, if the **Electric field components for** setting in the **Components** section for the physics interface is set to **Three-component vector**. The variables are available for postprocessing as global variables, with names based on what port boundary the variable is applicable for, the polarization direction, and the mode number. The context above is encoded in a variable name of the form `tag.JXYYY_Z`, where

• `tag` is the physics interface tag, for example `ewfd`.

• `X` is `R` if the port is located on the same side as the exciting port or otherwise `T`.

- `YYY` is `OOP` or `IP` for the out-of-plane and in-plane modes, respectively.

- `Z` is the mode number that in 2D is provided as a single signed integer and in 3D as two signed integers separated by an underscore. The signed integers use the prefix `p` for positive values and `n` for negative values.

Thus, the variable `ewfd.JROOP_0_n1` represents the Jones vector element for the out-of-plane mode in 3D with mode numbers $m = 0$ and $n = -1$ for a port located on the same boundary as the exciting Periodic port.

There are also variables for the norm of the Jones vector, named as `tag.normJX_Z`, where `X` and `Z` represents the boundary location and mode number, respectively. Thus, the variable `ewfd.normJR_0_n1` represents the Jones vector norm in 3D with mode numbers $m = 0$ and $n = -1$ for a port located on the same boundary as the exciting Periodic port.

The base vectors are available for evaluation on the port boundaries, with variable names like `tag.eJXYYY[xyz]_Z`, where `tag`, `X`, `YYY`, and `Z` have the same meanings as for the variables discussed above and `[xyz]` is one of the Cartesian components. Thus, `ewfd.eJTIPx_p1`, is the x-component of the in-plane base vector for a 2D port, with mode number $m = +1$, that is not located on the same boundary as the exciting port.

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 and Boundary 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

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

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

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

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 may require special treatment, especially since it can lead to "singular matrix" or "undefined value" messages if not treated correctly. Under normal circumstances, the automatically generated solver settings should handle the cases described in the table above. However, the following discussion provide some background to the problem of defining the eigenvalue linearization point. The complication is not only the nonlinearity itself, it is also the way it enters the equations. For example the impedance boundary conditions with nonzero boundary conductivity has the term

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

where $(-\lambda) = -\delta + j\omega$. When the solver starts to solve the eigenfrequency problem it linearizes the entire formulation with respect to the eigenvalue around a certain linearization point. By default this linearization point is set to the value provided to the **Search for eigenvalues around** field, for the three cases listed in the table above. Normally, this should be a good value for the linearization point. For instance, for the impedance boundary condition, this avoids setting the eigenvalue λ to zero in the denominator in the equation above. For other cases than those listed in the table above, the default linearization point is zero.

If the default values for the linearization point is not suitable for your particular problem, you can manually 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 can 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, if 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 searches 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 eigenvalue shifts, use the same value with the real part removed from the eigenvalue or, equivalently, use the real part of the eigenfrequency.

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.

<table>
<tr><td>@</td><td>

• For a list of the studies available by physics interface, see The Wave Optics Module Physics Interface Guide

• Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

</td></tr>
</table>

<table>
<tr><td>▥</td><td>

Band-Gap Analysis of a Photonic Crystal: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/ bandgap_photonic_crystal**

</td></tr>
</table>

Mode Analysis and Boundary Mode Analysis

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

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

The spatial parameter, $\alpha = -\lambda$, can have a real part and an imaginary part. For mode analysis the propagation constant, β, is equal to the imaginary part and the real part, δ_z, represents the damping along the propagation direction. Thus,

$$\alpha = \delta_z + j\beta = -\lambda,$$

where λ is the eigenvalue.

For boundary mode analysis, the propagation constant, β, is complex,

$$\alpha = j\beta = -\lambda.$$

VARIABLES INFLUENCED BY MODE ANALYSIS

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

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

In the table above, lambda is the eigenvalue and k0 is the vacuum wave number.

VARIABLES INFLUENCED BY BOUNDARY MODE ANALYSIS

The table below lists the variables that are influenced by the boundary-mode analysis:

NAME	EXPRESSION	CAN BE COMPLEX	DESCRIPTION
beta_i	j*lambda	Yes	Propagation constant
neff_i	beta_i/k0	Yes	Effective mode index

The name suffix indicates that the variables are defined for the port named i.

▥	For an example of Boundary Mode Analysis, see the model *Directional Coupler*: Application 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 Interface Guide • Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

Material Libraries

The Wave Optics Module features a Material Library with material properties for optical glasses, metals, polymers, gases, and other compounds to assist in wave optics simulation.

▥	For an example of how to use the Optical Material Library, see *Optical Scattering off a Gold Nanosphere*: Application Library path **Wave_Optics_Module/Optical_Scattering/scattering_nanosphere**

The Optical Material Library contains material property data from the following companies' products:

CDGM GLASS CO., LTD
H-FK61, H-FK61B, H-FK71, H-FK95N, H-QK1, H-QK3L, H-K1, H-K2, H-K3, H-K5, H-K6, H-K7, H-K8, H-K9L, H-K9LGT, H-K9L*, H-K9LA, H-K10, H-K11, H-K12, H-K50, H-K51, H-ZPK1A, H-ZPK2A, H-ZPK3, H-ZPK5, H-ZPK7, H-BaK1, H-BaK2, H-BaK3, H-BaK4, H-BaK5, H-BaK6, H-BaK7, H-BaK7GT, H-BaK7A, H-BaK8, H-ZK1, H-ZK2, H-ZK3, H-ZK3A, H-ZK4, H-ZK5, H-ZK6, H-ZK7, H-ZK7A, H-ZK8, H-ZK9B, H-ZK9A, H-ZK10, H-ZK10L, H-ZK11, H-ZK14, H-ZK20, H-ZK21, H-ZK50, H-ZK50GT, H-LaK1, H-LaK2A, H-LaK3, H-LaK4L, H-LaK5A, H-LaK6A, H-LaK7A, H-LaK8B, H-LaK8A, H-LaK10, H-LaK11, H-LaK12, H-LaK50A, H-LaK51A, H-LaK52, H-LaK53B, H-LaK53A, H-LaK54, H-LaK59A, H-LaK61, H-LaK67, H-KF6, H-QF1, H-QF3, H-QF6A, H-QF8, H-QF14, H-QF50, H-QF50A, H-QF56, H-F1, H-F2, H-F3, H-F4, H-F5, H-F13, H-F51, H-BaF2, H-BaF3, H-BaF4, H-BaF5, H-BaF6, H-BaF7, H-BaF8, H-ZBaF1, H-ZBaF3, H-ZBaF4, H-ZBaF5, H-ZBaF16, H-ZBaF20, H-ZBaF21, H-ZBaF50, H-ZBaF52, H-ZF1A, H-ZF1, H-ZF2, H-ZF3, H-ZF4A, H-ZF4AGT, H-ZF5, H-ZF6, H-ZF7LA, H-ZF7LAGT, H-ZF10, H-ZF11, H-ZF12, H-ZF13, H-ZF13GT, H-ZF39, H-ZF50, H-ZF52, H-ZF52GT, H-ZF52TT, H-ZF62, H-ZF62GT, H-ZF71, H-ZF71GT, H-ZF72A, H-ZF72AGT, H-ZF73, H-ZF73GT, H-ZF88, H-ZF88GT, H-LaF1, H-LaF2, H-LaF3B, H-LaF4, H-LaF4GT, H-LaFL5, H-LaF6LA, H-LaF7, H-LaF10LA, H-LaF50B, H-LaF51, H-LaF52, H-LaF53, H-LaF54, H-LaF55, H-LaF62, H-ZLaF1, H-ZLaF2A, H-ZLaF3, H-ZLaF4LA, H-ZLaF4LB, H-ZLaF50E, H-ZLaF50D, H-ZLaF51, H-ZLaF52A, H-ZLaF52, H-ZLaF53B, H-ZLaF53BGT, H-ZLaF55D, H-ZLaF56B, H-ZLaF66, H-ZLaF66GT, H-ZLaF68C, H-ZLaF68N, H-ZLaF69, H-ZLaF69A, H-ZLaF71,

H-ZLaF71AGT, H-ZLaF73, H-ZLaF75A, H-ZLaF75B, H-ZLaF76, H-ZLaF76A, H-ZLaF78B, H-ZLaF89L, H-ZLaF90, H-ZLaF92, H-TF3L, H-TF5, D-FK61, D-FK61A, D-FK95, D-QK3L, D-K9, D-K9GT, D-K59, D-PK3, D-ZPK1A, D-ZPK3, D-ZPK5, D-ZPK7, D-ZK2, D-ZK3, D-ZK79, D-LaK5, D-LaK6, D-LaK52, D-LaK70, D-ZF10, D-ZF93, D-LaF50, D-LaF050, D-LaF53, D-LaF79, D-ZLaF50, D-ZLaF52LA, D-ZLaF61, D-ZLaF67, D-ZLaF81, D-ZLaF85A, D-ZLaF85L, H-QK3LGTi, H-K90GTi, QF50GTi, HWS1, HWS2, HWS3, HWS4, HWS5, HWS6, HWS7, HWS9, and HWS27.

The foregoing materials are the property of CDGM GLASS CO., LTD. For product information concerning these materials, see http://www.cdgmgd.com/. For other questions concerning the materials, please contact gmgd@cdgmgd.com.

Product catalog: www.cdgmglass.com/Glass-Finder.

CORNING INCORPORATED

Corning® Gorilla® Glass 3, Corning® Gorilla® Glass 5, Corning® Gorilla® Glass 6, Antimicrobial Corning® Gorilla® Glass, Corning® Astra™ Glass, Corning® Lotus™ NXT Glass, Corning® Eagle XG® Glass, ULE® Corning Code 7972 Ultra Low Expansion Glass, Corning® Unicrown Glass, Corning® UV Clear Glass, Corning® 1.6/41 White High Index Glass, Corning® 1.7/35 White High Index Glass, Corning® 1.7/42 White High Index Glass, Corning® 1.8/35 White High Index Glass, Corning® 1.9/31 White High Index Glass, Corning® Calcium Fluoride (CaF2) - Code 9575, Corning® HPFS® 8655 Fused Silica, and Corning® HPFS® 7980 Fused Silica.

Material Data supplied by Corning Incorporated, corning.com. ASTRA, CORNING, EAGLE XG, HPFS, GORILLA, LOTUS, and ULE are trademarks or registered trademarks of Corning Incorporated.

OHARA INC.

S-FPL51, S-FPL53, S-FPL55, S-FPM 2, S-FPM 3, S-FPM 4, S-FPM 5, S-FSL 5, S-BSL 7, S-BSM 2, S-BSM10, S-BSM14, S-BSM15, S-BSM16, S-BSM18, S-BSM25, S-BSM28, S-BSM71, S-BSM81, S-NSL 3, S-NSL36, S-BAL 2, S-BAL 3, S-BAL12, S-BAL14, S-BAL35, S-BAL41, S-BAL42, S-BAM 4, S-BAM12, S-BAH10, S-BAH11, S-BAH27, S-BAH28, S-PHM52, S-PHM52Q, S-PHM53, S-TIL 1, S-TIL 2, S-TIL 6, S-TIL25, S-TIL26, S-TIL27, S-TIM 2, S-TIM 5, S-TIM 8, S-TIM22, S-TIM25, S-TIM27, S-TIM28, S-TIM35, S-TIH 1, S-TIH 3, S-TIH 4, S-TIH 6, S-TIH10, S-TIH11, S-TIH13, S-TIH14, S-TIH18, S-TIH23, S-TIH53, S-TIH53W, S-TIH57, S-LAL 7, S-LAL 7Q, S-LAL 8, S-LAL 9, S-LAL10, S-LAL12, S-LAL12Q,

S-LAL13, S-LAL14, S-LAL18, S-LAL19, S-LAL20, S-LAL21, S-LAL54, S-LAL54Q, S-LAL58, S-LAL59, S-LAL61, S-LAM 2, S-LAM 3, S-LAM 7, S-LAM54, S-LAM55, S-LAM60, S-LAM61, S-LAM66, S-LAM73, S-LAH51, S-LAH52, S-LAH52Q, S-LAH53, S-LAH53V, S-LAH55V, S-LAH55VS, S-LAH58, S-LAH59, S-LAH60, S-LAH60MQ, S-LAH60V, S-LAH63, S-LAH63Q, S-LAH64, S-LAH65V, S-LAH65VS, S-LAH66, S-LAH71, S-LAH79, S-LAH88, S-LAH89, S-LAH92, S-LAH93,S-LAH95, S-LAH96, S-LAH97, S-LAH98, S-LAH99, S-FTM16, S-NBM51, S-NBH 5, S-NBH 8, S-NBH51, S-NBH52V, S-NBH53V, S-NBH55, S-NBH56, S-NBH57, S-NBH58, S-NPH 1, S-NPH 1W, S-NPH 2, S-NPH 3, S-NPH 4, S-NPH 5, S-NPH 7, L-BSL 7, L-BAL35, L-BAL35P, L-BAL42, L-BAL42P, L-BAL43, L-TIM28, L-TIM28P, L-LAL13, L-LAL15, L-LAM60, L-LAM69, L-LAH53, L-LAH84, L-LAH85V, L-LAH90, L-LAH91, and L-LAH94.

The foregoing Material are the property of Ohara Inc. For product information concerning these Material, see www.ohara-inc.co.jp/en/product/catalog/. For other questions concerning the Material, please contact Ohara Inc. through www.ohara-inc.co.jp/en/contact/.

Product catalog: www.oharacorp.com/catalog.html.

SCHOTT AG

F2, F2HT, F5, FK5HTi, K10, K7, LAFN7, LASF35, LF5, LF5HTi, LLF1, LLF1HTi, N-BAF10, N-BAF4, N-BAF51, N-BAF52, N-BAK1, N-BAK2, N-BAK4, N-BAK4HT, N-BALF4, N-BALF5, N-BASF2, N-BASF64, N-BK10, N-BK7, N-BK7HT, N-BK7HTi, N-F2, N-FK5, N-FK51A, N-FK58, N-K5, N-KF9,N-KZFS11, N-KZFS2, N-KZFS4, N-KZFS4HT, N-KZFS5, N-KZFS8, N-LAF2, N-LAF21, N-LAF33, N-LAF34, N-LAF35, N-LAF7, N-LAK10, N-LAK12, N-LAK14, N-LAK21, N-LAK22, N-LAK33B, N-LAK34, N-LAK7, N-LAK8, N-LAK9, N-LASF31A, N-LASF40, N-LASF41, N-LASF43, N-LASF44, N-LASF45, N-LASF45HT, N-LASF46A, N-LASF46B, N-LASF9, N-LASF9HT, N-PK51, N-PK52A, N-PSK3, N-PSK53A, N-SF1, N-SF10, N-SF11, N-SF14, N-SF15, N-SF2, N-SF4, N-SF5, N-SF57, N-SF57HT, N-SF57HTultra, N-SF6, N-SF66, N-SF6HT, N-SF6HTultra, N-SF8, N-SK11, N-SK14, N-SK16, N-SK2, N-SK2HT, N-SK4, N-SK5, N-SSK2, N-SSK5, N-SSK8, N-ZK7, N-ZK7A, P-BK7, P-LAF37, P-LAK35, P-LASF47, P-LASF50, P-LASF51, P-SF68, P-SF69, P-SF8, P-SK57, P-SK57Q1, P-SK58A, P-SK60, SF1, SF10, SF11, SF2, SF4, SF5, SF56A, SF57, SF57HTultra, SF6, and SF6HT.

The foregoing materials are the property of Schott AG. For product information concerning these materials, see https://www.schott.com/advanced_optics/. For other questions concerning the materials, please contact info.optics@schott.com.

Product catalog: www.schott.com/products/optical-glass.

ORGANIC, INORGANIC, AND MISCELLANEOUS

The remaining categories of **Organic**, **Inorganic**, and **Miscellaneous** are derived from the refractive index database. Authors: Mikhail Polyanskiy. Availability: https://refractiveindex.info/. License: Public Domain through CCO 1.0 Universal.

Part Libraries

In wave optics simulations, it is often necessary to set up complex dielectric waveguide geometries. This can be conveniently accomplished using the Part Library for the Wave Optics Module.

The Part Library contains slab waveguide shapes for 2D geometries and rectangular waveguide shapes for 3D geometries, including the following parts:

- Straight waveguides
- Tapered waveguides
- Bent (ring) waveguides
- S-bend waveguides
- Couplers

The parts are built from fully parameterized sequences of geometry instructions. For example, you can load the `slab_waveguide_straight` part into a model and then specify the core and cladding widths, the element length, and the core offset. Wave Optics parts include selections for the core and the cladding domains that can be used when defining the material properties. Furthermore, there are also other selections defined for setting up ports and other boundary conditions and to simplify the definition of the mesh sequences.

The 3D rectangular waveguide structures define work planes, to easily connect different waveguide elements.

	The following Wave Optics tutorials use the Part Library to create their geometry sequences:
▥	• *Mach–Zehnder Modulator*: Application Library path **Wave_Optics_Module/Waveguides_and_Couplers/ mach_zehnder_modulator** • *Optical Ring Resonator Notch Filter*: Application Library path **Wave_Optics_Module/Waveguides_and_Couplers/optical_ring_resonator**

🔍	Part Libraries in the *COMSOL Multiphysics Reference Manual*.

Reduced Order Modeling

When designing bandpass-filter type high-Q devices in the frequency domain, it may be necessary to apply many frequency samples to describe the passband accurately. The reduced order modeling technique can help accelerate the modeling of such devices as a fine frequency resolution can be used for a modest simulation time.

Two simulation methods: the asymptotic waveform evaluation (AWE) and frequency-domain modal methods, both are designed to help overcome the conventional issue of a longer simulation time when using a very fine frequency resolution or running a very wideband simulation. The AWE is efficient when it comes to describing smooth frequency responses with a single resonance or no resonance at all. The frequency-domain modal method, meanwhile, is useful for quickly analyzing multistage filters or filters of a high number of elements that have multiple resonances in a target passband.

In this section:

• Adaptive Frequency Sweep Using Asymptotic Waveform Evaluation (AWE) Method

• Frequency Domain, Modal Method

	Adaptive Frequency Sweep and Frequency Domain, Modal in the *COMSOL Multiphysics Reference Manual*

Adaptive Frequency Sweep Using Asymptotic Waveform Evaluation (AWE) Method

The AWE method is very useful when simulating resonant circuits, especially single-resonance bandpass-filter type devices with many frequency points. When using the **Adaptive Frequency Sweep** study, the simulation time with a much finer frequency resolution can be almost the same as a coarse resolution regular **Frequency Domain** simulation.

CHOOSING APPROPRIATE ASYMPTOTIC WAVEFORM EVALUATION (AWE) EXPRESSIONS

The simulation time may vary depending on the user input for the **AWE expressions**. Any model variable works as an AWE expression, so long as it has a smooth curve shape like a Gaussian pulse as a function of frequency. The absolute value of S_{21}, `abs(comp1.ewfd.S21)`, often works as the input for the AWE expression in the case of two-port bandpass filters. For one-port devices like antennas, the absolute value of S_{11} is a good choice. If the frequency response of the AWE expression contains an infinite gradient — the case for the S_{11} value of an antenna with excellent impedance matching at a single frequency point — the simulation will take longer to complete because it requires many data points to describe the sharp dip. When the loss in a one-port device is negligible, an alternative expression such as `sqrt(1-abs(comp1.ewfd.S11)^2)` may work more efficiently than using `abs(comp1.ewfd.S11)` directly.

DATA MANAGEMENT

With a very fine frequency step simulation, the solutions contain a lot of data. As a result, the model file size will increase tremendously when it is saved. By selecting the **Store fields in output** check box in the **Values of Dependent Variables** section of the **Frequency Domain** study step settings, it is possible to define for what part of the model the computed solution should be saved. When only S-parameters are of interest, it is not necessary to store all of the field solutions. Instead, only store the field on the selections for the port boundaries, as those will be used for the S-parameter calculations.

In the **Values of Dependent Variables** section, change the selection in the **Store fields in output** combo box from **All** to **For selections** and then add the explicit selections that include the port boundaries. The explicit selection can be easily created from the port feature by clicking **Create Selection** icon in the **Boundary Selection** settings once the selection is specified.

AWE CONFIGURATION IN FREQUENCY DOMAIN STUDY STEP SETTINGS

The solver performs fast adaptive frequency sweeping using an AWE method. To trigger the AWE in a **Frequency Domain** study, the following steps are required:

• Expand **Study Extensions section** in **Frequency Domain** study step settings.

• Click the **Use asymptotic waveform evaluation** check box in the **Study Extensions** section.

• Specify the user input for the **AWE expressions**.

Frequency Domain, Modal Method

Bandpass-frequency responses of a passive circuit often result from a combination of multiple resonances. **Eigenfrequency** analysis is used for capturing the resonance frequencies of a device. In a subsequent step, the information from the **Eigenfrequency** solutions can be reused in a **Frequency Domain, Modal** study to generate a solution for the driven problem.

EIGENFREQUENCY STUDY STEP SETTINGS

To perform a **Frequency Domain, Modal** analysis, it is necessary to configure the **Eigenfrequency** study step properly. As the output of the **Eigenfrequency** study may include unphysical results (so called spurious modes), appropriate settings help refine the **Eigenfrequency** study results.

- Set **Eigenfrequency search method** to **Manual**
- Adjust **Desired number of eigenfrequencies** if necessary.
- Set **Search for eigenfrequencies around** to the estimate of the lowest passband frequency.
- Set **Eigenfrequency search method around shift** to **Larger real part**.

DATA MANAGEMENT

The **Store fields in output** check box in the **Values of Dependent Variables** section can be applied to the **Frequency Domain, Modal** study — if you are interested only in S-parameters. By storing solutions only on port boundaries, the saved model file size will decrease a lot.

	Note that the phase of the computed S-parameters in the **Frequency Domain, Modal** study can be different from that of the regular frequency sweep model due to that all relevant eigenmodes might not be included in the simulation. It is recommended to perform an initial eigenfrequency investigation, to find all relevant eigenfrequencies contributing to the frequency response. If not all relevant eigenfrequencies are included in the simulation, the results are compatible only for phase-independent S-parameter values, such as dB-scaled, absolute value, reflectivity, or transmittivity.

Electromagnetic Quantities

Table 2-3 shows the symbol and SI unit for most of the physical quantities that are included with this module.

TABLE 2-3: 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 2-3: 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

3

Wave Optics Interfaces

This chapter describes the physics interfaces found under the **Optics>Wave Optics** branch (▯▮◖) when adding a physics interface.

In this chapter:

- The Electromagnetic Waves, Frequency Domain Interface
- The Electromagnetic Waves, Transient Interface
- The Electromagnetic Waves, Time Explicit Interface
- The Electromagnetic Waves, Beam Envelopes Interface
- The Electromagnetic Waves, Boundary Elements Interface
- Theory for the Electromagnetic Waves Interfaces
- Theory for the Electromagnetic Waves, Time Explicit Interface

The Electromagnetic Waves, Frequency Domain Interface

The **Electromagnetic Waves, Frequency Domain (ewfd)** interface (⦚⩗⦚) is used to solve for time-harmonic electromagnetic field distributions. It is found under the **Wave Optics** branch (⦚◊⦚) when adding a physics interface,

For this physics interface, the maximum mesh element size should be limited to a fraction of the wavelength. The domain size that can be simulated thus scales with the amount of available computer memory and the wavelength. The physics interface supports the Frequency Domain, Wavelength Domain, Eigenfrequency, Mode Analysis, and Boundary Mode Analysis study types. The Frequency Domain and Wavelength Domain study types are used for source driven simulations for a single frequency/wavelength or a sequence of frequencies/wavelengths. The Eigenfrequency study type is used to find resonance frequencies and their associated eigenmodes in resonant cavities.

This physics interface solves the time-harmonic wave equation for the electric field.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Wave Equation, Electric**, **Perfect Electric Conductor**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Electromagnetic Waves, Frequency Domain** to select physics features from the context menu.

The Mode analysis study type is applicable only for 2D and 2D axisymmetric 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 2D, 2D axisymmetry, and 3D and applies to boundaries representing waveguide ports.

Physics-Controlled Mesh
The physics-controlled mesh is controlled from the **Settings** window for the **Mesh** node (if the **Sequence type** is **Physics-controlled mesh**). In the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the check box in the **Use** column on the same row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

When the **Use** check box for the physics interface is selected, this invokes a parameter for the maximum mesh element size in free space. The physics-controlled mesh

automatically scales the maximum mesh element size as the wavelength changes in different dielectric and magnetic regions. If the model is configured by any periodic conditions, identical meshes are generated on each pair of periodic boundaries. Perfectly matched layers are built with a structured mesh, specifically, a swept mesh in 3D and a mapped mesh in 2D.

When the **Use** check box is selected for the physics interface, in the section for the physics interface below the table, choose one of the four options for the **Maximum mesh element size control parameter** — **From study** (the default), **User defined**, **Frequency**, or **Wavelength**. When **From study** is selected, $1/5$ of the vacuum wavelength from the highest frequency defined in the study step is used for the maximum mesh element size. For the option **User defined**, enter a suitable **Maximum element size in free space**. For example, $1/5$ of the vacuum wavelength or smaller. When **Frequency** is selected, enter the highest frequency intended to be used during the simulation. The maximum mesh element size in free space is $1/5$ of the vacuum wavelength for the entered frequency. For the **Wavelength** option, enter the smallest vacuum wavelength intended to be used during the simulation. The maximum mesh element size in free space is $1/5$ of the entered wavelength.

When **Resolve wave in lossy media** is selected, the outer boundaries of lossy media domains are meshed with a maximum mesh element size in free space given by the minimum value of half a skin depth and $1/5$ of the vacuum wavelength.

The maximum mesh element size in dielectric media is equal to the maximum mesh element size in vacuum divided by the square root of the product of the relative permittivity and permeability.

!	In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

SETTINGS

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to

different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is ewfd.

FORMULATION

From the **Formulation** list, select whether to solve for the **Full field** (the default) or the **Scattered field**.

For **Scattered field** select a **Background wave type** according to the following table:

TABLE 3-1: BACKGROUND WAVE TYPE BASED ON COMPONENT DIMENSION

COMPONENT	BACKGROUND WAVE TYPE
2D	User defined (default), Gaussian beam
2D Axisymmetric	User defined (default), Circularly polarized plane wave
3D	User defined (default), Gaussian beam, Linearly polarized plane wave

User Defined

Enter the component expressions for the **Background electric field** $\mathbf{E}_b$ (SI unit: V/m). The entered expressions must be differentiable.

📄 Notice that expressions including coupling operators are not differentiable and cannot be used as background fields.

Gaussian Beam

For **Gaussian beam** select the **Gaussian beam type** — **Paraxial approximation** (the default) or **Plane wave expansion**.

When selecting **Paraxial approximation**, the Gaussian beam background field is a solution to the paraxial wave equation, which is an approximation to the Helmholtz equation solved for by the **Electromagnetic Waves, Frequency Domain (ewfd)** interface. The approximation is valid for Gaussian beams that have a beam radius that is much larger than the wavelength. Since the paraxial Gaussian beam background field is an approximation to the Helmholtz equation, for tightly focused beams, you can get a nonzero scattered field solution, even if you do not have any scatterers. The option **Plane wave expansion** means that the electric field for the Gaussian beam is approximated by an expansion of the electric field into a number of plane waves. Since each plane wave is a solution to the Helmholtz equation, the plane wave expansion of the electric field is also a solution to the Helmholtz equation. Thus, this option can be used also for tightly focused Gaussian beams.

If the beam spot radius is smaller than the wavelength, evanescent plane waves need to be included in the expansion. The evanescent waves decay exponentially in the propagation direction, why it only makes sense to model such tightly focused beams if the focal plane coincides with the input boundary. If the focal plane is located inside the modeled domain, the field can be dominated by the exponentially decaying evanescent waves. Those waves can have a very high field strength before the focal plane even though they only provide a small contribution to the field at the focal plane.

For **Plane wave expansion** select **Wave vector distribution type** — **Automatic** (the default) or **User defined**. For **Automatic** also check **Allow evanescent waves**, to include evanescent waves in the plane wave expansion. For **User defined** also enter values for the **Wave vector count** $N_{\mathbf{k}}$ (the default value is 13) and **Maximum transverse wave number** $k_{t,max}$ (SI unit: rad/m, default value is `(2*(sqrt(2*log(10))))/ewfd.w0`). Use an odd number for the **Wave vector count** $N_{\mathbf{k}}$ to make sure that a wave vector pointing in the main propagation direction is included in the plane-wave expansion. The **Wave vector count** $N_{\mathbf{k}}$ specifies the number of wave vectors that will be included per transverse dimension. So for 3D the total number of wave vectors will be $N_{\mathbf{k}} \cdot N_{\mathbf{k}}$.

    📄   Evanescent waves are included in the plane wave expansion if the **Maximum transverse wave number** $k_{t,max}$ is larger that the specified **Wave number** k. When the **Wave vector distribution type** is set to **Automatic**, evanescent waves are included in the expansion if the **Allow evanescent waves** check box is selected.

A plane wave expansion with a finite number of plane waves included will make the field periodic in the plane orthogonal to the main propagation direction. If the separation between the transverse wave vector components, given by $2k_{t,max}/(N_{\mathbf{k}} - 1)$, is too small, replicas of the Gaussian beam background field can appear. To avoid that, increase the value for the **Wave vector count** $N_{\mathbf{k}}$.

The number of plane waves included in the expansion can be quite large, especially for 3D. For instance, using the default settings, $2 \cdot 13 \cdot 13 = 338$ plane waves will be included (the factor 2 accounts for the two possible polarizations for each wave vector). Thus, initializing the plane-wave expansion for the Gaussian beam background field can take some time in 3D.

For more information about the Gaussian beam theory, see Gaussian Beams as Background Fields and Input Fields.

Define the Gaussian beam background field using the parameters below:

- Select a **Beam orientation**: **Along the x-axis** (the default), **Along the y-axis**, or for 3D components, **Along the z-axis**.

- Enter a **Beam radius** w_0 (SI unit: m). The default is $20\pi/$`ewfd.k`$_0$ m (10 vacuum wavelengths).

- Enter a **Focal plane along the axis** p_0 (SI unit: m). The default is 0 m.

- Select an **Input quantity: Electric field amplitude** (the default) or **Power**.

- Enter the component expressions for the **Transverse background electric field amplitude, Gaussian beam** $\mathbf{E}_{T\mathrm{bg}0}$ (SI unit: V/m) if the **Input quantity** is **Electric field amplitude**. Notice that this is the transverse Gaussian beam amplitude in the focal plane. When the **Gaussian beam type** is set to **Paraxial approximation** the background field is always orthogonal (transverse) to **Beam orientation**. However, when the **Gaussian beam type** is set to **Plane wave expansion**, the background field amplitude can also have a component in the propagation direction. Specify here only the field amplitude components that are orthogonal to the propagation direction. COMSOL computes automatically the component in the propagation direction, if needed.

- If the **Input quantity** is set to **Power**, enter the **Input power** (SI unit: W in 2D axisymmetry and 3D and W/m in 2D) and the component expressions for the **Non-normalized transverse electric field amplitude, Gaussian beam** $\mathbf{E}_{T\mathrm{bg}0}$ (SI unit: V/m).

- Enter a **Wave number** k (SI unit: rad/m). The default is `ewfd.k`$_0$ rad/m. The wave number must evaluate to a value that is the same for a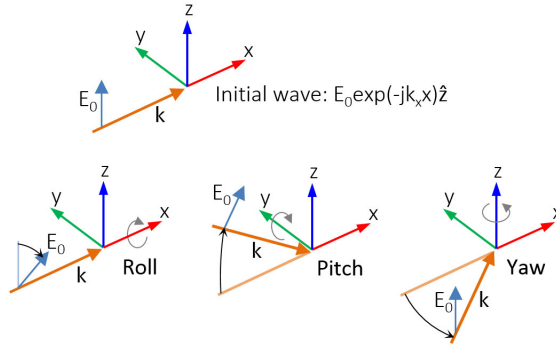ll the domains the scattered field is applied to. Setting the **Wave number** k to a positive value, means that the wave is propagating in the positive x-, y-, or z-axis direction, whereas setting the **Wave number** k to a negative value means that the wave is propagating in the negative x-, y-, or z-axis direction.

	Nanorods: Application Library path **Wave_Optics_Module/Optical_Scattering/nanorods** demonstrates how to set up the Gaussian background field, based on the plane-wave expansion.

Linearly Polarized Plane Wave

The initial background wave is predefined as $\mathbf{E}_0 = \exp(-jk_x x)\mathbf{z}$. This field is transformed by three successive rotations along the roll, pitch, and yaw angles, in that

order. For a graphic representation of the initial background field and the definition of the three rotations compare with Figure 3-1 below.

- Enter an **Electric field amplitude** E_0 (SI unit: V/m). The default is 1 V/m.

- Enter a **Roll angle** (SI unit: rad), which is a right-handed rotation with respect to the +x direction. The default is 0 rad, corresponding to polarization along the +z direction.

- Enter a **Pitch angle** (SI unit: rad), which is a right-handed rotation with respect to the +y direction. The default is 0 rad, corresponding to the initial direction of propagation pointing in the +x direction.

- Enter a **Yaw angle** (SI unit: rad), which is a right-handed rotation with respect to the +z direction.

- Enter a **Wave number** k (SI unit: rad/m). The default is ewfd.k$_0$ rad/m. The wave number must evaluate to a value that is the same for the domains the scattered field is applied to.

Figure 3-1: Schematic of the directions for the wave vector **k**, *the electric field* **E**$_0$, *and the roll, pitch and yaw rotations. The top image represents an initial wave propagating in the x direction with a polarization along the z direction.*

Circularly Polarized Plane Wave

The background wave is defined as

$$\mathbf{E}_b(r, \varphi, z) = \tilde{\mathbf{E}}_b(r, z)e^{-jm\varphi},$$

where

$$\tilde{\mathbf{E}}_b(r, z) = E_0(\hat{\mathbf{r}} - jm\hat{\varphi})e^{-jkz},$$

m is the azimuthal mode number (+1 or -1) varying depending on the **Circular polarization type** and **Direction of propagation** settings, and $\hat{\mathbf{r}}$ and $\hat{\varphi}$ are the unit vectors in the r and φ directions, respectively.

- Select the **Circular polarization type** — **Right handed** or **Left handed**.
- Select the **Direction of propagation** — **+z** or **-z**.
- Enter an **Electric field amplitude** E_0 (SI unit: V/m). The default is 1 V/m.
- Enter an **Wave number** k (SI unit: rad/m). The default is ewfd.k0 rad/m.

COMPONENTS

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

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 **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 and 2D axisymmetric components, when solving for **Three-component vector** or **In-plane vector**.

For 2D components, assign a wave vector component to the **Out-of-plane wave number** field. For 2D axisymmetric components, assign an integer constant or an integer parameter expression to the **Azimuthal mode number** field.

PORT SWEEP SETTINGS

Select the **Use manual port sweep** check box to enable the port sweep. When selected, this invokes a parametric sweep over the ports in addition to the frequency/wavelength sweep already added. The generated lumped parameters are in the form of an S-parameter matrix.

For **Use manual port sweep** enter a **Sweep parameter name** to assign a specific name to the parameter that controls the port number solved for during the sweep. Before making the port sweep, the parameter must also have been added to the list of parameters in the **Parameters** section of the **Parameters** node under the **Global**

Definitions node. This process can be automated by clicking the **Configure Sweep Settings** button. The **Configure Sweep Settings** button helps add a necessary port sweep parameter and a **Parametric Sweep** study step in the last study node. If there is already a **Parametric Sweep** study step, the sweep settings are adjusted for the port sweep.

| ! | In the *COMSOL Multiphysics Reference Manual* see the Frequency Domain Source Sweep section for a discussion of how to use the Frequency Domain Source Sweep study type to perform efficient port sweeps. |

Select **Export Touchstone file** and the S-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 pair)**: **Magnitude angle**, **Magnitude (dB) angle**, or **Real imaginary**.

Enter a **Reference impedance for Touchstone file export** Z_{ref} (SI unit: Ω) that is used only for the header in the exported Touchstone file. The default is 50 Ω.

| ! | It is also possible to export Touchstone files during postprocessing under the **Results** node. In the *COMSOL Multiphysics Reference Manual*, see the Touchstone section for more information on how to export Touchstone files as a postprocessing step. |

PORT OPTIONS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

The electric field on port boundaries is expanded as

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_{inc}(\mathbf{r}) + \sum_i S_i \mathbf{E}_i(\mathbf{r}) \exp(-\alpha_i \mathbf{n} \cdot (\mathbf{r} - \mathbf{r}_0)),$$

where $\mathbf{E}$, $\mathbf{r}$, $\mathbf{E}_{inc}$, S_i, $\mathbf{E}_i$, α_i, $\mathbf{n}$, $\mathbf{r}_0$ are, respectively, the electric field on the port boundary, the position vector, the incident electric field, the expansion coefficient (or S-parameter), the mode field, the propagation constant for the mode, the normal vector, and a position on the port boundary.

Select the **Use constraint-free ports** check box to calculate the expansion coefficients as overlap integrals

$$S_i = \frac{1}{2P_i}\int_S (\mathbf{E} - \mathbf{E}_{\text{inc}}) \times \mathbf{H}_i^\star \cdot \mathbf{n} \; dS \;\; ,$$

where P_i, S, and $\mathbf{H}_i$ are the mode power, the port boundary area, and the magnetic mode field, respectively.

In the default port formulation, when the **Use constraint-free port** check box is disabled (the default setting), the expansion coefficients (or S-parameters) are calculated by adding a scalar dependent variable for each coefficient and then adding a constraint to enforce the series expansion above.

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.

DISCRETIZATION

Select the shape order for the **Electric field** dependent variable — **Linear**, **Linear type 2**, **Quadratic** (the default), **Quadratic type 2**, **Cubic**, or **Cubic type 2**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

⊕
- Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Frequency Domain Interface
- Theory for the Electromagnetic Waves Interfaces

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

The Electromagnetic Waves, Frequency Domain Interface has these domain, boundary, edge, point, and pair nodes and subnodes. The nodes are listed in alphabetical order and are available from the **Physics** ribbon toolbar (Windows users),

Physics context menu (Mac or Linux users), or by right-clicking to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

DOMAIN

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

- Initial Values
- Polarization
- 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 **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. The following conditions are available (listed in alphabetical order):

- Diffraction Order
- Electric Field
- Impedance Boundary Condition
- Layered Transition Boundary Condition
- Magnetic Field
- Orthogonal Polarization
- Perfect Electric Conductor

- Perfect Magnetic Conductor
- Periodic Condition
- Port
- Scattering Boundary Condition
- Surface Current Density
- Surface Magnetic Current Density
- 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
- Reference Point
- Surface Current Density
- Surface Magnetic Current Density
- Symmetry Axis Reference Point

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

	In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Wave Equation, Electric

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

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

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

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

where c_0 is the speed of light in vacuum.

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

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

The wave equation is thereby rewritten as

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

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

Notice that the ansatz above just explains how the wave equation is modified when the out-of-plane wave vector component k_z is not zero. As an example, for a plane wave with a nonzero out-of-plane wave vector component, the electric field is of course given by

$$\mathbf{E}(x, y, z) = \tilde{\mathbf{E}}(x, y) \exp(-ik_z z) = \mathbf{A} \exp(-i(k_x x + k_y y + k_z z)) ,$$

where $\mathbf{A}$ is a constant amplitude and k_x, k_y, and k_z are the wave vector components.

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

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

where φ is the unit vector in the out-of-plane φ direction.

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

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

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

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

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

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

Note that the following material models can automatically be synchronized to any of the other **Electric displacement field model** settings:

- **Relative permittivity**
- **Refractive index**
- **Loss tangent, loss angle**
- **Loss tangent, dissipation factor**
- **Dielectric loss**

As an example, the material can be specified to use a Refractive index material model. Then the real and imaginary parts of the refractive index can be synchronized to compute the complex relative permittivity, if the **Electric displacement field model** is set to **Relative permittivity**.

When synchronizing to the **Refractive index Electric displacement field model**, the source material model is assumed to be isotropic.

When synchronizing to the **Loss tangent, loss angle** and **Loss tangent, dissipation factor Electric displacement field models**, the loss angle δ and the dissipation factor $\tan\delta$, respectively, must be converted to isotropic values.

Relative Permittivity

When **Relative permittivity** is selected, the default **Relative permittivity** ε_r takes values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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 **Full**. Enter values or expressions in the field or matrix.

The diagonal components of the input refractive index matrix correspond to the semi-axes of the so called index ellipsoid. You can orient the index ellipsoid by first creating a suitably oriented coordinate system below the **Definitions** node for the model component. Then select the created coordinate system in the **Coordinate system** setting in the **Coordinate System Selection** section in the settings for the **Wave Equation, Electric** feature.

⚠ 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, Loss Angle

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

Loss Tangent, Dissipation Factor

When **Loss tangent, dissipation factor** is selected, the default **Relative permittivity (real part)** ε' and **Loss tangent, dissipation factor** $\tan\delta$ take values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix. Then if **User defined** is selected for **Loss tangent, dissipation factor** $\tan\delta$, enter a value to specify a dissipation 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**. For **User defined** select

Isotropic, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix.

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

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

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

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

When **Drude-Lorentz dispersion model** is selected, the default **Relative permittivity, high frequency** ε_∞ (dimensionless) takes its value **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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 (rad/s)**.

Debye Dispersion Model
The **Debye dispersion model** is given by

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

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

When **Debye dispersion model** is selected, the default **Relative permittivity, high frequency** ε_∞ (dimensionless) takes its value **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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 (m^2)**.

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).

- For **Relative permeability** the relative permeability μ_r uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix.
- For **Magnetic losses** the default values for **Relative permeability (real part)** μ' and **Relative permeability (imaginary part)** μ'' are taken **From material**. For **User defined** enter different values.

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

- For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the current and enter values or expressions in the field or matrix.
- For **Linearized resistivity** 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: $\Omega \cdot m$) are taken **From material**. For **User defined** enter other values or expressions for any of these variables.

	For an example using the **Drude-Lorentz dispersion model**, see *Nanorods*: Application 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. Add additional **Initial Values** nodes from the **Physics** toolbar.

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 physics interface defines.

EXTERNAL CURRENT DENSITY

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

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. If a **Perfectly Matched Layer** (PML) node has been added before adding the **Far-Field Domain**, all of the domains in the Electromagnetic Waves, Frequency Domain interface adjacent to the PML are automatically selected by default. If there is no PML, all of the domains are selected. The selection can be modified. In that case, select only 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.

🔍	Modeling with Far-Field Calculations

▥	*Optical Scattering off a Gold Nanosphere*: Application Library path **Wave_Optics_Module/Optical_Scattering/scattering_nanosphere**

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. By default, all exterior boundaries of the Far-Field Domain are selected. If a **Perfectly Matched Layer** (PML) node has been added before adding the **Far-Field Domain**, all exterior boundaries of the Far-Field Domain adjacent to the PML are selected. Symmetry reduction of the geometry makes it relevant to select boundaries defining a nonclosed 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 they are facing away from infinity or toward infinity.

FAR-FIELD CALCULATION

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

Select as needed 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).

	If perfectly matched layers are added to the model after the Far-Field Domain is configured, then it is necessary to press the **Reset Far-Field Boundaries** button to reassign all exterior boundaries.

	Optical Scattering off a Gold Nanosphere: Application Library path **Wave_Optics_Module/Optical_Scattering/scattering_nanosphere**

Polarization

The **Polarization** node adds an externally generated polarization $\mathbf{P}_i$, which contributes to the total polarization

$$\mathbf{P} = \varepsilon_0(\varepsilon_r - 1)\mathbf{E} + \sum_i \mathbf{P}_i .$$

As indicated above, each **Polarization** node adds a contribution to the total polarization.

Add **Polarization** nodes from the **Physics** toolbar or by right-clicking the physics interface and selecting the **Polarization** item from the context menu.

POLARIZATION

Enter the components (**x**, **y**, and **z** for 3D components for example) of the **Polarization** $\mathbf{P}_i$ (SI unit: C/m^2).

	Second Harmonic Generation in the Frequency Domain: Application Library path **Wave_Optics_Module/Verification_Examples/ second_harmonic_generation_frequency_domain**

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 modeling of a lossless metallic surface (for example, a ground plane) or as a symmetry type boundary condition. It imposes symmetry for magnetic fields and "magnetic currents" and antisymmetry for electric fields and electric currents. It supports induced electric surface currents and thus any prescribed or induced electric currents (volume, surface, or edge currents) flowing into a *perfect electric conductor* boundary is automatically balanced by induced surface currents.
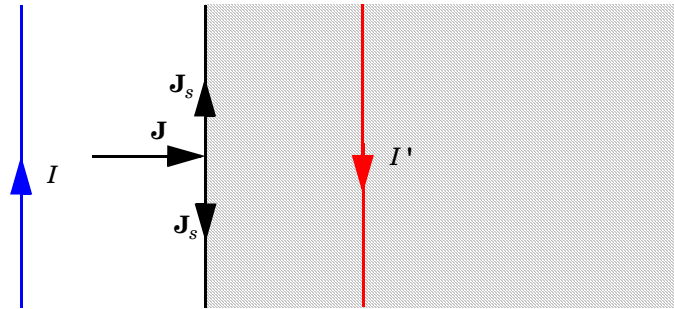
Figure 3-2: The perfect electric conductor boundary condition is used on exterior and interior boundaries representing the surface of a lossless metallic conductor or (on exterior boundaries) representing a symmetry cut. The shaded (metallic) region is not part of the model but still carries effective 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.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

▥	*Self-Focusing*: Application Library path **Wave_Optics_Module/ Nonlinear_Optics/self_focusing**

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 3-3: 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.

▥	*Optical Scattering off a Gold Nanosphere*: Application Library path **Wave_Optics_Module/Optical_Scattering/scattering_nanosphere**
	Self-Focusing: Application Library path **Wave_Optics_Module/Nonlinear_Optics/self_focusing**

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.

A port assumes that the cross section's geometry and material is constant in the normal direction. Furthermore, the port boundary is assumed to be flat, resulting in a constant normal across the boundary.

In 3D, the following subnodes are available from the context menu, by right-clicking the **Port** node, or from the **Physics** toolbar, **Attributes** menu:

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

PORT PROPERTIES

Enter a unique **Port name**. Only nonnegative integer numbers can be used as **Port 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**, **Coaxial**, **Circular**, or **Periodic**.

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

Numeric ports require a **Boundary Mode Analysis** study type. It should appear before the frequency or wavelength 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 (or Wavelength Domain 1), manually. **Numeric** ports are by default computed for the

deformed mesh whereas other types of ports compute the mode shape using geometry information.

The Boundary Mode Analysis study step stores the frequency f_{ref} and propagation constant β_{ref} for which it was run. For a TE, TM, or TEM mode, the propagation constant β for an arbitrary frequency f is given by

$$\beta^2 = \beta_{ref}^2 + k^2(1 - (f_{ref}/f)^2).$$

In addition, for TE, TM, and TEM modes, the mode field shape is independent of the frequency. Thus, when making a frequency sweep including only TE, TM, and TEM modes, the Boundary Mode Analysis study steps can be done for just one frequency, with the propagation constants obtained from the expression above for the other frequencies. For waveguides consisting of multiple dielectric materials, like optical fibers, where there are no TE, TM, or, TEM modes, the Boundary Mode Analysis steps must be recomputed for each frequency.

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 laser heating) more than one excited port might be wanted, but the S-parameter variables cannot be correctly computed, so when several ports are excited, the S-parameter output is turned off.

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 in 3D and 2D axisymmetry and W/m in 2D).

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.

Activate Slit Condition

Select the **Activate slit condition on interior port** check box to use the **Port** boundary condition on interior boundaries.

Then select a **Slit type** — **PEC-backed** (the default) or **Domain-backed**. The **PEC-backed** type makes the port on interior boundaries perform as it does on exterior boundaries. The **Domain-backed** type can be combined with perfectly matched layers to absorb the excited mode from a source port and other higher order modes.

Click **Toggle Power Flow Direction** button to define the power flow for the port. For an excited port, the power flow should point in to the excited domain and for a listener port the power flow should point out from the excited domain. The power flow direction is visualized with a red arrow on the port boundary in the Graphics window.

📝	When the **Slit type** is set to **Domain-backed**, there must be no waves reflected from the domain backing the port. Thus, the backing domain must have homogeneous material and geometric properties and it should be truncated by a PML domain or a nonreflecting boundary condition.
📝	When an active slit port is adjacent to a periodic condition, the periodic condition needs to be split at the active port, as the field is discontinuous at the point (in 2D) or edge (in 3D) where the slit port and the periodic condition meet.

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.

Set the **Mode phase** θ_{in} (SI unit: rad) for the port mode field. The default is 0 radians. For instance, if the inspected port mode field is polarized in the opposite direction compared to the expected direction, a **Mode phase** of π (enter pi in the field) can be used for polarizing the mode field in the expected direction. Notice that a change of the **Mode phase**, either on the exciting or the listener port, changes also the S-parameter coupling the exciting and the listener port. However, a change of the **Mode phase** on the exciting port does not modify the reflection coefficient (normally denoted S11) associated with the exciting port.

User Defined

For **User defined** specify the eigenmode of the outgoing wave at the port. Even if **Wave excitation at this port** is set to **On**, the mode field should represent the outgoing wave. The mode field can be entered with an arbitrary amplitude and is normalized internally.

- Enter the components of the **Electric mode field** $\mathbf{E}_0$ (SI unit: V/m) or the **Magnetic mode field** $\mathbf{H}_0$ (SI unit: A/m). The entered expressions must be differentiable.

- 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.

📄	Notice that for models saved before COMSOL 6.0, the mode field represented the incoming (exciting) wave. When opening such a model, the mode field is now automatically migrated to represent the outgoing wave.

📄	Notice that expressions including coupling operators are not differentiable and cannot be used as mode fields.

Rectangular

For **Rectangular** specify a unique rectangular mode.

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

For 2D components, 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.

Coaxial

In 2D axisymmetry, **Coaxial** does not support nonzero azimuthal mode number. The **Azimuthal mode number** in the **Physics interface** should be defined as zero.

Circular

For **Circular** specify a unique circular mode.

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

- Select the **Mode number** from the list.

For 3D components, enter the **Mode number**, for example, 11 for a TE_{11} mode, or 01 for a TM_{01} mode. When **Circular** is selected as the type of port in 3D, the Circular Port Reference Axis subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. It defines the orientation of fields on a port boundary.

For 2D axisymmetry components, select whether the **Azimuthal mode number** is defined in the **Physics interface** or if it is **User defined**. For **User defined** 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

For **Periodic**, specify parameters for the mode field. When **Periodic** is selected, the Diffraction Order port subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Select an **Input quantity** — **Electric field** or **Magnetic field** — and define the mode field amplitude for the outgoing wave at the port. Even if **Wave excitation at this port** is set to **On**, the mode field amplitude should represent the outgoing wave that corresponds to the actual incoming wave.

- For 2D components and if the **Input quantity** is set to **Electric field**, define the **Electric mode 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 mode field amplitude**. For a TM wave set the x, y, and z components to 0, 0, and 1.

- Define the **Angle of incidence**, if **Wave excitation at this port** is **On**.

For 3D components, if **Wave excitation at this port** is **On**, 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

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

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

where $\mathbf{k}$ is the wave vector, $\mathbf{k}_{parallel}$ 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 $\hat{\mathbf{a}}_1$ is one of the normalized primitive unit cell vectors from the periodic structure defined from Periodic Port Reference Point.

The **Elevation angle of incidence** α_1 is the angle between $\mathbf{n}$ and $\mathbf{k}$.

The **Azimuth angle of incidence** is the counterclockwise rotating angle from the primitive vector $\mathbf{a}_1$ around the axis built with Periodic Port Reference Point and $\mathbf{n}$.

For periodic ports with hexagonal port boundaries, the definition of the vector $\mathbf{a}_1$ is slightly different from the default definition. In this case, the unit cell is actually a rhomboid, with primitive vectors pointing in other directions than the side vectors of the hexagon. Thus, for a hexagonal periodic port, the vector $\mathbf{a}_1$ is defined along one of the sides of the hexagon, and it is not one of the primitive vectors of the hexagonal point lattice. The **Azimuth angle of incidence** α_2 is still measured from the vector $\mathbf{a}_1$, even though this vector now refers to a side vector of the hexagonal port boundary and not a primitive vector.

▥	*Hexagonal Grating*: Application Library path **Wave_Optics_Module/ Gratings_and_Metamaterials/hexagonal_grating**

For 2D components 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.

🗎	Notice that for models saved before COMSOL 6.0, the mode field represented the incoming (exciting) wave. When opening such a model, the mode field is now automatically migrated to represent the outgoing wave.

🗎	Notice that the mode field defined for the Periodic port assumes homogeneous isotropic material properties in the domain adjacent to the selected port boundary.

The propagation directions for listener **Periodic** ports are deduced from the angle setting(s) for the source **Periodic** port and the refractive indices defined for the source and the listener ports. Thus, adding source **Periodic** ports with different propagation angles will give ambiguous propagation directions for the listener **Periodic** ports.

Default Polarization plots are automatically generated for **Periodic** ports in 3D and in 2D, if the **Electric field components solved for** setting in the **Components** section for the physics interface is set to **Three-component vector**. The Polarization plot includes polarization ellipses for each diffraction order. The polarization ellipse line graphs are generated by plotting the in-plane Jones vector element versus the out-of-plane Jones vector element.

For more information about Jones Vectors and Polarization plots, see Jones Vectors and Polarization Plots.

AUTOMATIC DIFFRACTION ORDER CALCULATION

This section is only available for **Periodic** ports to provide parameter settings that are used when automatically adding Diffraction Order subnodes to **Periodic** ports.

- Select the **Include in automatic diffraction order calculation** check box to add **Diffraction Order** subnodes to the selected **Periodic** port, when the **Add Diffraction Orders** button is clicked from the exciting **Periodic** port.
- Define the **Refractive index, real part** at the boundary.
- Define the **Maximum frequency** — **From study** (the default) or **User defined**. When **From study** is selected, the **Maximum frequency** is taken from the study step associated with the physics interface. For **User defined**, enter the maximum frequency f_{max} (SI unit: Hz). The default value is 0 Hz. If a single frequency is used, insert the frequency, or if a frequency sweep is performed, insert the maximum frequency of the sweep. This parameter is only available when **Wave excitation at this port** is **On**.

When all parameters are defined, click the **Add Diffraction Orders** button from the exciting **Periodic** port to automatically create **Diffraction Order** ports as subnodes to all **Periodic** ports having the **Include in automatic diffraction order calculation** check box selected.

CALCULATOR

This utility is available for **Rectangular** and **Circular** port types. Enter a **Relative permittivity** (default is 1) of the material fully filled in a waveguide and then click the **Compute Waveguide Cutoff Frequency** button to compute the cutoff frequency for the particular waveguide geometry and the selected **Mode type**. The result is displayed in the **Messages** window.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

ℚ	• S-Parameters and Ports • S-Parameter Variables

▥	2D model with periodic ports — *Plasmonic Wire Grating*: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/plasmonic_wire_grating** 3D model with numeric ports — *Directional Coupler*: Application Library path **Wave_Optics_Module/Waveguides_and_Couplers/directional_coupler**

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, the **Circular Port Reference Axis** subnode is available from the context menu (right-click the Port parent node) or from the **Physics** toolbar, **Attributes** menu. 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.

Diffraction Order

The **Diffraction Order** port is available in 3D and 2D. When the **Type of Port** is set to **Periodic** under Port Properties, this subnode is available from the context menu (right-click the Port parent node) or from the **Physics** toolbar, **Attributes** menu.

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. Additional **Diffraction Order** ports subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

PORT PROPERTIES

Enter a unique **Port name**. Only nonnegative integer numbers can be used as **Port 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.

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**.

Diffraction Order
Specify an integer constant or an integer parameter expression for the **Diffraction order** m (the default is 0) and in 3D n (the default is 0).

Note that **In-plane vector** and **Out-of-plane vector** are based on the plane of diffraction which is constructed with the diffraction wave vector and the outward normal vector of the port boundary. The diffraction wave vector is defined by

$$\mathbf{k}_{\text{diffraction,parallel}} = \mathbf{k}_F + m\mathbf{G}_1 + n\mathbf{G}_2$$

$$\mathbf{k}_{\text{diffraction}} = \mathbf{k}_{\text{diffraction,parallel}} + \mathbf{n}k_{\text{diffraction,perpendicular}}$$

$$k_{\text{diffraction,perpendicular}} = \sqrt{k^2 - k^2_{\text{diffraction,parallel}}}$$

where m and n are diffraction orders, $k \geq k_{\text{diffraction,parallel}}$, k is the magnitude of the wave vector and $k_{\text{diffraction,parallel}}$ is the magnitude of $\mathbf{k}_{\text{diffraction,parallel}}$. The reciprocal lattice vectors, $\mathbf{G}_1$ and $\mathbf{G}_2$ are defined from Periodic Port Reference Point.

In-plane vector lies on the plane of diffraction while **Out-of-plane vector** is normal to the plane of diffraction.

For a 2D component, **In-plane vector** is available when the settings for the physics interface is set to either **In-plane vector** or **Three-component vector** under Components. **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 Components.

In 2D, the diffraction wave vector is defined by

$$\mathbf{k}_{\text{diffraction,parallel}} = \mathbf{k}_F + m\mathbf{G}_1,$$

where the reciprocal lattice vector is defined by

$$\mathbf{G}_1 = \frac{2\pi}{a}\mathbf{n} \times \mathbf{z},$$

where $\mathbf{n}$ is the port boundary normal and $\mathbf{z}$ is the unit vector in the out-of-plane direction. Since the normal $\mathbf{n}$ points in opposite directions for the exciting port boundary and the transmission side port boundary, $\mathbf{G}_1$ will point in different directions on the two opposing port boundaries. Thereby, also the mode numbers for the two port boundaries will be different. For example, mode $m = 1$ on the excitation side, corresponds to mode $m = -1$ on the transmission side.

Enter a value or expression for the **Mode phase** θ_{in} (SI unit: rad). The default is 0 radians. The **Mode phase** setting is further discussed for the Port feature.

	Notice that the mode field defined for the Diffraction Order feature assumes isotropic material properties in the domain adjacent to the selected feature boundary.

	• S-Parameters and Ports • S-Parameter Variables

	Plasmonic Wire Grating: Application Library path **Wave_Optics_Module/ Gratings_and_Metamaterials/plasmonic_wire_grating**

Orthogonal Polarization

The **Orthogonal Polarization** port is available in 3D and 2D. When the **Type of Port** is set to **Periodic** under Port Properties, this subnode is available from the context menu (right-click the Port parent node) or from the **Physics** toolbar, **Attributes** menu.

Use the **Orthogonal Polarization** port to define port with a mode that is orthogonal to the mode of the parent **Periodic Port**. Normally a **Orthogonal Polarization** node is added automatically during the **Periodic** port setup, but the **Orthogonal Polarization** port subnode is also available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. Only one **Orthogonal Polarization** node can be added per parent **Periodic Port**.

PORT PROPERTIES

Enter a unique **Port name**. Only nonnegative integer numbers can be used as **Port 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 **Orthogonal Polarization** port is a listener port feature.

PORT MODE SETTINGS

Diffraction Order
The **Orthogonal Polarization** port represent a zero-order mode (same as assumed for the parent Periodic port). Thus, the **Diffraction order** settings are just for information and cannot be edited.

Enter a value or expression for the **Mode phase** θ_{in} (SI unit: rad). The default is 0 radians. The **Mode phase** setting is further discussed for the Port feature.

	Notice that the mode field defined for the **Orthogonal Polarization** feature assumes isotropic material properties in the domain adjacent to the selected feature boundary.

	• S-Parameters and Ports
	• S-Parameter Variables

Periodic Port Reference Point

The **Periodic Port Reference Point** subnode is available only in 3D. When the **Type of Port** is set to **Periodic** under Port Properties, this subnode is available from the context menu (right-click the Port parent node) or from the **Physics** toolbar, **Attributes** menu.

The **Periodic Port Reference Point** is used to uniquely identify two primitive unit cell vectors, $\mathbf{a}_1$ and $\mathbf{a}_2$, and two reciprocal lattice vectors, $\mathbf{G}_1$ and $\mathbf{G}_2$. These reciprocal vectors are defined in terms of the unit cell 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

$$\frac{\mathbf{a}_1 \times \mathbf{a}_2}{\left|\mathbf{a}_1 \times \mathbf{a}_2\right|} = \mathbf{n}$$

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

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

POINT SELECTION

The primitive unit cell vectors, $\mathbf{a}_1$ and $\mathbf{a}_2$ are defined from two edges sharing the **Periodic Port Reference Point** on a port boundary. The two vectors can have unequal lengths and are not necessarily orthogonal. They start from the **Periodic Port Reference Point**.

For listener (passive, observation, and not excited) ports, if the outward normal vector on the listener port boundary is opposite to that of the source port, the listener port reference point needs to be mirrored from the source port reference point based on the center coordinate of the model domain. For example, if the source port reference point is at {−1,−1,1} in a cubic domain around the origin, the mirrored listener port reference point is {1,1,−1}. In this example, if the first and second primitive unit cell vectors are $\mathbf{a}_1$ and $\mathbf{a}_2$ on the source port, the first and second primitive unit cell vectors on the listener port will be -$\mathbf{a}_2$ and -$\mathbf{a}_1$, respectively, as the cross product between the first and second primitive unit cell vectors must point in the direction of the port normal. On the listener port, the normal points in the opposite direction to the normal on the source port. With the sign changes and the primitive unit cell vector index swaps, between the source and the listener ports, also the grating vectors change sign and swap indices, comparing the source and listener ports. Thus, the mode numbers

will also be different on the listener port compared to the mode numbers on the source port.

If the lattice vectors are collinear with two Cartesian axes, then the lattice vectors can be defined without the **Periodic Port Reference Point**. For the port where $\mathbf{n}$ points along a positive Cartesian direction, $\mathbf{a}_1$ and $\mathbf{a}_2$ are also assigned to point along positive Cartesian directions. Conversely, for the port where $\mathbf{n}$ points along a negative Cartesian direction, $\mathbf{a}_1$ and $\mathbf{a}_2$ are assigned to point along negative Cartesian directions. The condition $\mathbf{a}_1 \times \mathbf{a}_2 \parallel \mathbf{n}$ is true on both ports. For example, if $\mathbf{n} = \mathbf{z}$, then $\mathbf{a}_1/|\mathbf{a}_1| = \mathbf{x}$ and $\mathbf{a}_2/|\mathbf{a}_2| = \mathbf{y}$ and if $\mathbf{n} = -\mathbf{z}$, then $\mathbf{a}_1/|\mathbf{a}_1| = -\mathbf{y}$ and $\mathbf{a}_2/|\mathbf{a}_2| = -\mathbf{x}$.

▥	*Plasmonic Wire Grating*: Application Library path **Wave_Optics_Module/ Gratings_and_Metamaterials/plasmonic_wire_grating**

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.

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 More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

Magnetic Field

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

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

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

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

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

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

The field $\mathbf{E}_0$ is the incident plane wave that travels in the direction $\mathbf{k}$. The boundary condition is transparent for incoming (but not outgoing) plane waves with any angle of incidence. In addition, to an incident plane wave, $\mathbf{E}_0$ can also be the electric field distribution for a Gaussian beam that propagates in the direction $\mathbf{k}$.

If there is an incident field, a Reference Point subnode can be added by right-clicking the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. The Reference Point subnode redefines the incident field to be expressed as

$$\mathbf{E}_0 e^{-jk(\mathbf{k} \cdot (\mathbf{r} - \mathbf{r}_{ref}))},$$

where $\mathbf{r}_{ref}$ is a reference point determined by the Reference Point subnode. If no reference point subnode is added, the reference point is calculated as the average position boundary selection.

	In 2D axisymmetry, when incident field can be specified, the default subnode Symmetry Axis Reference Point is available. This subnode defines a reference point at the intersection between the symmetry axis and the Scattering boundary condition's boundary selection.

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, the Perfectly Matched Layer feature is available as 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.

The domain material adjacent to the boundary where the Scattering Boundary Condition is applied can be lossy.

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

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

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

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

SCATTERING BOUNDARY CONDITION

Select an **Incident field** — **No incident field** (the default), **Wave given by E field**, **Wave given by H field**, or **Gaussian beam**. Enter the expressions for the components for the **Incident electric field** $\mathbf{E}_0$ or **Incident magnetic field** $\mathbf{H}_0$.

If the **Incident field** is set to **Gaussian beam**, select an **Input quantity: Electric field amplitude** (the default) or **Power**. If the **Input quantity** is **Electric field amplitude**, enter the component expressions for the **Gaussian beam electric field amplitude** $\mathbf{E}_{g0}$ (SI unit: V/m). If the **Input quantity** is set to **Power**, enter the **Input power** (SI unit: W in 2D axisymmetry and 3D and W/m in 2D) and the component expressions for the **Gaussian beam non-normalized electric field amplitude** $\mathbf{E}_{g0}$ (SI unit: V/m). Also edit the **Beam radius** w_0 (SI unit: m) and the **Distance to focal plane** p_0 (SI unit: m). The default values are `((10*2)*pi)/ewfd.k0` and `0` m, respectively. The optical axis for the Gaussian beam is defined by a line including a reference point on the feature selection with a direction specified by the **Incident wave direction** (see below). By default, the reference point is the average position for the feature selection. However, by adding a Reference Point subnode any available point (or the average of several selected points) on the feature selection can be used as the reference point. The focal plane for the Gaussian beam is located the **Distance to focal plane** p_0 from the reference point in the **Incident wave direction**.

If the **Incident field** is not set to **No incident field**, edit the **Incident wave direction** $\mathbf{k}_{dir}$ for the vector coordinates. The default direction is in the opposite direction to the boundary normal. For 2D axisymmetry, the **Incident wave direction** $\mathbf{k}_{dir}$ should be parallel or anti-parallel to the symmetry axis.

Select a **Scattered wave type** for which the boundary is absorbing — **Plane wave** (the default), **Spherical wave**, or **Cylindrical wave**.

- For the Electromagnetic Waves, Frequency Domain interface, select an **Order** —**First order** (the default) or **Second order**.

- For **Cylindrical wave** also enter coordinates for the **Source point** $\mathbf{r}_0$ (SI unit: m) and **Source axis direction** $\mathbf{r}_{axis}$ (dimensionless). For 2D the **Source axis direction** is assumed to be in the z direction, whereas in 2D axisymmetry it is assumed to be along the axis of rotation.

- For **Spherical wave** enter coordinates for the **Source point** $\mathbf{r}_0$ (SI unit: m).

MODE ANALYSIS

Expand the **Mode Analysis** section and check the **Subtract propagation constant from material wave number** check box to calculate the wave number for the scattered wave as

$$k_n = \sqrt{k^2 - \beta^2},$$

where k_n is the wave number for the scattered wave propagating in the normal direction, k is the material wave number, and β is the propagation constant,

determined from the mode analysis. If the check box is unchecked (the default), $k_n = k$.

	For more information about the Gaussian beam theory, see Gaussian Beams as Background Fields and Input Fields.

	Beam Splitter: Application Library path **Wave_Optics_Module/ Optical_Scattering/beam_splitter**

	Leaky Modes in a Microstructured Optical Fiber: Application Library path **Wave_Optics_Module/Verification_Examples/ microstructured_optical_fiber**

INITIAL VALUES FOR INCIDENT WAVE

For the Electromagnetic Waves, Transient interface enter the components for the initial value of the **Magnetic vector potential A**$_0$ (SI unit: Wb/m).

	Second Harmonic Generation of a Gaussian Beam: Application Library path **Wave_Optics_Module/Nonlinear_Optics/second_harmonic_generation**

DISPERSION AND ABSORPTION

This section is only available for the Electromagnetic Waves, Transient interface. To display it, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Select the **Dispersion and absorption model** that will be used when calculating the wave number and attenuation constant for the incident and scattered waves — **Low loss**

approximation (the default), or **High loss**. For **High loss** also enter a **Carrier frequency** f_0 (SI unit: Hz). The default is 300 THz.

When the **Dispersion and absorption model** is set to **Low loss approximation** the refractive index is calculated from the relative permittivity and the relative permeability as

$$n = \sqrt{\varepsilon_r \mu_r} \, .$$

Similarly, the absorption coefficient is calculated as

$$\gamma = \frac{1}{2}\sigma \sqrt{\frac{\mu_0 \mu_r}{\varepsilon_0 \varepsilon_r}} = \frac{1}{2}\sigma Z_c \, ,$$

where Z_c is the characteristic impedance.

When the **Dispersion and absorption model** is set to **High loss**, the real and the imaginary parts of the complex refractive index is solved for from the real and the imaginary parts of the relative permittivity, using the relations

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

and

$$2n\kappa = \varepsilon''_r \mu_r = \frac{\sigma \mu_r}{\omega \varepsilon_0} \, .$$

The absorption coefficient is then given by

$$\gamma = \frac{\omega}{c}\kappa \, .$$

Reference Point

The **Reference Point** subnode is available only when there is an available incident field defined in the parent node. Then this subnode is available from the context menu (right-click the Scattering Boundary Condition or Matched Boundary Condition parent node) or from the **Physics** toolbar, **Attributes** menu.

The **Reference Point** subnode defines a reference position $\mathbf{r}_{\mathrm{ref}}$ that is calculated as the average position from the point selection in the **Reference Point** subnode or as a user-defined position on the parent feature boundary.

In the parent node, the incident field is then defined using the reference position:

$$\mathbf{E}_0 e^{-jk(\mathbf{k} \cdot (\mathbf{r} - \mathbf{r}_{ref}))} \, .$$

POINT SELECTION

Select the points that should be used when calculating the reference position. The reference position is calculated as the average position of the selected points. The point selection is only effective when **Definition** in the **Reference Point** section is set to **Point selection**.

REFERENCE POINT

Select the **Definition** for the reference point — **Point selection** (the default) or **User defined**. When **User defined** is selected, enter the expressions for the components for the **Reference point** r_0. The **Reference point** must be a point on the parent feature's boundary selection.

Symmetry Axis Reference Point

The **Symmetry Axis Reference Point** subnode is available in 2D axisymmetry, when there is an incident field defined in the parent node. Then this subnode is available as a default subnode to the Scattering Boundary Condition or Matched Boundary Condition parent node.

The **Symmetry Axis Reference Point** subnode defines a reference position $\mathbf{r}_{\mathrm{ref}}$ at the intersection point between the parent node's boundary selection and the symmetry axis. If the parent selection doesn't intersect the symmetry axis, the reference point is defined by the parent node's selection or an added Reference Point subnode to the parent node.

In the parent node, the incident field is then defined using the reference position:

$$\mathbf{E}_0 e^{-jk(\mathbf{k} \cdot (\mathbf{r} - \mathbf{r}_{ref}))} \, .$$

Impedance Boundary Condition

The **Impedance Boundary Condition**

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

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

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

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

When used with the Electromagnetic Waves, Beam Envelopes interface, the propagation direction in the exterior layer can be specified. Setting the propagation direction to be in the normal direction is the default option and results in the behavior as described above. However, setting the propagation direction to be given by the wave vector direction takes the tangential wave vector components from the **Wave Vectors** settings for the physics and the longitudinal component is derived to make the wave number satisfy the wave number for the exterior layer. This option is useful if the exterior domain is a dielectric material.

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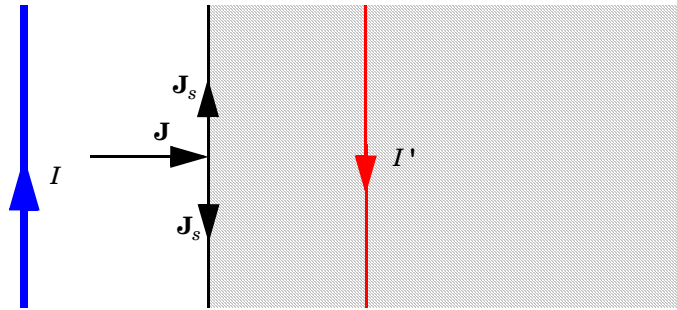
Figure 3-4: The impedance boundary condition is used on exterior boundaries representing the surface of a lossy domain. The shaded (lossy) region is not part of the model. The effective induced image currents are of reduced magnitude due to losses. Any current flowing into the boundary is perfectly balanced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

PROPAGATION DIRECTION

This section is only available for the Electromagnetic Waves, Beam Envelopes interface. Select a **Propagation direction** — **Normal direction** (the default) or **From wave vector**. The **Normal direction** option assumes that the wave in the exterior material propagates essentially in the normal direction, whereas the **From wave vector** option assumes that the tangential wave vector component is continuous at the boundary, as specified by the wave vectors $\mathbf{k}_1$ and $\mathbf{k}_2$ for the Electromagnetic Waves, Beam Envelopes interface. The normal component for the wave vector in the exterior material is obtained from the wave number, given the material parameters of the exterior domain. Thus, this option implements Snell's law of refraction at the boundary, which makes this option useful also for dielectric exterior materials.

IMPEDANCE BOUNDARY CONDITION

Select an **Electric displacement field model** — **Relative permittivity**, **Refractive index** (the default), **Loss tangent, loss angle**, **Loss tangent, dissipation factor**, **Dielectric loss**, **Drude-Lorentz dispersion model**, **Debye dispersion model**, or **Sellmeier dispersion model**. See the Wave Equation, Electric node, Electric Displacement Field section, for all settings.

SOURCE ELECTRIC FIELD

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Enter a **Source electric field** $\mathbf{E}_s$ (SI unit: V/m). The default is 0 V/m.

<table>
<tr>
<td>▥</td>
<td>*Fresnel Equations*: Application Library path **Wave_Optics_Module/ Verification_Examples/fresnel_equations**</td>
</tr>
</table>

Surface Current Density

The **Surface Current Density** boundary condition

$$-\mathbf{n} \times \mathbf{H} = \mathbf{J}_s$$

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

specifies a surface current density at both exterior and interior boundaries, respectively. 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.

For Transition Boundary Condition, the **Surface Current Density** subnode as an one-sided surface current density is available from the context menu (right-click the Transition Boundary Condition parent node) or from the **Physics** toolbar, **Attributes** menu.

SURFACE CURRENT DENSITY

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

For the **Surface Current Density** subnode, select **Side** — **Upside** (the default) or **Downside** to define on which side the **Surface Current Density** is applied. The red arrow visualized on the selected boundaries always indicates the upside.

Surface Magnetic Current Density

The **Surface Magnetic Current Density** boundary condition

$$\mathbf{n} \times \mathbf{E} = \mathbf{J}_{ms}$$

$$\mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) = -\mathbf{J}_{ms}$$

specifies a surface magnetic current density at both exterior and interior boundaries, respectively. The magnetic 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 magnetic current density and avoids unexpected results when a magnetic current density with a component normal to the surface is given.

SURFACE MAGNETIC CURRENT DENSITY

Enter values or expressions for the components of the **Surface magnetic current density** $\mathbf{J}_{ms0}$ (SI unit: V/m).

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

Transition Boundary Condition

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

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

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

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

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

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

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

> The **Transition Boundary Condition** is based on the assumption that the wave propagates in the normal direction in the thin layer. Thus, the wave could be incident in the normal direction or the wave could be refracted to propagate in a direction close to the normal direction. The latter condition is fulfilled for a good conductor.
>
> The thickness of the layer should also be less than the radius of curvature for the boundary.
>
> A consequence of the normal direction propagation assumption is that the **Transition Boundary Condition** is not compatible with mode analysis, as for mode analysis it is assumed that the wave predominantly propagates in the out-of-plane direction whereas the normal to the boundary is in an in-plane direction.

> For the Electromagnetic Waves, Beam Envelopes interface, the propagation in the layer can be specified to be given from the wave vectors specified for the physics and Snell's law of refraction. For this option, the assumption of normal propagation in the layer is not required. Thus, this option is useful also when the layer is a dielectric material.

> When adding a Transition Boundary Condition node, the phase specified for the Electromagnetic Waves, Beam Envelopes interface can be discontinuous.

PROPAGATION DIRECTION

This section is only available for the Electromagnetic Waves, Beam Envelopes interface. Select a **Propagation direction** — **Normal direction** (the default) or **From wave vector**. The **Normal direction** option assumes that the waves in the layer propagate essentially in the normal direction, whereas the **From wave vector** option assumes that the tangential wave vector component is continuous at the layer boundaries, as specified by the wave vectors $\mathbf{k}_1$ and $\mathbf{k}_2$ for the Electromagnetic Waves, Beam Envelopes interface. The normal component for the wave vector in the layer is obtained from the wave number, given the specified material parameters. Thus, this

option implements Snell's law of refraction for the layer, which makes this option useful also for dielectric layers.

TRANSITION BOUNDARY CONDITION

Select an **Electric displacement field model** — **Relative permittivity**, **Refractive index** (the default), **Loss tangent, loss angle**, **Loss tangent, dissipation factor**, **Dielectric loss**, **Drude-Lorentz dispersion model**, **Debye dispersion model**, or **Sellmeier dispersion model**. See the Wave Equation, Electric node, Electric Displacement Field section, for all settings.

Select the **Electrically thick layer** check box (unselected by default) to make the two domains adjacent to the boundary uncoupled. Use this setting, for instance, when the thickness is greater than three times of the skin depth. When the **Electrically thick layer** check box is unselected, enter a **Thickness** d (SI unit: m). The default is 0.01 m.

▥	• *Beam Splitter*: Application Library path **Wave_Optics_Module/ Optical_Scattering/beam_splitter** • *Focusing Lens*: Application Library path **Wave_Optics_Module/ Verification_Examples/focusing_lens**

Layered Transition Boundary Condition

The **Layered Transition Boundary Condition** is an extension of the **Transition Boundary Condition** that allows to model a sequence of geometrically thin layers using a **Layered Material**. It represents a discontinuity in the tangential electric field. For each layer in the **Layered Material**, the transfer and surface impedances are obtained from the layer thickness and material properties. The impedances are then used to relate the discontinuity in the tangential electric field to the current flowing on the surface of either side (up/down) of the corresponding layer. Mathematically this reads:

$$\mathbf{J}_{s,up,i} = \frac{(Z_S\mathbf{E}_{t,i} - Z_T\mathbf{E}_{t,i-1})}{Z_S^2 - Z_T^2}$$

$$\mathbf{J}_{s,down,i} = \frac{(Z_S\mathbf{E}_{t,i-1} - Z_T\mathbf{E}_{t,i})}{Z_S^2 - Z_T^2}$$

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

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

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

where the index $i = 1, 2,..., n$ refers to the layer number. The system of equations above is solved for each layer in the **Layered Material**. The index i has been omitted from the expressions of the impedances and the wave vector k in order to improve their readability.
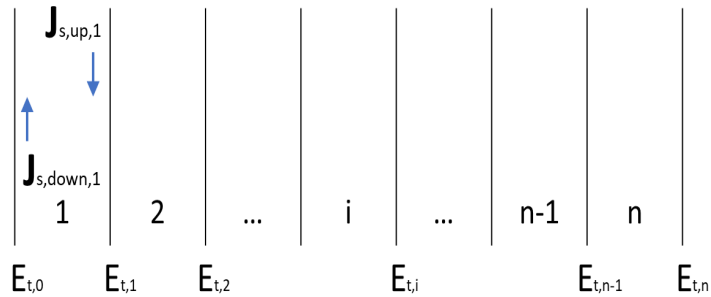
Figure 3-5: The layered material is composed of n layers. The surface currents on the up and downside of each layer are determined from the transfer and surface impedances and are functions of the tangential electric fields.

- See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the COMSOL Multiphysics Reference Manual for details on the definition of layered materials.

The **Layered Transition Boundary Condition** is based on the assumption that the wave propagates in the normal direction in the thin layer. Thus, the wave could be incident in the normal direction or the wave could be refracted to propagate in a direction close to the normal direction. The latter condition is fulfilled for a good conductor.

The thickness of the layer should be less than the radius of curvature for the boundary.

A consequence of the normal direction propagation assumption is that the **Layered Transition Boundary Condition** is not compatible with mode analysis, as for mode analysis it is assumed that the wave predominantly propagates in the out-of-plane direction whereas the normal to the boundary is in an in-plane direction.

SHELL PROPERTIES

The **Shell Properties** section displays which layered material the **Layered Transition Boundary Condition** is coupled to.

Clear the **Use all layers** check box in order to select a specific layered material from the list. The **Layered Transition Boundary Condition** feature is then applicable only on the boundaries where the chosen material is defined.

You can visualize the selected layered material and the layers that constitute it by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

The thickness of the layered material should be set as follows, depending on the type of material:

• In a **Material** node, the layer **Thickness** is set in the **Material Contents** section by adding a **Shell** property group from the **Material Properties** section in the material **Settings** window. This automatically adds a **Shell** subnode under the **Material** node, transforming it into a layered material.

• When the layered material is a **Single Layer Material**, the **Thickness** is set in the **Material Contents** section in the **Settings** window. Alternatively it can be set in the **Layer Definition** section of the **Shell** property group **Settings** window.

• For a general **Layered Material**, added through a **Layered Material Link** or a **Layered Material Stack**, the **Thickness** is set in the **Layer Definition** section of the **Settings** window. Several layers may be defined in the table, and the **Thickness** should be

defined for each of them. The total thickness of the layered material is the sum of all the layers thicknesses.

LAYERED TRANSITION BOUNDARY CONDITION

The **Layered Transition Boundary Condition** section lists the material properties of the thin layered material that this boundary condition approximates:

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

The defaults use the values **From material**, taking the properties from the layered material specified for the boundary. Otherwise, choose **User defined** and enter different values or expressions. In the latter case all layers constituting the chosen layered material will take on the same value for the selected property.

Periodic Condition

The **Periodic Condition** sets up a periodicity between the selected boundaries. The **Destination Selection** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

BOUNDARY SELECTION

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, add a **Destination Selection** subnode. By default it contains the selection that COMSOL Multiphysics has identified.

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 Interface and The Electromagnetic Waves, Beam Envelopes Interface only) to use a Floquet periodicity (Bloch-Floquet periodicity).

    - For **Floquet periodicity** also enter the source for the **k-vector for Floquet periodicity**.

    - For **User defined** specify the components of the **k-vector for Floquet periodicity** $\mathbf{k}_F$ (SI unit: rad/m).

    - For **From periodic port** 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 More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box. For information about the **Constraint Settings** section, see Constraint Settings in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF SOURCE

For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

🔍	Periodic Boundary Conditions

▥	*Fresnel Equations*: Application Library path **Wave_Optics_Module/Verification_Examples/fresnel_equations**
	Plasmonic Wire Grating: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/plasmonic_wire_grating**

Magnetic Current

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. 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**.

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 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.

DIPOLE SPECIFICATION

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

DIPOLE PARAMETERS

Based on the **Dipole specification** selection:

- For **Magnitude and direction** enter coordinates for the **Electric current dipole moment direction** $\mathbf{n}_p$ and **Electric current dipole moment, magnitude** p (SI unit: A·m).

- For **Dipole moment** enter coordinates for the **Electric current dipole moment** $\mathbf{p}$ (SI unit: A·m).

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 $I^{\star}a$.

DIPOLE SPECIFICATION

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

DIPOLE PARAMETERS

Based on the **Dipole specification** selection:

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

- For **Dipole moment** enter coordinates for the **Magnetic dipole moment** $\mathbf{m}$ (SI unit: $m^2 \cdot A$).

Line Current (Out-of-Plane)

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

LINE CURRENT (OUT-OF-PLANE)

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

The Electromagnetic Waves, Transient Interface

The **Electromagnetic Waves, Transient (ewt)** interface ( ), found under the **Wave Optics** branch ( ) when adding a physics interface, is used to solve a time-domain wave equation for the magnetic vector potential. The sources can be in the form of point dipoles, line currents, or incident fields on boundaries or domains. It is primarily used to model electromagnetic wave propagation in different media and structures when a time-domain solution is required — for example, for non-sinusoidal waveforms or for nonlinear media. Typical applications involve the propagation of electromagnetic pulses and the generation of harmonics in nonlinear optical media.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Wave Equation, Electric**, **Perfect Electric Conductor**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and mass sources. You can also right-click **Electromagnetic Waves, Transient** to select physics features from the context menu.

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

SETTINGS

The **Label** is the default physics interface name.

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

The default **Name** (for the first physics interface in the model) is `ewt`.

COMPONENTS

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

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.

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.

DISCRETIZATION

Select the shape order for the **Magnetic vector potential** dependent variable — **Linear**, **Quadratic** (the default), or **Cubic**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

	• Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Transient Interface
🔍	• Theory for the Electromagnetic Waves Interfaces

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

The Electromagnetic Waves, Transient Interface shares most of its nodes with The Electromagnetic Waves, Frequency Domain Interface.

The domain, boundary, edge, point, and pair nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.
💡	

DOMAIN

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

- Wave Equation, Electric

- Initial Values
- Drude–Lorentz Polarization
- Far-Field Domain
- Far-Field Calculation

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 **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 Density

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 components)
- Line Current (Out-of-Plane) (2D and 2D axisymmetric components)
- Magnetic Point Dipole (2D and 3D components)
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Surface Current Density

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

In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

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.

ELECTRIC DISPLACEMENT FIELD

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

Relative Permittivity

When **Relative permittivity** is selected, the default **Relative permittivity** ε_r (dimensionless) takes values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** and enter values or expressions in the field or matrix.

Refractive Index

When **Refractive index** is selected, the default **Refractive index** n (dimensionless) takes the value **From material**. To specify 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 **Full**. Enter values or expressions in the field or matrix.

> ⚠️ Notice that only the real part of the refractive index is used for the transient formulation.

Polarization

For **Polarization** enter coordinates for the **Polarization P** (SI unit: C/m^2).

Remanent Electric Displacement

For **Remanent electric displacement** enter coordinates for the **Remanent electric displacement** $\mathbf{D_r}$ (SI unit: C/m^2). Then select **User defined** or **From Material** as above for the **Relative permittivity** ε_r.

Drude–Lorentz Dispersion Model

For **Drude-Lorentz Dispersion Model** select **User defined** or **From Material** for the **Relative permittivity, high-frequency** ε_∞ and enter a value for the **Plasma frequency** ω_p (SI unit: rad/s).

When **Drude-Lorentz Dispersion Model** is selected, the **Drude-Lorentz Polarization** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. Each **Drude-Lorentz Polarization** subnode adds another polarization term $\mathbf{P}_n$ to the electric displacement field $\mathbf{D}$, defined by

$$
\mathbf{D} = \varepsilon_0 \varepsilon_\infty \mathbf{E} + \sum_{n=1}^{N} \mathbf{P}_n,
$$

where the polarization is the solution to the ordinary differential equation

$$\left(\frac{\partial^2}{\partial t^2} + \Gamma_n \frac{\partial}{\partial t} + \omega_n^2 \right) \mathbf{P}_n = \varepsilon_0 f_n \omega_p^2 \mathbf{E} \, .$$

For more information, see the Drude–Lorentz Polarization feature.

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
For **Relative permeability** the relative permeability μ_r uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the magnetic field, and then enter values or expressions in the field or matrix.

Remanent Flux Density
For **Remanent flux density** the relative permeability μ_r uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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
For **Magnetization** 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**.

- For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the current and enter values or expressions in the field or matrix.

- For **Linearized resistivity** 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**. For **User defined** 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.

INITIAL VALUES

Enter values or expressions for the initial values of the components of the magnetic vector potential **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.

Drude–Lorentz Polarization

This subfeature is available only when **Drude-Lorentz Dispersion Model** is selected as the **Electric displacement field model** in the Wave Equation, Electric feature node. Then the subnodes are made available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu.

Each **Drude-Lorentz Polarization** subnode adds another polarization term $\mathbf{P}_n$ to the electric displacement field **D**, defined by

$$\mathbf{D} = \varepsilon_0\varepsilon_\infty\mathbf{E} + \sum_{n=1}^{N} \mathbf{P}_n ,$$

where the polarization is the solution to the ordinary differential equation

$$\left(\frac{\partial^2}{\partial t^2} + \Gamma_n\frac{\partial}{\partial t} + \omega_n^2\right)\mathbf{P}_n = \varepsilon_0 f_n \omega_p^2 \mathbf{E} .$$

Here Γ_n is a damping coefficient, ω_n is a resonance frequency, f_n is an oscillator strength, and ω_p is the plasma frequency.

Enter values or expressions for the **Oscillator strength** f_n (SI unit: 1), the **Resonance frequency** ω_n (SI unit: rad/s), and the **Damping in time** coefficient Γ_n (SI unit: rad/s).

INITIAL VALUES

Enter values or expressions for the initial values of the components of the Drude–Lorentz polarization $\mathbf{P}_n$ (SI unit: C/m^2) and its time derivative $\partial\mathbf{P}_n/\partial t$ (SI unit: A/m^2).

DISCRETIZATION

To display this section, click the **Show More Options** button (⊚) and select **Discretization** from the **Show More Options** dialog box. Select the element order from the list box for the Drude-Lorentz polarization $\mathbf{P}_n$.

▥	*Time-Domain Modeling of Dispersive Drude–Lorentz Media*: Application Library path **Wave_Optics_Module/ Gratings_and_Metamaterials/drude_lorentz_media**

The Electromagnetic Waves, Time Explicit Interface

The **Electromagnetic Waves, Time Explicit (teew)** interface (⦿), found under the **Wave Optics** branch (⦿) when adding a physics interface, is used to model time-dependent electromagnetic wave propagation in linear media. The sources can be in the form of volumetric electric or magnetic currents, or electric surface currents or fields on boundaries.

This physics interface solves two first-order partial differential equations (Faraday's law and Maxwell-Ampère's law) for the electric and magnetic fields using the time explicit discontinuous Galerkin method.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Wave Equations**, **Perfect Electric Conductor**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Electromagnetic Waves, Time Explicit** to select physics features from the context menu.

The interface includes absorbing layers that are used to set up effective nonreflecting like boundary conditions. These features are added from the **Definitions** toolbar, by clicking **Absorbing Layer**. If COMSOL Multiphysics is not running in full-screen mode nor in a large window, **Absorbing Layer** is accessible on the **Definitions** toolbar by first clicking **Coordinate Systems** and then **Absorbing Layer**. You can also right-click **Definitions** in the **Model Builder** and select **Absorbing Layer** from the context menu.

SETTINGS

The **Label** is the default physics interface name.

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

The default **Name** (for the first physics interface in the model) is `teew`.

COMPONENTS

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

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.

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.

DISCRETIZATION

Select the shape order for the **Electric and magnetic fields** dependent variables (the same order for both fields) — **Linear**, **Quadratic**, or **Cubic** (the default). For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

🔍
- Domain, Boundary, and Pair Nodes for the Electromagnetic Waves, Time Explicit Interface
- Theory for the Electromagnetic Waves, Time Explicit Interface

---

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

The Electromagnetic Waves, Time Explicit Interface has these domain and boundary nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

💡
In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

	In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

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

- Background Field
- Electric Field
- Electric Current Density
- Flux/Source
- Initial Values
- Magnetic Current Density

- Magnetic Field
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Scattering Boundary Condition
- Surface Current Density
- Wave Equations

Wave Equations

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

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

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

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

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

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

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**.

For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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 $0.5\,Z$ 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 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 a 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 is a smaller part than for a poorly resolved solution. The effect is stronger for poorly resolved solutions than for well resolved ones. This is one of the reasons why this filter is useful in an absorbing layer where the energy is transferred to the higher-order coefficients through a coordinate transformation. See Ref. 1 (Chapter 5) for more information.

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

When using **Absorbing Layer** features, add an additional **Wave Equations** feature for the corresponding domain selection. Select the **Activate** check box and add filter parameters. An example of a filter parameter combination that can be used for a **Wave Equations** feature active on an **Absorbing Layer** domain selection is $\alpha = 0.1$, $\eta_c = 0.01$, and $s = 2$. However, other combinations could work better, depending on the particular application.

Q | Absorbing Layers

Reference
1. J.S. Hesthaven and T. 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

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

INITIAL VALUES

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

Electric Current Density

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

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

ELECTRIC CURRENT DENSITY

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

Magnetic Current Density

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

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

MAGNETIC CURRENT DENSITY

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

Electric Field

The **Electric Field** boundary condition

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

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

ELECTRIC FIELD

Enter values or expressions for the components of the **Electric field 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.

Magnetic Field

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

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

MAGNETIC FIELD

Enter values or expressions for the components of the **Magnetic field 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 density 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.

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.

SURFACE CURRENT DENSITY

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

Scattering Boundary Condition

The **Scattering Boundary Condition**

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

specifies the tangential component of both electric and magnetic fields.

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 select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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. This boundary condition is available when **Advanced Physics Options** is selected in the **Show More Options** dialog box on the **Model Builder** toolbar.

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).

Background Field

The **Background Field** feature triggers the scattered field formulation, where the dependent variable is the relative field. The same wave equations are used as in the full field formulation, but the total field that enters the equations are written as the sum of the relative field and the background field, $\mathbf{E} = \mathbf{E}_{\text{relative}} + \mathbf{E}_{\text{background}}$, and it is the dependent variable $\mathbf{E}_{\text{relative}}$ that is solved for. When the background field is a solution of the wave equation, the relative field is the scattered field.

SETTINGS

Select a **Background wave type** — **User defined** (the default), or **Modulated Gaussian pulse**.

User Defined
Enter the component expressions for the **Background electric field $\mathbf{E}_b$** (SI unit: V/m) and **Background magnetic field $\mathbf{H}_b$** (SI unit: A/m). The entered expressions must be differentiable in time domain since the derivative of the background field is used in the governing equations.

Modulated Gaussian pulse
Select a **Direction of propagation** — **+x** (the default), **-x**, **+y**, **-y**, or for 3D components, **Along the +z**, or **-z**.

Select a **Polarization direction**— **y** (the default), **z**, or **x**. The list of available polarization varies based on the selection of **Direction of propagation**.

- Enter a **Center frequency** f_0 (SI unit: Hz). The default is 1 GHz.
- Enter a **Phase velocity** v_p (SI unit: m/s). The default is c_const.
- Enter a **Wave impedance** Z (SI unit: Ω). The default is Z0_const.
- Enter a **Distance from origin to wave launching plane** d_{offset} (SI unit: m). The default is 0 m.

For a modulated Gaussian pulse propagating in the positive x direction, the electric field is expressed as

$$E(x, t) \;=\; \frac{1}{\tau\sqrt{2\pi}}\exp\left(-\frac{\left(t-\mu-\dfrac{x+d_{offset}}{v_p}\right)^2}{2\tau^2}\right)\sin\left(2\pi f_0\left(t-\frac{x}{v_p}\right)\right)$$

where τ is the pulse duration, defined as $1/2f_0$, μ is a time delay set to $2/f_0$, and v_p is the phase velocity. The time delay μ is used to excite a modulated Gaussian pulse whose initial magnitude is very small when it is launched and gradually increases as it propagates.

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. By default, all of the domains are selected. The selection can be modified. In that case, select only 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.

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. By default, all exterior boundaries of the Far-Field Domain are selected. Symmetry reduction of the geometry makes it relevant to select boundaries defining a nonclosed 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 they are facing away from infinity or toward infinity.

FAR-FIELD CALCULATION

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

Select as needed 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).

!	A **Time to Frequency FFT** study step must be added after the **Time Dependent** study step to generate the necessary frequency-domain data, used in the far-field analysis.

The Electromagnetic Waves, Beam Envelopes Interface

The **Electromagnetic Waves, Beam Envelopes (ewbe)** interface (⚬), found under the **Wave Optics** branch (⚬) when adding a physics interface, is used to compute electric and magnetic field distributions for systems and devices where the field amplitude varies slowly on a wavelength scale.

The physics interface can be used efficiently for unidirectional and bidirectional propagation of electromagnetic beams. However, for optical scattering phenomena, where the field is scattered into many different directions, the Electromagnetic Waves, Frequency Domain interface is better suited.

With this physics interface the electric field is factored into a product of a slowly varying envelope function (slowly on the scale of a wavelength) and a rapidly varying phase function. The phase function is a priori prescribed, so the physics interface solves the time-harmonic wave equation for the slowly varying envelope function.

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

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Wave Equation, Beam Envelopes**, **Perfect Electric Conductor**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Electromagnetic Waves, Beam Envelopes** to select physics features from the context menu.

Physics-controlled mesh
The physics-controlled mesh is controlled from the **Mesh** node's **Settings** window (if the **Sequence type** is **Physics-controlled mesh**). There, in the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the check box in the **Use** column on the same table row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

When the **Use** check box for the physics interface is selected, in the section for the physics interface below the table, choose the **Mesh type** — **Swept mesh** (default for 3D), **Mapped mesh** (default for 2D), **Tetrahedral mesh** (3D), and **Triangular mesh** (2D).

When a structured **Mesh type** (either **Swept mesh** in 3D or **Mapped mesh** in 2D) is selected, enter values for **Number of transverse mesh elements** (default is 10) and **Number of longitudinal mesh elements** (default is 10). The entered **Number of transverse mesh elements** will be distributed along the longest side of the input boundary. A boundary is identified as an input boundary if there is an active feature, like a Port, a Scattering Boundary Condition, *etc*, added to that boundary and the feature defines an incident wave. The mesh will be denser in domains where the refractive index is larger. Similarly, the entered **Number of longitudinal mesh elements** will be distributed along propagation direction. Also here, the mesh will be denser in domains where the refractive index is larger.

If no input features are defined, for instance for an eigenfrequency simulation, the longitudinal direction is assumed to be the longest direction of the geometry and the transverse plane is orthogonal to the longitudinal direction.

When an unstructured **Mesh type** (either **Tetrahedral mesh** in 3D or **Triangular mesh** in 2D) is selected, enter a value for the **Maximum element size in free space**. The physics-controlled mesh automatically scales the maximum mesh element size as the material wavelength changes in different dielectric and magnetic regions.

If the model is configured by any periodic conditions, identical meshes are generated on each pair of periodic boundaries. Perfectly matched layers are built with a structured mesh, specifically, a swept mesh in 3D and a mapped mesh in 2D.

▥	For an example using the **Physics-controlled mesh** with a **Swept mesh**, see *Directional Coupler*: Application Library path **Wave_Optics_Module/ Waveguides_and_Couplers/directional_coupler**.

▥	For an example using the **Physics-controlled mesh** with a **Triangular mesh**, see *Gaussian Beam Incident at the Brewster Angle*: Application Library path **Wave_Optics_Module/Optical_Scattering/brewster_interface**

> In the *COMSOL Multiphysics Reference Manual* see the
> Physics-Controlled Mesh section for more information about how to
> define the physics-controlled mesh.

SETTINGS

The **Label** is the default physics interface name.

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

The default **Name** (for the first physics interface in the model) is ewbe.

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**.

Select the **Type of phase specification** — **Wave vector** (the default) or **User defined**.

In the tables, if **Wave vector** is selected for **Type of phase specification**, 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).

If **User defined** is selected for **Type of phase specification**, enter an expression for **Phase, first wave ϕ_1** (SI unit: rad) and, if **Bidirectional** is selected, for **Phase, second wave ϕ_2** (SI unit: rad).

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

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r})\exp(-j\phi_1(\mathbf{r})),$$

where $\mathbf{E}_1$ is the electric field envelope that is solved for and $\exp(-j\phi_1(\mathbf{r}))$ is the prescribed rapidly varying phase function. When **Wave vector** is selected for **Type of phase specification**, the phase is defined as

$$\phi_1(\mathbf{r}) = \mathbf{k}_1 \cdot \mathbf{r}.$$

Notice that the wave vector $\mathbf{k}_1$ is assumed to be the same for all domains selected for the physics interface. This also means that the phase will satisfy the condition of being continuous everywhere. If the wave is assumed to bend or there are different materials in the domains, the phase approximation above is not good and it is better to select a **User defined Type of phase specification**. When specifying the phase expression $\phi_1(\mathbf{r})$, it is important that it is continuous everywhere.

> The requirement that the phase is everywhere continuous is not applicable for the following boundary conditions:
>
> - Transition Boundary Condition
> - Field Continuity
>
> These two boundary conditions allow that the phase is different in the two domains adjacent to the boundary.

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 Electromagnetic Waves, Frequency Domain 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

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r})\exp(-j\phi_1(\mathbf{r})) + \mathbf{E}_2(\mathbf{r})\exp(-j\phi_2(\mathbf{r})),$$

where $\mathbf{E}_2$ and $\exp(-j\phi_2(\mathbf{r}))$ are the electric field envelope and the prescribed phase function for the second wave. When specifying **User defined** phases, ϕ_1 and ϕ_2, each phase should be continuous across the boundaries.

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 Common Physics Interface and Feature Settings and Nodes.

	In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

	For 2D and 3D, the default value for $\mathbf{k}_1$ (or ϕ_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 reversed, whereas the other components are the same as for wave vector of the incident wave.
	The default value for the **User defined** phase for the second wave, ϕ_2, represents a wave propagating in the opposite direction to the first wave.

	For 2D axisymmetry, the default value for $\mathbf{k}_1$ (or ϕ_1) represents a wave vector pointing in the z direction, whereas $\mathbf{k}_2$ (or ϕ_2) represents a wave propagating in the opposite direction to the first wave.

	For an example using the **User defined Type of phase specification**, see *Gaussian Beam Incident at the Brewster Angle*: Application Library path **Wave_Optics_Module/Optical_Scattering/brewster_interface**.

USER DEFINED WAVE VECTOR SPECIFICATION

This section is available when the **Type of phase specification** is set to **User defined**. Expand the section and 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).

> The default values for the wave vectors are the gradients of the corresponding phases defined in the **Wave Vector** settings. These values will be correct for most cases. However, they will be wrong for Perfectly Matched Layer domains. There, it is better to explicitly specify the wave vector. For example, if the wave solution is expected to approximate a plane wave in vacuum (or air), it would be better to enter the vacuum wave number ewbe.k0 in the appropriate component 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 in addition to the automatically generated frequency or wavelength sweep.

For **Activate port sweep** 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 physics interface, the S-parameters are subject to **Touchstone file export**. Click **Browse** to locate the file, or enter a filename 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 Ω.

PORT OPTIONS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

The electric field on port boundaries is expanded as

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_{\mathrm{inc}}(\mathbf{r}) + \sum_i S_i \mathbf{E}_i(\mathbf{r}) \exp(-\alpha_i \mathbf{n} \cdot (\mathbf{r} - \mathbf{r}_0)),$$

where $\mathbf{E}$, $\mathbf{r}$, $\mathbf{E}_{\mathrm{inc}}$, S_i, $\mathbf{E}_i$, α_i, $\mathbf{n}$, $\mathbf{r}_0$ are, respectively, the electric field on the port boundary, the position vector, the incident electric field, the expansion coefficient (or

S-parameter), the mode field, the propagation constant for the mode, the normal vector, and a position on the port boundary.

Select the **Use constraint-free ports** check box to calculate the expansion coefficients as overlap integrals

$$S_i = \frac{1}{2P_i} \int_S (\mathbf{E} - \mathbf{E}_{inc}) \times \mathbf{H}_i^* \cdot \mathbf{n} \; dS \; ,$$

where P_i, S, and $\mathbf{H}_i$ are the mode power, the port boundary area, and the magnetic mode field, respectively.

In the default port formulation, when the **Use constraint-free port** check box is disabled (the default setting), the expansion coefficients (or S-parameters) are calculated by adding a scalar dependent variable for each coefficient and then adding a constraint to enforce the series expansion above.

LOSS CALCULATION

Select the **Use averaged loss calculation** check box to remove cross terms between the two waves, when calculating the electromagnetic loss (and heat source) in the bidirectional formulation. This setting can be useful when the cross terms create a beating field that is not resolved by the mesh. If this spatially fast varying heat source distribution anyhow is washed out by the heat transfer, it can be advantageous to not include the cross terms when calculating the electromagnetic loss (and heat source). If the cross terms are of importance when solving the associated heat transfer problem, use a mesh that is fine enough to resolve the fast spatial variations of the heat source.

DISCRETIZATION

Select the shape order for the **Electric field envelopes** dependent variables — **Linear**, **Linear type 2**, **Quadratic** (the default), **Quadratic type 2**, **Cubic**, or **Cubic type 2**. For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

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

---

The dependent variables (field variables) are for the:

- **Electric field envelope, first wave $\mathbf{E1}$** and its components (in the **Electric field envelope components, first wave** fields).

- **Electric field envelope, second wave $\mathbf{E2}$** 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.

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

The Electromagnetic Waves, Beam Envelopes Interface has these domain, boundary, edge, point, and pair nodes available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

DOMAIN

- Initial Values
- Polarization
- Wave Equation, Beam Envelopes

BOUNDARY CONDITIONS

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

$$\mathbf{n}_{II} \times (\mathbf{E}_I - \mathbf{E}_{II}) = \mathbf{0}$$
$$\mathbf{n}_{II} \times (\mathbf{H}_I - \mathbf{H}_{II}) = \mathbf{0}\,,$$

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 physics interface and described in this section:

- Electric Field
- Field Continuity
- Magnetic Field

- Matched Boundary Condition
- Scattering Boundary Condition
- Surface Current Density

These features are also available and described for The Electromagnetic Waves, Frequency Domain Interface:

- Diffraction Order
- Impedance Boundary Condition
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Condition

- Port
- Reference Point
- Symmetry Axis Reference Point
- Transition Boundary Condition

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

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r})\exp(-j\varphi_1(\mathbf{r})),$$

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

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

where

$$\mathbf{k}_1 = \nabla\varphi_1.$$

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

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

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

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r})\exp(-j\varphi_1(\mathbf{r})) + \mathbf{E}_2(\mathbf{r})\exp(-j\varphi_2(\mathbf{r}))$$

The second field adds an additional wave equation to solve

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

where

$$\mathbf{k}_2 = \nabla\varphi_2.$$

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.

Polarization

The **Polarization** node adds an externally generated polarization $\mathbf{P}_i$, which contributes to the total polarization

$$\mathbf{P} = \varepsilon_0(\varepsilon_r - 1)\mathbf{E} + \sum_i \mathbf{P}_i.$$

As indicated above, each **Polarization** node adds a contribution to the total polarization. For the bidirectional formulation a polarization should be input for each wave and the polarization contribution is the sum of the two waves' polarizations

$$\mathbf{P}_i = \mathbf{P}_{1i} + \mathbf{P}_{2i}.$$

POLARIZATION

Enter the components (**x**, **y**, and **z** for 3D components for example) of the **Polarization, first wave** $\mathbf{P}_{i1}$ (SI unit: C/m^2) and, for the bidirectional formulation, the **Polarization, second wave** $\mathbf{P}_{i2}$ (SI unit: C/m^2).

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. Add more **Initial Values** nodes from the **Physics** toolbar.

INITIAL VALUES

Enter values or expressions for the initial values of the components of the **Electric field envelope, first wave** $\mathbf{E1}$ and **Electric field envelope, second wave** $\mathbf{E2}$ (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.

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 More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

Field Continuity

To display the **Field Continuity** boundary condition, click the **Show More Options** button (👁) and select **Advanced Physics Options**. This boundary condition is available when the **Number of directions** in the Wave Vectors section is set to **Unidirectional**. The boundary condition assures that the tangential electric and magnetic fields components are continuous on interior boundaries, also when the **Phase, first wave ϕ_1** expression is discontinuous at the boundary.

> This boundary condition should not be used if there are different materials on the two sides of the boundary. In that case, there will be reflections at the boundary and it is probably more appropriate to use **Bidirectional** propagation.

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options**.

	Optical Ring Resonator Notch Filter: Application Library path **Wave_Optics_Module/Waveguides_and_Couplers/optical_ring_resonator**

Magnetic Field

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

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

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 $\mathbf{H}_{02}$**.

Matched Boundary Condition

Use the **Matched Boundary Condition** to make a boundary transparent for a wave (or waves) with the phase(s) as prescribed in the Wave Vectors settings of The Electromagnetic Waves, Beam Envelopes Interface settings.

Since the **Wave Vectors** settings are taken into account, this boundary condition is low-reflecting also for waves propagating with a large angle to the normal of the boundary, in contrast to the Scattering Boundary Condition, where the scattered beam should propagate almost in parallel to the boundary normal to be efficiently absorbed.

If there is an incident field, a Reference Point subnode can be added by right-clicking the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. The Reference Point subnode redefines the incident field to be expressed as

$$\mathbf{E}_0 e^{-jk(\mathbf{k}\,\cdot\,(\mathbf{r}-\mathbf{r}_{ref}))} ,$$

where $\mathbf{r}_{\text{ref}}$ is a reference point determined as the average point from the point selection in the Reference Point subnode.

In 2D axisymmetry, the default subnode Symmetry Axis Reference Point is available. This subnode defines a reference point at the intersection between the symmetry axis and the Scattering boundary condition's boundary selection.

MATCHED BOUNDARY CONDITION

When Wave Vectors is set to bidirectional, select an **Input wave** — **First wave** (the default) or **Second wave** — that the specified input electric field is associated with.

Select an **Incident field** to specify whether the input wave is specified by an **Electric field**, a **Magnetic field**, or a **Gaussian Beam**.

When **Wave Vectors** is set to unidirectional, the default setting is **No incident field**. For this case, it is assumed that the wave is propagating out from the domain the selected boundaries are adjacent to.

Specify the **Incident electric field envelope** $\mathbf{E}_0$ (SI unit: V/m) or **Incident magnetic field envelope** $\mathbf{H}_0$ (SI unit: A/m), depending on the **Incident field** selected. Notice that you only specify the envelope factor of the incident electric or magnetic field. The envelope function is internally multiplied by the phase function, as specified in the Wave Vectors settings, to form the complete incident electric or magnetic fields.

If the **Incident field** is set to **Gaussian beam**, select an **Input quantity: Electric field amplitude** (the default) or **Power**. If the **Input quantity** is **Electric field amplitude**, enter the component expressions for the **Gaussian beam electric field amplitude** $\mathbf{E}_{\text{g0}}$ (SI unit: V/m). If the **Input quantity** is set to **Power**, enter the **Input power** (SI unit: W in 2D axisymmetry and 3D and W/m in 2D) and the component expressions for the **Gaussian beam non-normalized electric field amplitude** $\mathbf{E}_{\text{g0}}$ (SI unit: V/m). Also edit the **Beam radius** w_0 (SI unit: m) and the **Distance to focal plane** p_0 (SI unit: m). The default values are `((10*2)*pi)/ewfd.k0` and `0` m, respectively. The optical axis for the Gaussian beam is defined by a line including a reference point on the feature selection with a direction specified by the propagation direction for the **Input wave** (or $\mathbf{k}_1$ for the unidirectional formulation). By default, the reference point is the average position for the feature selection. However, by adding a Reference Point subnode any available point (or the average of several selected points) on the feature selection can be used as

the reference point. The focal plane for the Gaussian beam is located the **Distance to focal plane** p_0 from the reference point in the propagation direction for the **Input wave** (or $\mathbf{k}_1$ for the unidirectional formulation).

When Wave Vectors is set to bidirectional, if no scattered field is expected, select the **No scattered field** check box. This prevents COMSOL from returning spurious solutions that otherwise could appear between boundaries with unconstrained scattered fields.

ℚ	For more information about the Gaussian beam theory, see Gaussian Beams as Background Fields and Input Fields.

▥	*Gaussian Beam Incident at the Brewster Angle*: Application Library path **Wave_Optics_Module/Optical_Scattering/brewster_interface**.
	Total Internal Reflection: Application Library path **Wave_Optics_Module/ Waveguides_and_Couplers/total_internal_reflection**.

Scattering Boundary Condition

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

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

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

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

The field $\mathbf{E}_0$ is the incident plane wave that travels in the direction $\mathbf{k}$. In addition, to an incident plane wave, $\mathbf{E}_0$ can also be the electric field distribution for a Gaussian beam that propagates 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 physics 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.

A Reference Point subnode can be added by right-clicking the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. The **Reference Point** subnode redefines the incident field to be expressed as

$$\mathbf{E}_0 e^{-jk(\mathbf{k} \cdot (\mathbf{r} - \mathbf{r}_{ref}))},$$

where $\mathbf{r}_{ref}$ is a reference point determined as the average point from the point selection in the **Reference Point** subnode.

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

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

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

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

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**), the magnetic field (**Wave given by H field**), or as a Gaussian beam (**Gaussian beam**).

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**.

If the **Incident field** is set to **Gaussian beam**, select an **Input quantity: Electric field amplitude** (the default) or **Power**. If the **Input quantity** is **Electric field amplitude**, enter the component expressions for the **Gaussian beam electric field amplitude** $\mathbf{E}_{g0}$ (SI unit: V/m). If the **Input quantity** is set to **Power**, enter the **Input power** (SI unit: W in 2D axisymmetry and 3D and W/m in 2D) and the component expressions for the **Gaussian beam non-normalized electric field amplitude** $\mathbf{E}_{g0}$ (SI unit: V/m). Also edit the **Beam radius** w_0 (SI unit: m) and the **Distance to focal plane** p_0 (SI unit: m). The default values are ((10*2)*pi)/ewfd.k0 and 0 m, respectively. The optical axis for the Gaussian beam is defined by a line including a reference point on the feature selection with a direction specified by the propagation direction for the selected **Input wave** (for unidirectional propagation it is the direction of $\mathbf{k}_1$). By default, the reference point is the average position for the feature selection. However, by adding a Reference Point subnode any available point (or the average of several selected points) on the feature selection can be used as the reference point. The focal plane for the Gaussian beam is located the **Distance to focal plane** p_0 from the reference point in the propagation direction for the selected **Input wave**.

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

- For **Cylindrical wave** enter coordinates for the **Source point** $\mathbf{r}_0$ (SI unit: m) and **Source axis direction** $\mathbf{r}_{axis}$ (dimensionless).

- For **Spherical wave** enter coordinates for the **Source point** $\mathbf{r}_0$ (SI unit: m).

| | For more information about the Gaussian beam theory, see Gaussian Beams as Background Fields and Input Fields. |

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.

SURFACE CURRENT DENSITY

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** $\mathbf{J}_{s02}$.

The Electromagnetic Waves, Boundary Elements Interface

The **Electromagnetic Waves, Boundary Elements (ebem)** interface ( ) is used to solve for time-harmonic electromagnetic field distributions. This interface is found under the **Wave Optics** branch ( ) when adding a physics interface,. The formulation is based on the boundary element method (BEM) and is available in 2D and 3D. The physics interface solves the vector Helmholtz equation for piecewise-constant material properties and uses the electric field as dependent variable.

The interface is fully multiphysics enabled and can be coupled seamlessly with the physics interfaces that are based on the finite element method (FEM). This approach allows modeling in a FEM-BEM framework, exploiting the strength of both formulations to the fullest. The BEM-based interface is especially well suited for radiation and scattering problems.

The advantage of the boundary element method is that only boundaries need to be meshed and the degrees of freedom (DOFs) solved for are restricted to the boundaries. This introduces some clear ease-of-use for handling complex geometries. However, the BEM technique results in fully populated or dense matrices that need dedicated numerical methods. The BEM method is so to speak more expensive per DOF than the FEM method, but has fewer DOFs. Assembling and solving these can be very demanding. This means that when solving models of small and medium size, The Electromagnetic Waves, Frequency Domain Interface will often be faster, than solving the same problem with the BEM interface. The challenge for the FEM interface is to set up open boundaries, for example, using Perfectly Matched Layers (PMLs) in an efficient way. When the geometries are complex or two structures are far apart, large air domains need to be meshed. This costs a lot on the computational side as the frequency is increased.

For large models (problems that contain many wavelengths, at high frequency or for large domains) the stabilized formulation option (see Stabilization) ensures efficient convergence at the cost of some additional degrees of freedom. For low to medium frequencies (small to medium models), running without stabilization is more efficient. The stabilized formulation only gives a benefit in computing time for the large models.

For this physics interface, the maximum mesh element size should well resolve the complex electric field on the boundaries. Thus, if the wave propagates tangentially to

the boundary, the maximum mesh size should be a fraction of the wavelength. However, if the wave propagates essentially in the normal direction to the boundary, the maximum mesh element size can be larger. The physics interface supports the Frequency Domain and Wavelength Domain study types. The Frequency Domain and Wavelength Domain study types are used for source driven simulations for a single frequency/wavelength or a sequence of frequencies/wavelengths.

| ! | In the *COMSOL Multiphysics Reference Manual* see the Theory for the Boundary Elements PDE section for more information about the Boundary Element Method. |

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Wave Equation, Electric**, **Perfect Electric Conductor**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Electromagnetic Waves, Boundary Elements** to select physics features from the context menu.

| 📝 | If both The Electromagnetic Waves, Frequency Domain Interface and The Electromagnetic Waves, Boundary Elements Interface are available, the Electric Field Coupling node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**. |
| 📝 | The Electromagnetic Waves, Frequency Domain Interface and The Electromagnetic Waves, Boundary Elements Interface can also be coupled by using the same name for the dependent variable for both interfaces. Then Electric Field Coupling is not needed. How to set the name for the dependent variable is described in the Dependent Variables section. |

Physics-Controlled Mesh

The physics-controlled mesh only defines mesh settings for the boundaries. It is controlled from the **Settings** window for the **Mesh** node (if the **Sequence type** is **Physics-controlled mesh**). In the table in the **Physics-Controlled Mesh** section, find the physics interface in the **Contributor** column and select or clear the check box in the **Use** column on the same row for enabling (the default) or disabling contributions from the physics interface to the physics-controlled mesh.

When the **Use** check box for the physics interface is selected, this invokes a parameter for the maximum mesh element size in free space. The physics-controlled mesh automatically scales the maximum mesh element size as the wavelength changes in different dielectric and magnetic regions.

When the **Use** check box is selected for the physics interface, in the section for the physics interface below the table, choose one of the four options for the **Maximum mesh element size control parameter** — **From study** (the default), **User defined**, **Frequency**, or **Wavelength**. When **From study** is selected, $1/5$ of the vacuum wavelength from the highest frequency defined in the study step is used for the maximum mesh element size. For the option **User defined**, enter a suitable **Maximum element size in free space**. For example, $1/5$ of the vacuum wavelength or smaller. When **Frequency** is selected, enter the highest frequency intended to be used during the simulation. The maximum mesh element size in free space is $1/5$ of the vacuum wavelength for the entered frequency. For the **Wavelength** option, enter the smallest vacuum wavelength intended to be used during the simulation. The maximum mesh element size in free space is $1/5$ of the entered wavelength.

The maximum mesh element size in dielectric media is equal to the maximum mesh element size in vacuum divided by the square root of the product of the relative permittivity and permeability.

!	In the *COMSOL Multiphysics Reference Manual* see the Physics-Controlled Mesh section for more information about how to define the physics-controlled mesh.

SETTINGS

The **Label** is the default physics interface name.

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

The default **Name** (for the first physics interface in the model) is ebem.

DOMAIN SELECTION

From the **Selection** list, select any of the options — **Manual**, **All domains**, **All voids**, or **All domains and voids** (the default). The geometric entity list displays the selected domain

entity numbers. Edit the list of selected domain entity numbers using the selection toolbar buttons to the right of the list or by selecting the geometric entities in the **Graphics** window. Entity numbers for voids can be entered by clicking the Paste (⬚) button in the selection toolbar and supplying the entity numbers in the dialog box. The entity number for the infinite void is 0, and finite voids have negative entity numbers.

Selections can also be entered using the **Selection List** window, available from the **Windows** menu on the **Home** toolbar.

> For more information about making selections, see Working with Geometric Entities in the *COMSOL Multiphysics Reference Manual*.

COMPONENTS

This section is available for 2D components.

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 **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.

FORMULATION

From the **Formulation** list, select whether to solve for the **Full field** (the default) or the **Scattered field**.

For **Scattered field** select a **Background wave type** according to the following table:

TABLE 3-2: BACKGROUND WAVE TYPE BASED ON COMPONENT DIMENSION

COMPONENT	BACKGROUND WAVE TYPE
2D	User defined (default), Gaussian beam
3D	User defined (default), Gaussian beam, Linearly polarized plane wave

> 📋 Notice that only metallic PEC scatterers are supported in the scattered field formulation. Thus, in the scattered field formulation, there can only be one Wave Equation, Electric node, representing the infinite void.

User Defined

Enter the component expressions for the **Background electric field $\mathbf{E}_b$** (SI unit: V/m). The entered expressions must be differentiable.

> 📋 Notice that expressions including coupling operators are not differentiable and cannot be used as background fields.

Gaussian Beam

For **Gaussian beam** select the **Gaussian beam type** — **Paraxial approximation** (the default) or **Plane wave expansion**.

When selecting **Paraxial approximation**, the Gaussian beam background field is a solution to the paraxial wave equation, which is an approximation to the Helmholtz equation solved for by the **Electromagnetic Waves, Boundary Elements (ebem)** interface. The approximation is valid for Gaussian beams that have a beam radius that is much larger than the wavelength. Since the paraxial Gaussian beam background field is an approximation to the Helmholtz equation, for tightly focused beams, you can get a nonzero scattered field solution, even if you do not have any scatterers. The option **Plane wave expansion** means that the electric field for the Gaussian beam is approximated by an expansion of the electric field into a number of plane waves. Since each plane wave is a solution to the Helmholtz equation, the plane wave expansion of the electric field is also a solution to the Helmholtz equation. Thus, this option can be used also for tightly focused Gaussian beams.

If the beam spot radius is smaller than the wavelength, evanescent plane waves need to be included in the expansion. The evanescent waves decay exponentially in the propagation direction, why it only makes sense to model such tightly focused beams if the focal plane coincides with the input boundary. If the focal plane is located inside

the modeled domain, the field can be dominated by the exponentially decaying evanescent waves. Those waves can have a very high field strength before the focal plane even though they only provide a small contribution to the field at the focal plane.

For **Plane wave expansion** select **Wave vector distribution type** — **Automatic** (the default) or **User defined**. For **Automatic** also check **Allow evanescent waves**, to include evanescent waves in the plane wave expansion. For **User defined** also enter values for the **Wave vector count** $N_\mathbf{k}$ (the default value is 13) and **Maximum transverse wave number** $k_{\mathrm{t,max}}$ (SI unit: rad/m, default value is `(2*(sqrt(2*log(10))))/ebem.w0`). Use an odd number for the **Wave vector count** $N_\mathbf{k}$ to make sure that a wave vector pointing in the main propagation direction is included in the plane-wave expansion. The **Wave vector count** $N_\mathbf{k}$ specifies the number of wave vectors that will be included per transverse dimension. So for 3D the total number of wave vectors will be $N_\mathbf{k} \cdot N_\mathbf{k}$.

📝   Evanescent waves are included in the plane wave expansion if the **Maximum transverse wave number** $k_{\mathrm{t,max}}$ is larger that the specified **Wave number** k. When the **Wave vector distribution type** is set to **Automatic**, evanescent waves are included in the expansion if the **Allow evanescent waves** check box is selected.

A plane wave expansion with a finite number of plane waves included will make the field periodic in the plane orthogonal to the main propagation direction. If the separation between the transverse wave vector components, given by $2k_{\mathrm{t,max}}/(N_\mathbf{k} - 1)$, is too small, replicas of the Gaussian beam background field can appear. To avoid that, increase the value for the **Wave vector count** $N_\mathbf{k}$.

The number of plane waves included in the expansion can be quite large, especially for 3D. For instance, using the default settings, $2 \cdot 13 \cdot 13 = 338$ plane waves will be included (the factor 2 accounts for the two possible polarizations for each wave vector). Thus, initializing the plane-wave expansion for the Gaussian beam background field can take some time in 3D.

For more information about the Gaussian beam theory, see Gaussian Beams as Background Fields and Input Fields.

Define the Gaussian beam background field using the parameters below:

• Select a **Beam orientation**: **Along the x-axis** (the default), **Along the y-axis**, or for 3D components, **Along the z-axis**.

- Enter a **Beam radius** w_0 (SI unit: m). The default is $20\pi/$ebem.k$_0$ m (10 vacuum wavelengths).

- Enter a **Focal plane along the axis** p_0 (SI unit: m). The default is 0 m.

- Select an **Input quantity: Electric field amplitude** (the default) or **Power**.

- Enter the component expressions for the **Transverse background electric field amplitude, Gaussian beam** $\mathbf{E}_{T\mathrm{bg}0}$ (SI unit: V/m) if the **Input quantity** is **Electric field amplitude**. Notice that this is the transverse Gaussian beam amplitude in the focal plane. When the **Gaussian beam type** is set to **Paraxial approximation** the background field is always orthogonal (transverse) to **Beam orientation**. However, when the **Gaussian beam type** is set to **Plane wave expansion**, the background field amplitude can also have a component in the propagation direction. Specify here only the field amplitude components that are orthogonal to the propagation direction. COMSOL computes automatically the component in the propagation direction, if needed.

- If the **Input quantity** is set to **Power**, enter the **Input power** (SI unit: W in 2D axisymmetry and 3D and W/m in 2D) and the component expressions for the **Non-normalized transverse electric field amplitude, Gaussian beam** $\mathbf{E}_{T\mathrm{bg}0}$ (SI unit: V/m).

- Enter a **Wave number** k (SI unit: rad/m). The default is ebem.k$_0$ rad/m. The wave number must evaluate to a value that is the same for all the domains the scattered field is applied to. Setting the **Wave number** k to a positive value, means that the wave is propagating in the positive x-, y-, or z-axis direction, whereas setting the **Wave number** k to a negative value means that the wave is propagating in the negative x-, y-, or z-axis direction.

Linearly Polarized Plane Wave

The initial background wave is predefined as $\mathbf{E}_0 = \exp(-jk_x x)\mathbf{z}$. This field is transformed by three successive rotations along the roll, pitch, and yaw angles, in that order. For a graphic representation of the initial background field and the definition of the three rotations compare with Figure 3-1 below.

- Enter an **Electric field amplitude** E_0 (SI unit: V/m). The default is 1 V/m.

- Enter a **Roll angle** (SI unit: rad), which is a right-handed rotation with respect to the $+x$ direction. The default is 0 rad, corresponding to polarization along the $+z$ direction.

- Enter a **Pitch angle** (SI unit: rad), which is a right-handed rotation with respect to the $+y$ direction. The default is 0 rad, corresponding to the initial direction of propagation pointing in the $+x$ direction.

- Enter a **Yaw angle** (SI unit: rad), which is a right-handed rotation with respect to the +z direction.

- Enter a **Wave number** k (SI unit: rad/m). The default is ebem.k$_0$ rad/m. The wave number must evaluate to a value that is the same for the domains the scattered field is applied to.

Figure 3-6: Schematic of the directions for the wave vector **k**, *the electric field* **E**$_0$, *and the roll, pitch and yaw rotations. The top image represents an initial wave propagating in the x direction with a polarization along the z direction.*

STABILIZATION

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog box.

For large models (problems that contain many wavelengths, at high frequency or for large domains) enable the **Use stabilization** option (enable by default) to ensure efficient convergence at the cost of some additional degrees of freedom.

When **Use stabilization** is selected, a text field for the **Stabilization parameter** is enabled with the default value sqrt(abs(ebem.k[m])). This is a parameter that should scale inversely with the wavelength. The default gives good performance in most cases.

FAR-FIELD APPROXIMATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog box.

> For more information about the **Far Field Approximation** settings, see Far-Field Approximation Settings in the *COMSOL Multiphysics Reference Manual*.

When **Use far-field approximation for matrix assembly** is selected, a text field for the **Minimum near field range in vacuum for preconditioning** is enabled with the default value `((2*pi)/ebem.k0)/10` (one tenth of a wavelength). For problems having a wide distribution of mesh element sizes, including mesh elements that are much smaller than the wavelength, a smaller value for this parameter may make the iterative solver convergence faster.

Using a smaller value for this parameter, may make problems having a large distribution of mesh element sizes, including mesh elements that are much smaller than the wavelength, converge faster with the iterative solver. However, a smaller value use more memory.

QUADRATURE

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box.

📄	For more information about the **Quadrature** settings, see Quadrature in the *COMSOL Multiphysics Reference Manual*.

DISCRETIZATION

From the **Electric field/Flux field** list, choose from predefined options for the boundary element discretization order for the electric field variable and the flux field (magnetic field) variable, respectively. The predefined options represent the suitable combinations of element orders such as **Quadratic/Linear** (the default). For more information about the **Discretization** section, see Settings for the Discretization Sections in the *COMSOL Multiphysics Reference Manual*.

DEPENDENT VARIABLES

The dependent variables (field variables) are for the **Electric field 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.

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

The Electromagnetic Waves, Frequency Domain Interface has these domain, boundary, edge, point, and pair nodes and subnodes. The nodes are listed in alphabetical order and are available from the **Physics** ribbon toolbar (Windows users),

Physics context menu (Mac or Linux users), or by right-clicking to access the context menu (all users).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

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. The Far-Field Calculation condition is described in this section.

The following features are also available and described for The Electromagnetic Waves, Frequency Domain Interface:

- Electric Field
- Perfect Magnetic Conductor
- Perfect Electric Conductor
- Surface Current Density

	In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Wave Equation, Electric

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

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

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

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

where c_0 is the speed of light in vacuum.

MATERIAL PROPERTIES

The default **Relative permittivity** ε_r, **Relative permeability** μ_r, and **Electrical conductivity** σ take values **From material**. For **User defined** enter a value or expression in the field.

> ⚠ Notice that the boundary element method is based on the availability of an analytic Green's function for the domain. Thus, the **Relative permittivity**, the **Relative permeability**, and the **Electrical conductivity** must all evaluate to constant values for each **Wave Equation, Electric** node.

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. Add additional **Initial Values** nodes from the **Physics** toolbar.

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.

Far-Field Calculation

Use a **Far-Field Calculation** to select boundaries corresponding to a single closed surface surrounding all radiating and scattering objects. By default, all exterior boundaries of the simulation domain are selected. Also use this feature to indicate symmetry planes and symmetry cuts applied to the geometry.

FAR-FIELD CALCULATION

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

Select, as applicable for your model, the **Symmetry in the x=0 plane**, **Symmetry in the y=0 plane**, or **Symmetry in the z=0 plane** check boxes to simplify the calculation of the far-field variable. The symmetry planes must 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 includes the parts of the geometry that are not in the model for symmetry reasons in the far-field analysis.

Electric Field Coupling

The **Electric Field Coupling** multiphysics node assures continuity of the electric potential across boundaries between The Electromagnetic Waves, Frequency Domain Interface and The Electromagnetic Waves, Boundary Elements Interface. The **Electric Field Coupling** node is available from the **Multiphysics** menu in the **Physics** toolbar or by right-clicking the **Multiphysics Couplings** node in **Model Builder**, if both The Electromagnetic Waves, Frequency Domain Interface and The Electromagnetic Waves, Boundary Elements Interface are available.

> 📋 The Electromagnetic Waves, Frequency Domain Interface and The Electromagnetic Waves, Boundary Elements Interface can also be coupled by using the same name for the dependent variable for both interfaces. Then **Electric Field Coupling** is not needed. How to set the name for the dependent variable is described in the Dependent Variables section.

BOUNDARY SELECTION

Select **Manual** or **All boundaries** from the **Selection** list. Make additional edits to the list of boundary entity numbers using the **Selection** toolbar buttons. When **All boundaries** is selected from the **Selection** list, the boundaries exterior to the Electromagnetic Waves, Frequency Domain interface that intersect the exterior boundaries to the Electromagnetic Waves, Boundary Elements interface are available in the boundary entity number list.

COUPLED INTERFACES

Select **Electromagnetic Waves, Frequency Domain** as **Primary interface** and **Electromagnetic Waves, Boundary Elements** as **Secondary interface**.

Theory for the Electromagnetic Waves Interfaces

The Electromagnetic Waves, Frequency Domain Interface, The Electromagnetic Waves, Beam Envelopes Interface, and The Electromagnetic Waves, Transient Interface theory is described in this section:

- Introduction to the Physics Interface Equations
- Frequency Domain Equation
- Time Domain Equation
- Curl Elements
- Eigenfrequency Calculations
- Gaussian Beams as Background Fields and Input Fields

Introduction to the Physics Interface Equations

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

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

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

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

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

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

Frequency Domain Equation

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

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

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

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

The first of these, based on the electric field is used in The Electromagnetic Waves, Frequency Domain Interface.

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

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

The wave number in vacuum k_0 is defined by

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

where c_0 is the speed of light in vacuum.

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

For The Electromagnetic Waves, Beam Envelopes 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 $\mathbf{k}_1$,

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_1(\mathbf{r})\exp(-j\varphi_1(\mathbf{r})).$$

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

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

where

$$\mathbf{k}_1 = \nabla \varphi_1 .$$

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

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

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

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

> **!** Limiting the electric field component solved for to the out-of-plane component for TE modes requires that the medium is homogeneous; that is, μ and ε are constant. When solving for the in-plane electric field components for TM modes, μ can 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 Electromagnetic Waves, Beam Envelopes Interface)

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

The wave equation is thereby rewritten as

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

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

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

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

and the wave equation is expressed as

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

where φ 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 nonzero 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 nondiagonal only in the xy-plane. μ_r denotes a 2-by-2 tensor, and ε_{rzz} and σ_{zz} are the relative permittivity and conductivity in the z direction.

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 nondiagonal only in the rz-plane. μ_r denotes a 2-by-2 tensor, while $\varepsilon_{r\varphi\varphi}$ and $\sigma_{\varphi\varphi}$ are the relative permittivity and conductivity in the φ direction.

INTRODUCING LOSSES IN THE FREQUENCY DOMAIN

Electric Losses

The frequency domain equations allow for several ways of introducing electric losses. Finite conductivity results in a *complex permittivity*,

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

The conductivity gives rise to ohmic losses in the medium.

A more general approach is to use a complex permittivity,

$$\varepsilon_c = \varepsilon_0(\varepsilon' - j\varepsilon'')$$

where ε' is the real part of ε_r, and all losses (dielectric and conduction losses) are given by ε''. The dielectric loss model can also single out the losses from finite conductivity (so that ε'' only represents dielectric losses) resulting in:

$$\varepsilon_c = \varepsilon_0\left(\varepsilon' - j\left(\frac{\sigma}{\omega\varepsilon_0} + \varepsilon''\right)\right)$$

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

$$\varepsilon_c = \varepsilon_0\varepsilon'(1 - j\tan\delta)$$

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

For the physics interfaces in the Wave Optics Module, the refractive index is the default electric displacement field model . In materials where μ_r is 1, the relation between the complex refractive index

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

and the complex relative permittivity is

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

that is

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

The inverse relations are

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

The parameter κ represents a damping of the electromagnetic wave. When specifying the refractive index, conductivity is not allowed as an input.

In the physics and optics literature, the time harmonic form is often written with a minus sign (and "i" instead of "j"):

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

This makes an important difference in how loss is represented by complex material coefficients like permittivity and refractive index, that is, by having a positive imaginary part rather than a negative one. Therefore, material data taken from the literature might have to be conjugated before using it 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 can be combined with any electric loss model except refractive index.

Time Domain Equation

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

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

This is the equation used by The Electromagnetic Waves, Transient Interface. It is suitable for the simulation of nonsinusoidal waveforms or nonlinear 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 nonzero vector potential component, namely in the z direction. The magnetic field lies in the modeling plane. Thus the equation in the time domain can be simplified to a scalar equation for A_z:

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

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

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

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

Axisymmetric Hybrid-Mode Waves

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

Axisymmetric TM Waves

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

Axisymmetric TE Waves

A TE wave has only a vector potential component in the φ direction, and the magnetic field lies in the modeling plane. Given these constraints, the 3D equation can be simplified to a scalar equation for A_φ. To write the fields in this form, it is also required

that ε_r and μ_r are nondiagonal only in the *rz*-plane. μ_r denotes a 2-by-2 tensor, while $\varepsilon_{r\varphi\varphi}$ and $\sigma_{\varphi\varphi}$ are the relative permittivity and conductivity in the φ direction.

Curl 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 physics interface conditions, which specify that the normal components of the electric fields and the tangential components of the 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 interface uses *curl elements*, which do not have this limitation. The curl element is also named as vector element or edge element.

The solution obtained when using curl elements also better fulfills the divergence conditions $\nabla \cdot \mathbf{D} = 0$ and $\nabla \cdot \mathbf{B} = 0$ than when using Lagrange elements.

📄 | For more information about the curl element, read the blog post:
www.comsol.com/blogs/what-is-the-curl-element-and-why-is-it-used/

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 density condition and the electric field condition, can specify a source in the eigenvalue problem. These conditions are available as a general tool to specify arbitrary expressions between the **H** field and the **E** field. Avoid specifying solution-independent sources for these conditions because the eigenvalue solver ignores them anyway.

Using the default parameters for the eigenfrequency study, it might find a large number of false eigenfrequencies, which are almost zero. This is a known consequence of using vector elements. To avoid these eigenfrequencies, change the parameters for the eigenvalue solver in the **Study Settings**. Adjust the settings so that the solver searches for eigenfrequencies closer to the lowest eigenfrequency than to zero.

Gaussian Beams as Background Fields and Input Fields

When solving for the scattered field, the background wave type can be set to a predefined Gaussian beam from within the **Settings** of The Electromagnetic Waves, Frequency Domain Interface. Additionally, Gaussian beams can be specified as the input field for the Scattering Boundary Condition and the Matched Boundary Condition.

In the paraxial approximation, the field for a Gaussian beam propagating along the z-axis is

$$\mathbf{E}_G(x, y, z) = \mathbf{E}_{G0}\frac{w_0}{w(z)}\exp\left[-\frac{\rho^2}{w^2(z)} - jkz - jk\frac{\rho^2}{2R(z)} + j\eta(z)\right],$$

where w_0 is the beam radius, p_0 is the focal plane on the z-axis, $\mathbf{E}_{G0}$ is the Gaussian beam electric field amplitude, and the spot radius for different positions along the propagation axis is given by

$$w(z) = w_0\sqrt{1 + \left(\frac{z - p_0}{z_0}\right)^2}.$$

$$R(z) = (z - p_0)\left[1 + \left(\frac{z_0}{z - p_0}\right)^2\right]$$

defines the radius of curvature for the phase of the field and the so-called Gouy phase shift is given by

$$\eta(z) = \mathrm{atan}\left(\frac{z - p_0}{z_0}\right).$$

The equations above are expressed using the Rayleigh range z_0 and the transverse coordinate ρ, defined by

$$z_0 = \frac{kw_0^2}{2}, \rho^2 = x^2 + y^2.$$

Note that the time-harmonic ansatz in COMSOL Multiphysics is $e^{j\omega t}$, and with this convention, the beam above propagates in the +z direction. The equations are modified accordingly for beams propagating along the other coordinate axes.

The field for a Gaussian beam is defined in a similar way for 2D components. In the particular case where the beam propagates along the x-axis, the field is defined as

$$\mathbf{E}_G(x, y) \; = \; \mathbf{E}_{G0} \sqrt{\frac{w_0}{w(x)}} \exp\left[-\frac{y^2}{w^2(x)} - jkx - jk\frac{y^2}{2R(x)} + j\frac{\eta(x)}{2} \right].$$

For a beam propagating along the y-axis, the coordinates x and y are interchanged.

Notice that the expressions above for Gaussian beams are not solutions to the Helmholtz equation, but to the so called paraxial approximation of the Helmholtz equation. This means that these equations become less accurate the smaller the spot radius is and should not be used when the spot radius is of the same size as or smaller than the wavelength.

To circumvent the problem that the paraxial approximation formula is not a solution to the Helmholtz equation, a plane wave expansion can be used to approximate a Gaussian beam *background field*. Since each plane wave is a solution to Helmholtz equation, also the expansion is a solution to Helmholtz equation.

The plane wave expansion approximates the Gaussian distribution in the focal plane

$$\mathbf{E}_{b,\,\mathrm{Gauss}}(\mathbf{r}) \; = \; E_0 \exp\left(-\frac{x^2 + y^2}{w_0^2} \right) \mathbf{e} \; =$$

$$\sum_{l=-L}^{L} \sum_{m=-M}^{M} \sum_{n=0}^{1} a_{lmn} \mathbf{u}_n(\mathbf{k}_{lm}) \exp(-i\mathbf{k}_{lm} \cdot \mathbf{r}),$$

where the beam is assumed to be propagating in the z direction, the focal plane is spanned by the x and y coordinates, $\mathbf{e}$ is the unit magnitude transverse polarization in the focal plane, l and m denote the indices for the wave vectors, the index n accounts for the two polarizations per wave vector $\mathbf{k}_{lm}$, a_{lmn} is the amplitude, $\mathbf{u}_n(\mathbf{k}_{lm})$ is the unit magnitude polarization, and $\mathbf{r}$ is the position vector.

Multiplying with the conjugate of the exponential factor above and the polarization factor $\mathbf{u}_n(\mathbf{k}_{lm})$ and applying a surface integral over the entire focal plane allows us to extract the amplitudes as

$$a_{lmn} \; = \; \frac{E_0 w_0^2 (\mathbf{e} \cdot \mathbf{u}_n(\mathbf{k}_{lm}))}{4\pi} \exp\left(-\frac{k_{t,\,lm}^2 w_0^2}{4} \right),$$

where $k_{t,lm}$ is the magnitude of the transverse wave vector component.

Theory for the Electromagnetic Waves, Time Explicit Interface

The Electromagnetic Waves, Time Explicit Interface theory is described in this section:

- The Equations
- In-Plane E Field or In-Plane H Field
- Fluxes as Dirichlet Boundary Conditions
- Absorbing Layers

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-Ampère'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

$$\mathbf{D} = \varepsilon_0\mathbf{E} + \mathbf{P}$$
$$\mathbf{B} = \mu_0(\mathbf{H} + \mathbf{M}) \qquad (3\text{-}4)$$
$$\mathbf{J} = \sigma\mathbf{E}$$

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 $4\pi\cdot10^{-7}$ 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

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

where χ_e is the electric susceptibility (which can be a scalar or a second-rank tensor). Similarly, the magnetization is directly proportional to the magnetic field, or

$$\partial \mathbf{M} / \partial \mathbf{H} = \chi_m \text{ and } \mathbf{M} = \chi_m \mathbf{H}$$

where χ_m is the magnetic susceptibility.

As a consequence, for linear materials, the constitutive relations in Equation 3-4 can be written as

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

Here, $\varepsilon = \varepsilon_0 \varepsilon_r$ and $\mu = \mu_0 \mu_r$ are the permittivity and permeability of the material. The relative permittivity ε_r and the relative permeability μ_r are usually scalar properties but these can be second-rank symmetric (Hermitian) tensors for a general anisotropic material.

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

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

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

(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'S EQUATIONS

In order to accommodate Maxwell's equations in the coefficients for the Wave Form PDE interface in the form

$$d_a \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \Gamma(\mathbf{u}) = \mathbf{f}$$

the curl of a vector is written in divergence form as

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

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

Maxwell's equations in 3D

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

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

are then accommodated to the Wave Form PDE as

$$d_E \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) = \mathbf{f}$$

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

with the "mass" coefficients

$$d_E = \varepsilon_0 \varepsilon_r \text{ and } d_H = \mu_0 \mu_r$$

the "flux" terms

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

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

THE LAX-FRIEDRICHS FLUX PARAMETERS

When using SI units (or other) for the electromagnetic fields and material properties, the Lax-Friedrichs flux parameter is not dimensionless and must have units of $\tau_E = 1/(2Z)$ for Ampère'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

$$\varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) = -\sigma \cdot \mathbf{E}$$

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

(3-7)

with the flux terms

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

(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 Ampère's 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

$$\varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) = -\sigma \mathbf{E}$$

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

(3-9)

with the flux terms

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

(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 Ampère's law, and two scalar $\tau_H = Z/2$ for Faraday's law, where Z is the impedance of a vacuum.

Fluxes as Dirichlet Boundary Conditions

Consider Maxwell's equations in 3D

$$\varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) = -\sigma \mathbf{E}$$

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

with the flux terms

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

and the divergence on $\Gamma_E(\mathbf{H})$ and $\Gamma_H(\mathbf{E})$ applied row-wise.

For Ampère'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 $\mathbf{n} \times \mathbf{E} = Z\mathbf{H}$.

Absorbing Layers

The Electromagnetic Waves, Time Explicit Interface includes so-called *absorbing layers,* also often referred to as *sponge layers.* The layers work by combining three techniques: a scaling system, filtering, and simple nonreflecting conditions. For a review of the method see, for example, Ref. 1.

The layers are set up by adding the **Absorbing Layer** under the **Definitions** node. This adds a special scaled system. The scaling effectively slows down the propagating waves and ensures that they hit the outer boundary in the normal direction. For the Absorbing Layer domain selection, add an additional Wave Equations feature, mark the **Activate** check box under the Filter Parameters section, and enter filter parameters. Filtering attenuates and filters out high-frequency components of the wave. Finally, at the outer boundary of the layer add a simple Scattering Boundary Condition condition, which will work well to remove all remaining waves as normal incidence has been ensured.

	For more detailed information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

For the **Absorbing Layers** select the **Type** (Cartesian, cylindrical, spherical, or user defined) under the **Geometry** section. Enter values for the **Physical Width** and **Pole Distance** under the **Scaling** section.

> For more detailed on the **Geometry** and **Scaling** see the Infinite Elements, Perfectly Matched Layers, and Absorbing Layers in the *COMSOL Multiphysics Reference Manual.*

For the layers to work optimally the filter should not be too aggressive. Moreover, the scaled coordinates in the layer domain should also vary smoothly. To inspect the scaled system you can plot the coordinate variables x_absorb_ab1, y_absorb_ab1, and z_absorb_ab1. Using the absorbing layers with the three combined techniques will enable the reduction of spurious reflections by a factor between 100 and 1000 compared to the incident amplitude.

> For an example of a filter parameter combination that can be used for a Wave Equations feature on an **Absorbing Layer** domain selection see the Filter Parameters section for the Wave Equations feature.

Reference

1. P.G. Petropoulos, L. Zhao, and A.C. Cangellaris, "A Reflectionless Sponge Layer Absorbing Boundary Condition for the Solution of Maxwell's Equations with High-Order Staggered Finite Difference Schemes," *J. Comp. Phys.*, vol. 139, pp. 184–208, 1998.

4

Heat Transfer Interfaces

This chapter describes The Laser Heating Interface found under the **Heat Transfer>Electromagnetic Heating** branch ( ) when adding a physics interface.

See The Heat Transfer Interfaces and The Joule Heating Interface in the *COMSOL Multiphysics Reference Manual* for other Heat Transfer interface and feature node settings.

The Laser Heating Interface

The **Laser Heating** interface ( ) is used to model electromagnetic heating for systems and devices where the electric field amplitude varies slowly on a wavelength scale. This multiphysics interface adds an Electromagnetic Waves, Beam Envelopes interface and a Heat Transfer in Solids interface. The multiphysics couplings add the electromagnetic losses from the electromagnetic waves as a heat source, and the electromagnetic material properties can depend on the temperature. The modeling approach is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale.

Combinations of frequency-domain modeling for the Electromagnetic Waves, Beam Envelopes interface and stationary modeling for the Heat Transfer in Solids interface, called frequency-stationary and, similarly, frequency-transient modeling, are supported in 2D and 3D. For these study types, the coupling between the frequency-domain modeling of the electromagnetic part and the stationary or transient modeling of the heat transfer part is bidirectional: the electromagnetic part defines a heat source for the heat transfer problem and the temperature distribution from the heat transfer problem can influence the material properties in the electromagnetic problem.

If the coupling is unidirectional — that is, if the electromagnetic problem does not depend on the temperature — **Frequency-Stationary, One-Way Electromagnetic Heating** and **Frequency-Transient, One-Way Electromagnetic Heating** studies can be added from the **Model Wizard**. These studies give, respectively, a **Frequency Domain** study step for the electromagnetic problem followed by a **Stationary** or **Time Dependent** study step for the heat transfer problem.

When a predefined **Laser Heating** interface is added from the **Heat Transfer>Electromagnetic Heating** branch ( ) of the **Model Wizard** or **Add Physics** window, **Electromagnetic Waves, Beam Envelopes** and **Heat Transfer in Solids** interfaces are added to the Model Builder.

In addition, a **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Electromagnetic Heat Source**.

On the Constituent Physics Interfaces
The Electromagnetic Waves, Beam Envelopes interface computes electric and magnetic field distributions for systems and devices where the field amplitude varies slowly on a wavelength scale. The physics interface can be used efficiently for unidirectional and bidirectional propagation of electromagnetic beams. In this physics

interface, the electric field is factored into a product of a slowly varying envelope function (slowly on the scale of a wavelength) and a rapidly varying phase function. The phase function is a priori prescribed, so the physics interface solves the time-harmonic wave equation for the slowly varying envelope function.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation. A Heat Transfer in Solids model is active by default on all domains. All functionality for including other domain types, such as a fluid domain, is also available. The temperature equation defined in solid domains corresponds to the differential form of Fourier's law that may contain additional contributions like heat sources.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURE

When physics interfaces are added using the predefined couplings, for example **Laser Heating**, specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Electromagnetic Waves, Beam Envelopes** and **Heat Transfer in Solids** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. You can add the **Electromagnetic Heating** coupling feature, but no modified settings are included.

	Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

TABLE 4-1: MODIFIED SETTINGS FOR A LASER HEATING INTERFACE

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Electromagnetic Waves, Beam Envelopes	No changes.

TABLE 4-1: MODIFIED SETTINGS FOR A LASER HEATING INTERFACE

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Solids	No changes.
Electromagnetic Heating	The **Domain Selection** is the same as that of the participating physics interfaces.
	The **Boundary Selection** is the same as the exterior and interior boundaries of the **Domain Selection** of the participating physics interfaces.
	The corresponding **Electromagnetic Waves, Beam Envelopes** and **Heat Transfer in Solids** interfaces are preselected in the Coupled Interfaces section (described in the *COMSOL Multiphysics Reference Manual*).

A side effect of adding physics interfaces one at a time is that four study types — Frequency-Stationary; Frequency-Transient; Frequency-Stationary, One-Way Electromagnetic Heating; and Frequency-Transient, One-Way Electromagnetic Heating — are not available for selection until *after* at least one coupling feature is added. In this case, it is better to initially not add any study at all, then add the coupling features to the **Multiphysics** node, and lastly, open the **Add Study** window and add a study sequence below the **Preset Studies for Selected Multiphysics** heading.

PHYSICS INTERFACES AND COUPLING FEATURE

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Coupling Feature
The Electromagnetic Heating coupling feature node is described for The Joule Heating Interface in the *COMSOL Multiphysics Reference Manual*.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

	In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- The available physics features for The Electromagnetic Waves, Beam Envelopes Interface are listed in the section Domain, Boundary, Edge, and Point Nodes for the Electromagnetic Waves, Beam Envelopes Interface.

- The available physics features for The Heat Transfer Interfaces are listed in the section Feature Nodes for the Heat Transfer in Solids Interface in the *COMSOL Multiphysics Reference Manual.*

5

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.

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

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

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

Index