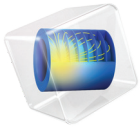


Wave Optics Module

User's Guide

6.4

COMSOL

Wave Optics Module User's Guide

Version: COMSOL 6.4

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 on the COMSOL Access page at www.comsol.com/support/case. Useful links:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- 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
- Learning Center: www.comsol.com/support/learning-center

Part number: CM023501

Contents

Chapter 1: Introduction

Chapter 2: Wave Optics Modeling

Chapter 3: Wave Optics Interfaces

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

These topics are included in this section:

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
Heat Transfer				
Electromagnetic Heating				
Laser Heating[1]		—	3D, 2D, 2D axisymmetric	frequency–stationary; frequency–transient; frequency–stationary, one-way electromagnetic heating; frequency–transient, one-way electromagnetic heating

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
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
Electromagnetic Waves, FEM-BEM[1]			3D, 2D	frequency domain, wavelength domain

[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.

COMPARING THE ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN AND THE ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS INTERFACES

Both the Electromagnetic Waves, Frequency Domain and the Electromagnetic Waves, Boundary Elements interfaces solve the time-harmonic Maxwell's equations. For the Frequency Domain interface, the dependent variable is the electric field in the domains. However, for the Electromagnetic Waves, Boundary Elements interface the dependent variable is the electric field on the boundaries. This physics interface solves for the dependent variable using the Boundary Element Method (BEM).

Typically, much fewer degrees of freedom (unknown variables) will be solved for when using the Boundary Elements interface. However, when solving the problem using BEM, the matrices are no longer sparse. So, even though the degrees of freedom will be much less when using the Boundary Elements interface, the problem may still require a large amount of memory during the solving process.

The Boundary Element Method is based on the knowledge of the Green's function for the vector Helmholtz equation. Since we only know the Green's function, if the domain material is homogeneous, the Electromagnetic Waves, Boundary Elements interface can only be used for problems where the materials are homogeneous or piece-wise homogeneous.

Some applications where it could be advantageous to use the Electromagnetic Waves, Boundary Elements interface are:

- Applications where there is a structure transmitting waves very far from the structure receiving the waves.
- Scattering problems with several scattering elements, potentially located far apart.
- Applications where a wave or beam is propagating through several simple elements, located far apart.

When discussing above about elements located far apart, it is meant that the elements are located many wavelengths apart. If the Electromagnetic Waves, Frequency Domain interface would be used for solving such problems, there will be a large number of degrees of freedom in the domain mesh. On the other hand, for the Electromagnetic Waves, Boundary Elements interface, only the size and complexity of the boundaries, not the separation between the elements, determine the number of degrees of freedom required to represent the problem.

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.

The physics interfaces automatically add default plots of the electric field to the **Results** branch. In addition, depending on the physics interface settings and the added features, other plots are often automatically added, like background field plots when the scattered-field formulation is used, far-field radiation pattern plots, optical cross section plots, plots of the port mode fields, *etc*.

Where Do I Access the Documentation and Application Libraries?

A number of online 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 on 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.
- In the main toolbar, click the **Help** (?) button.
- From the main menu, select **Help** > **Help**.

Opening the Documentation Window

Mac

Linux

To open the **Documentation** window:

- Press Ctrl+F1.
- In 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.

Opening the Application Libraries Window

To open the **Application Libraries** window ():

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 will be 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/product-update
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/forum
Events	www.comsol.com/events
COMSOL Application Gallery	www.comsol.com/models
COMSOL Video Gallery	www.comsol.com/videos
Learning Center	www.comsol.com/support/learning-center
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 Application Library also illustrate the different aspects of the simulation process. Topics include Preparing for Wave Optics Modeling, Simplifying Geometries, and Scattered Field Formulation.

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

OPTICS

The 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 Electromagnetic Waves, FEM-BEM Interface allows to build hybrid FEM-BEM models, where the boundary element method (BEM) is used to compute the electric fields outside the finite element method (FEM) domains.

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 Wave Optics Module Application Library also illustrate the different aspects of the simulation process.

In this chapter:

- 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

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 unduly 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 fields on the two parallel boundaries. 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.
- *Cyclic symmetry* — There is a phase shift between the fields on the two sector boundaries. The phase shift is determined by an azimuthal mode number and the sector geometry angle.

Periodic boundary conditions must have compatible meshes. This can be done automatically by enabling **Physics-controlled mesh** in the **Sequence Type** section 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
- Periodic Boundary Conditions

Modeling Periodic Structures

A periodic structure is defined by the periodic unit cell. This unit cell is then repeated infinitely in one or two dimensions.

Periodic structures appear in two different modeling scenarios — driven problems and non-driven problems. For driven problems, a port is used for launching an incident wave. What is of interest here is to calculate how much of that input wave that is transmitted, reflected, absorbed, and diffracted to higher diffraction orders. For non-driven problems there is no excitation source. A typical simulation is to prescribe a certain wave vector that defines a phase relationship between the fields on the opposing parallel boundaries of the unit cell and then perform an eigenfrequency simulation to find all the frequencies and mode fields that can satisfy the prescribed periodic phase relationship. By sweeping the magnitude and direction of the wave vector, it is possible to generate band structure diagrams for the particular periodic cell.

To simplify the simulation of driven periodic problems, the Wave Optics Module has a dedicated Periodic Structure domain condition. The condition automatically adds the required ports and periodic conditions as subnodes to the Periodic Structure node. The selections for the ports and the periodic conditions are handled automatically, after the user has assigned the selection for the excitation port.

The Periodic Structure selection defines the periodic unit cell. The reference direction $\mathbf{a}_1$ is defined from edge and point selections in the Periodic Structure and its Reference Direction subnode. Given the first reference direction $\mathbf{a}_1$, the second reference direction $\mathbf{a}_2$ is calculated from

$$\mathbf{a}_2 = \mathbf{a}_0 \times \mathbf{a}_1 ,$$

where $\mathbf{a}_0$ is the periodic structure axis direction that is pointing in the direction from the passive port toward the excitation port. Thus, the axis direction equals the normal direction of the excited port.

Given the reference directions, the wave vector for the incident plane wave is given by

$$\mathbf{k}_{\text{inc}} = k[\sin\alpha_1(\cos\alpha_2\mathbf{a}_1 + \sin\alpha_2\mathbf{a}_2) - \cos\alpha_1\mathbf{a}_0] ,$$

where k is the material wavenumber at the excitation port and α_1 and α_2 are the elevation and azimuth angles, respectively. The elevation angle α_1 is the angle between the wave vector $\mathbf{k}_{\text{inc}}$ and the periodic structure axis ($\mathbf{k}_{\text{inc}} \cdot \mathbf{a}_0 \leq 0$, as α_1 is in the range

from 0 to $\pi/2$ radians). For normal incidence, this angle is zero. The azimuth angle α_2 is the angle between the first reference direction $\mathbf{a}_1$ and the projection of the incident wave vector $\mathbf{k}_{\text{inc}}$ on the plane spanned by $\mathbf{a}_1$ and $\mathbf{a}_2$ (the port planes).

The first reference direction $\mathbf{a}_1$ also defines the direction of the first primitive vector of the periodic unit cell. That is,

$$\mathbf{b}_1 = b_1\mathbf{a}_1 .$$

The length b_1 is obtained from a selection of the excited port edges. In 3D, for parallelogram-like unit cells, the second primitive vector $\mathbf{b}_2$ is defined from the unit cell edges that are not parallel to $\mathbf{b}_1$. Furthermore, $\mathbf{b}_2$ is defined such that

$$\mathbf{b}_1 \times \mathbf{b}_2 = c\,\mathbf{a}_0, c > 0 . \tag{2-1}$$

For hexagonal unit cells, three edge vectors are defined from three consecutive hexagon edges:

- $\mathbf{v}_1$ is defined from the hexagon edge selected by the user.
- $\mathbf{v}_2$ is defined by the hexagon edge that starts where the edge that defines $\mathbf{v}_1$ ends.
- $\mathbf{v}_3$ is defined by the hexagon edge that starts where the edge that defines $\mathbf{v}_2$ ends.

From these three vectors, the first primitive cell vector is defined by

$$\mathbf{b}_1 = \mathbf{v}_1 - \mathbf{v}_3$$

and the second primitive cell vector is defined from

$$\mathbf{b}_2 = \mathbf{v}_1 + \mathbf{v}_2 ,$$

assuming that Equation 2-1 applies. If the cross product does not fulfill Equation 2-1, the expressions for $\mathbf{b}_1$ and $\mathbf{b}_2$ are swapped.

From the primitive vectors for the unit cell, the reciprocal lattice vectors can be calculated as

$$\mathbf{G}_1 = \frac{2\pi}{V}\mathbf{b}_2 \times \mathbf{a}_0$$

and

$$\mathbf{G}_2 = \frac{2\pi}{V}\mathbf{a}_0 \times \mathbf{b}_1 ,$$

where the volume V is given by

$$V = \mathbf{a}_0 \cdot \mathbf{b}_1 \times \mathbf{b}_2 \, .$$

In 2D, there is periodicity in only one direction. Thus, the second primitive vector has an infinite length

$$\mathbf{b}_2 = b_2 \mathbf{a}_2, \; b_2 \to \infty \, .$$

Thereby, the first reciprocal vector becomes

$$\mathbf{G}_1 = \frac{2\pi}{b_1} \mathbf{a}_1$$

and, consequently,

$$\mathbf{G}_2 = \frac{2\pi}{b_2} \mathbf{a}_2 = 0 \, .$$

The Periodic Structure and its Reference Direction subnode define the periodic structure variables in Table 2-1. The variables should be prefixed with the physics interface tag. So, to evaluate the x-component of the unit cell axis, write `ewfd.axisx` if the physics tag is `ewfd`.

TABLE 2-1: PERIODIC STRUCTURE VARIABLES

VARIABLE	DESCRIPTION	AVAILABLE COMPONENT
axisx	Unit cell axis, x-component	2D, 3D
axisy	Unit cell axis, y-component	2D, 3D
axisz	Unit cell axis, z-component	2D, 3D
aUnit1x	First reference direction, x-component	2D, 3D
aUnit1y	First reference direction, y-component	2D, 3D
aUnit1z	First reference direction, z-component	2D, 3D
aUnit2x	Second reference direction, x-component	2D, 3D
aUnit2y	Second reference direction, y-component	2D, 3D
aUnit2z	Second reference direction, z-component	2D, 3D
bx	Primitive vector, x-component	2D
by	Primitive vector, y-component	2D
bz	Primitive vector, z-component	2D
b1x	First primitive vector, x-component	3D
b1y	First primitive vector, y-component	3D

TABLE 2-1: PERIODIC STRUCTURE VARIABLES

VARIABLE	DESCRIPTION	AVAILABLE COMPONENT
b1z	First primitive vector, z-component	3D
b2x	Second primitive vector, x-component	3D
b2y	Second primitive vector, y-component	3D
b2z	Second primitive vector, z-component	3D
Gx	Reciprocal lattice vector, x-component	2D
Gy	Reciprocal lattice vector, y-component	2D
Gz	Reciprocal lattice vector, z-component	2D
G1x	First reciprocal lattice vector, x-component	3D
G1y	First reciprocal lattice vector, y-component	3D
G1z	First reciprocal lattice vector, z-component	3D
G2x	Second reciprocal lattice vector, x-component	3D
G2y	Second reciprocal lattice vector, y-component	3D
G2z	Second reciprocal lattice vector, z-component	3D

When modeling periodic structures only using Port features, similar variables for reference directions, axes, primitive cell vectors, and reciprocal lattice vectors are defined. However, their definitions and names are slightly different from the ones discussed here. For more information, see Additional Variables for Periodic Structure Calculations.

Additional variables, related to the plane-wave modes used in periodic structure calculations, are described in

- S-Parameter Variables
- Jones Vector Variables

To learn how to use the Periodic Structure node to solve driven 3D periodic problems, see *Hexagonal Grating*: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/hexagonal_grating**.

For a 2D driven periodic problem, see *Plasmonic Wire Grating*: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/plasmonic_wire_grating**.

For a non-driven example, see *Band-Gap Analysis of a Photonic Crystal*: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/bandgap_photonic_crystal**.

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:

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

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 it 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:

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

The $\mathbf{E}_{\text{scattered}}$ 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 $\mathbf{E}_{\text{correction}}$ 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 $\mathbf{E}_{\text{cancellation}}$ component will be nonzero and equal to $-\mathbf{E}_{\text{background}}$ 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.

A default **Electric Field, Background** plot of the instantaneous background electric field norm is automatically added to any model that uses the scattered–field formulation, except when the **Background wave type** is set to **Linearly polarized plane wave** for 2D Axisymmetry. For this case, a default plot is added of a component of the linearly polarized plane wave background field.

TABLE 2-2: SCATTERED-FIELD FORMULATION VARIABLES

VARIABLE	DESCRIPTION	AVAILABLE COMPONENT
normEb	Background electric field norm	2D, 2D Axisymmetric, 3D
normEbi	Instantaneous background electric field norm	2D, 2D Axisymmetric, 3D
Ebx	Background electric field, x-component	2D, 3D
Eby	Background electric field, y-component	2D, 3D
Ebz	Background electric field, z-component	2D, 2D Axisymmetric, 3D
Ebr	Background electric field, r-component	2D Axisymmetric
Ebphi	Background electric field, z-component	2D Axisymmetric
relEx	Relative electric field, x-component	2D, 3D
relEy	Relative electric field, y-component	2D, 3D
relEz	Relative electric field, z-component	2D, 2D Axisymmetric, 3D
relEr	Relative electric field, r-component	2D Axisymmetric
relEphi	Relative electric field, phi-component	2D Axisymmetric

TABLE 2-3: BACKGROUND FIELD INTENSITY AND POWER VARIABLES

VARIABLE	DESCRIPTION	BACKGROUND WAVE TYPE	SELECTION
Ib	Background plane wave intensity	Linearly polarized plane wave, Circularly polarized plane wave	Global
Ib	Background Gaussian beam intensity	Gaussian beam	Domain
Ib0	Background Gaussian beam maximum intensity	Gaussian beam	Global
Pb0	Background Gaussian beam total power	Gaussian beam	Global

Table 2-2 lists the most important variables related to the electric field, defined only for the scattered-field formulation. Table 2-3 describes the intensity and power variables, defined for different background wave types.

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 (Far-Field Domain) 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 (Far-Field Domain) 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`.

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, the reference direction from which the angle is measured and the normal to the plane the far field is computed for can also 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 or 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 the far-field norm are automatically added to any model that uses far-field calculation features combined with a far-field domain feature. If a model is driven by ports, an antenna realized gain plot is added instead. An **Export Expressions** subfeature is also added under the default **Radiation Pattern** node to include elevation and azimuth angle information. This subfeature is useful when performing **Add Plot Data to Export** from the context menu.

TABLE 2-4: VARIABLES AND OPERATORS GENERATED BY FAR FIELD.

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

TABLE 2-4: VARIABLES AND OPERATORS GENERATED BY FAR FIELD.

	NAME	DESCRIPTION	AVAILABLE COMPONENT
	mRCS3D	Monostatic radar cross section	3D
	Efarphi	Far-field variable, phi component	2D Axisymmetric, 3D
	Efartheta	Far-field variable, theta component	3D
	EfarRHCP	Far-field variable, right-hand circular polarization	3D
	EfarLHCP	Far-field variable, left-hand circular polarization	3D
	rGainEfar	Far-field realized gain	2D Axisymmetric, 3D
	rGaindBEfar	Far-field realized gain, dBi	2D Axisymmetric, 3D
	rGainLHCP	Far-field realized gain, left-hand circular polarization	3D
	rGaindBLHCP	Far-field realized gain, left-hand circular polarization, dBc	3D
	rGainRHCP	Far-field realized gain, right-hand circular polarization	3D
	rGaindBRHCP	Far-field realized gain, right-hand circular polarization, dBc	3D
	TRP	Total radiated power	2D Axisymmetric, 3D
	TRPdB	Total radiated power, dB	2D Axisymmetric, 3D
	bRCS2D	Bistatic radar cross section per unit length	2D
	mRCS2D	Monostatic radar cross section per unit length	2D
	maxD	Maximum directivity[1]	2D Axisymmetric, 3D
	maxDdB	Maximum directivity, dB[1]	2D Axisymmetric, 3D
	maxGain	Maximum gain[1]	2D Axisymmetric, 3D
	maxGaindB	Maximum gain, dB[1]	2D Axisymmetric, 3D
	maxRGain	Maximum realized gain[1]	2D Axisymmetric, 3D
	maxRGaindB	Maximum realized gain, dB[1]	2D Axisymmetric, 3D
Functions[4]	norm3DEfar	3D far-field norm[2]	2D Axisymmetric
	normdB3DEfar	3D far-field norm, dB[2]	2D Axisymmetric

TABLE 2-4: VARIABLES AND OPERATORS GENERATED BY FAR FIELD.

	NAME	DESCRIPTION	AVAILABLE COMPONENT
	af3	Uniform three dimensional array factor[3]	2D Axisymmetric, 3D
	af2	Uniform two dimensional array factor[3]	2D
	afhex	Hexagonal uniform array factor[3]	3D
	normEfar_opt	Far-field norm	2D, 2D Axisymmetric, 3D
	normdBEfar_opt	Far-field norm, dB	2D, 2D Axisymmetric, 3D
	Efarx_opt	Far-field variable, x component	2D, 3D
	Efary_opt	Far-field variable, y component	2D, 3D
	Efarz_opt	Far-field variable, z component	2D, 2D Axisymmetric, 3D
	Efarr_opt	Far-field variable, r component	2D Axisymmetric
	EIRP_opt	Effective isotropic radiated power	2D Axisymmetric, 3D
	EIRPdB_opt	Effective isotropic radiated power, dB	2D Axisymmetric, 3D
	axialRatio_opt	Axial ratio	3D
	axialRatiodB_opt	Axial ratio, dB	3D
	Efarphi_opt	Far-field variable, phi component	2D Axisymmetric, 3D
	Efartheta_opt	Far-field variable, theta component	3D
	rGainEfar_opt	Far-field realized gain	2D Axisymmetric, 3D
	rGaindBEfar_opt	Far-field realized gain, dBi	2D Axisymmetric, 3D
	maxRGain_opt	Maximum realized gain[1]	2D Axisymmetric, 3D
	maxRGaindB_opt	Maximum realized gain, dB[1]	2D Axisymmetric, 3D

[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**

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

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 a 0-degree azimuthal angle. This is equivalent to plotting this variable on the *xz*-plane. Similarly, `norm3DEfar_TE12(pi/2)` is the evaluation at a 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

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

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

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

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 2-5: INPUT ARGUMENTS OF UNIFORM 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

TABLE 2-5: INPUT ARGUMENTS OF UNIFORM ARRAY FACTOR OPERATOR.

ARGUMENT	DESCRIPTION	UNIT	COMPONENT
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

Hexagonal uniform array factor When array elements are distributed not on a conventional rectangular grid but on a triangular grid forming a hexagonal shape array, the two-dimensional `afhex` operator is available in a 3D model.

TABLE 2-6: INPUT ARGUMENTS OF HEXAGONAL UNIFORM ARRAY FACTOR OPERATOR.

ARGUMENT	DESCRIPTION	UNIT	COMPONENT
np	Number of elements along diagonal axis	Dimensionless	3D
dx	Distance between array elements along *x*-axis	Wavelength	3D
dy	Distance between array elements along *y*-axis	Wavelength	3D

For an odd number of array elements, the number of elements on the diagonal axis is also an odd number. For instance, when `np` is set to 15, the number of elements, n, on a hexagonal array edge, $(\mathrm{np}+1)/2$, evaluates to 8 and the total number of array elements, $3n^2 - 3n + 1$, is then 169.

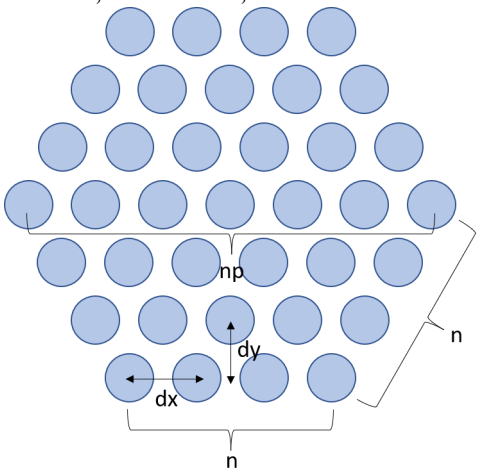


Figure 2-1: Configuration for an odd number hexagonal array.

For an even number of array elements, the number of elements on the diagonal axis is also an even number. The shape of the hexagon is uneven and configured with two edge sizes. The number of elements on the smaller edge n is $\mathsf{np}/2$ and the other edge has $n + 1$ elements. When np is set to 8, the total number of array elements, $3n^2$, is 48.

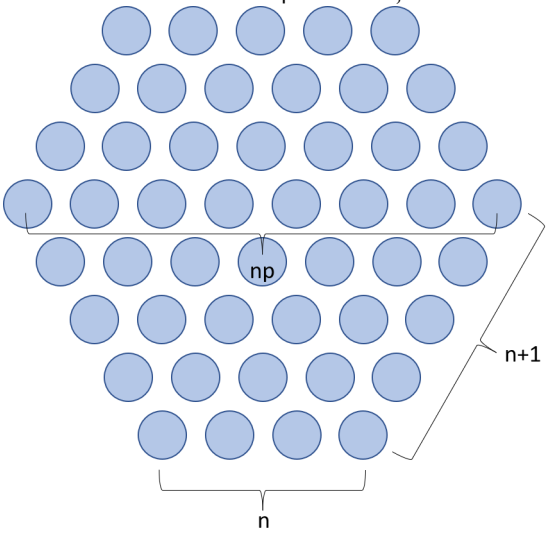


Figure 2-2: Configuration for an even number hexagonal array.

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

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

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

The antenna gain is

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

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

Modeling with Far-Field Calculations (Far-Field Domain, Inhomogeneous)

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

Scatterer on Substrate: Application Library path **Wave_Optics_Module/Optical_Scattering/scatterer_on_substrate**

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

The Electromagnetic Waves, Frequency Domain interface supports far-field analysis for a structure consisting of a superstrate and a substrate. To define the far-field variables use theFar-Field Calculation (Far-Field Domain, Inhomogeneous) node. Select the domain or domain group for the superstrate, the substrate, and the far-field calculation. The Far-Field Domain, Inhomogeneous, the Superstrate, the Substrate, and the Far-Field Calculation (Far-Field Domain, Inhomogeneous) nodes get their selections automatically. The selection in the Far-Field Calculation (Far-Field Domain, Inhomogeneous) node is the boundaries where the algorithm integrates the near field.

Enter a name for the far electric field. For each variable name entered, the software generates 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, if 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`.

The magnitude of the far field and its value in dB are also generated as the variables `normEfar` and `normdBEfar`, respectively.

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, the reference direction from which the angle is measured and the normal to the plane the far field is computed for can also be specified. For 3D **Radiation Pattern** plots, you also specify an expression for the surface color.

Default **Radiation Pattern** plots using **Polar Plot Group** of the far-field norm are automatically added to any model that uses far-field calculation features combined with a far-field domain, inhomogeneous feature.

TABLE 2-7: VARIABLES AND OPERATORS GENERATED BY FAR FIELD.

	NAME	DESCRIPTION	AVAILABLE COMPONENT
Variables	normEfar	Far-field norm	3D
	normdBEfar	Far-field norm, dB	3D
	Efarx	Far-field variable, x component	3D
	Efary	Far-field variable, y component	3D
	Efarz	Far-field variable, z component	3D
	axialRatioInhg	Axial ratio	3D
	axialRatiodBInhg	Axial ratio, dB	3D
	Efarphi	Far-field variable, phi component	3D

TABLE 2-7: VARIABLES AND OPERATORS GENERATED BY FAR FIELD.

	NAME	DESCRIPTION	AVAILABLE COMPONENT
	Efartheta	Far-field variable, theta component	3D
	maxDInhg	Maximum directivity	3D
	maxDdBInhg	Maximum directivity, dB	3D

3D example with a Polar Plot Group *Scatterer on Substrate*: Application Library path **Wave_Optics_Module/Optical_Scattering/scatterer_on_substrate**

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

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

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

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

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

$$\nabla \cdot \mathbf{J} = -\frac{\partial \rho}{\partial t} \tag{2-3}$$

Constitutive Relations

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

$$\begin{aligned} \mathbf{D} &= \varepsilon_0 \mathbf{E} + \mathbf{P} \\ \mathbf{B} &= \mu_0(\mathbf{H} + \mathbf{M}) \\ \mathbf{J} &= \sigma \mathbf{E} \end{aligned} \tag{2-4}$$

where ε_0 and μ_0 are the permittivity and permeability of vacuum, and σ is the electric conductivity. The constants ε_0 and μ_0 are available in COMSOL Multiphysics as predefined physical constants. In the SI unit system, ε_0 has an approximate value of $1/(36\pi)\cdot 10^{-9}$ F/m; μ_0 has an approximate value of $4\pi\cdot 10^{-7}$ 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:

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

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

ENERGY- AND COENERGY-BASED MAGNETIC CONSTITUTIVE RELATIONS

Another general approach to defining the constitutive relation of a magnetic material stems from the conservation of magnetic energy density and magnetic coenergy density. For nonlinear magnetic materials without hysteresis, these densities are defined as follows:

$$dW_m = \int_0^{\mathbf{B}} \mathbf{H} \cdot d\mathbf{B}'$$

$$dW'_m = \int_0^{\mathbf{H}} \mathbf{B} \cdot d\mathbf{H}'$$

In this context, the magnetic field $\mathbf{H}$ can be derived from the magnetic flux density $\mathbf{B}$ according to

$$\mathbf{H} = \frac{\partial dW_m}{\partial \mathbf{B}}$$

Equivalently, the magnetic flux density can be derived from the magnetic field as

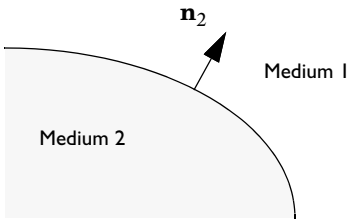
$$\mathbf{B} = \frac{\partial dW_m'}{\partial \mathbf{H}}$$

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

$$\begin{aligned} \mathbf{n}_2 \cdot (\mathbf{D}_1 - \mathbf{D}_2) &= \rho_s \\ \mathbf{n}_2 \times (\mathbf{E}_1 - \mathbf{E}_2) &= \mathbf{0} \\ \mathbf{n}_2 \cdot (\mathbf{B}_1 - \mathbf{B}_2) &= 0 \\ \mathbf{n}_2 \times (\mathbf{H}_1 - \mathbf{H}_2) &= \mathbf{J}_s \end{aligned} \tag{2-5}$$

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 2-5, 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 electric conductivity and thus no internal electric field. Otherwise, it would produce an infinite current density according to the third fundamental constitutive relation. At an interface between a dielectric and a perfect conductor, the boundary conditions for the $\mathbf{E}$ and $\mathbf{D}$ fields are simplified. 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:

$$\begin{aligned} -\mathbf{n}_2 \cdot \mathbf{D}_2 &= \rho_s \\ -\mathbf{n}_2 \times \mathbf{E}_2 &= 0 \\ -\mathbf{n}_2 \cdot \mathbf{B}_2 &= 0 \\ -\mathbf{n}_2 \times \mathbf{H}_2 &= \mathbf{J}_s \end{aligned} \tag{2-6}$$

Potentials

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

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

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

Electromagnetic Energy

The electric and magnetic energies are defined as

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

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

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

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

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

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

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

Material Properties

Until now, there has only been a formal introduction of the constitutive relations. These seemingly simple relations can be quite complicated at times. There are four

main groups of materials 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 electric 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*.

Frequency Domain

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.

FREQUENCY DOMAIN FORMULATION

With the use of phasors, Maxwell's equations can be formulated in the frequency domain. One of the main advantages 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 $j\omega$. In this way, linear differential equations are converted to algebraic equations that are much easier to solve.

For sake of simplicity, when writing variables and equations in the frequency domain, the tilde is dropped from the variable denoting the phasor. However, it is important to remember that the field that has been calculated is a phasor and not a physical field. Also note that the phasor $\tilde{\mathbf{E}}(\mathbf{r})$ is visualized in the plot as $\mathrm{Re}(\tilde{\mathbf{E}}(\mathbf{r}))$ 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.

The frequency domain formulation is only applicable for equations linear in the fields and for one specific frequency. In general, it cannot be used with materials whose properties depend on the fields themselves.

SIGN CONVENTION

The time dependency of a time-harmonic field can be written in two ways: $e^{j\omega t}$ or $e^{-j\omega t}$. In COMSOL Multiphysics, the former sign convention $e^{j\omega t}$ is used.

The time-harmonic sign convention dictates the sign convention for some material coefficients in the frequency domain, such as complex permittivity, complex permeability, and complex refractive index. For example, in the dielectric loss model, the complex relative permittivity is defined as

$$\varepsilon_r = \varepsilon' - j\varepsilon''$$

where ε' and ε'' are two material parameters that are named as the *real part* and *imaginary part* of the relative permittivity, respectively. Inserting the complex relative permittivity in the frequency domain Maxwell–Ampère's law gives

$$\nabla \times \mathbf{H} = j\omega\varepsilon_0\varepsilon'\mathbf{E} + (\omega\varepsilon_0\varepsilon'' + \sigma)\mathbf{E}$$

where the term $\omega\varepsilon_0\varepsilon''$ contributes to energy loss that is indistinguishable from the loss quantified by the electric conductivity σ. With the sign convention used here, a positive material parameter ε'' corresponds to a loss although the imaginary part of the complex number $\varepsilon' - j\varepsilon''$, evaluated as `imag(`$\varepsilon' - j\varepsilon''$`)`, is negative. See Introducing Losses in the Frequency Domain for details.

Special Calculations

In this section:

S-Parameter Calculations

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

$$\mathbf{E}_{\mathrm{T}c} = \mathbf{E}_{\mathrm{T}1} + \sum_{i=1} S_{i1}\mathbf{E}_{\mathrm{T}i} \ ,$$

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

$$\mathbf{E}_{\mathrm{T}c} = \sum_{i=1} S_{i1}\mathbf{E}_{\mathrm{T}i}$$

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 S

$$\frac{1}{2}\int_S (\mathbf{E}_{\mathrm{T}c} - \mathbf{E}_{\mathrm{T}1}) \times \mathbf{H}^*_{\mathrm{T}i} \cdot \mathbf{n} dS = \frac{1}{2}\sum_{i=1} S_{i1}\int_S \mathbf{E}_{\mathrm{T}i} \times \mathbf{H}^*_{\mathrm{T}i} \cdot \mathbf{n} dS \ .$$

Since the mode fields are orthogonal, the S-parameter is given by

$$S_{i1} = \frac{\int_S (\mathbf{E}_{\mathrm{T}c} - \mathbf{E}_{\mathrm{T}1}) \times \mathbf{H}_{\mathrm{T}i}^* \cdot \mathbf{n} dS}{\int_S \mathbf{E}_{\mathrm{T}i} \times \mathbf{H}_{\mathrm{T}i}^* \cdot \mathbf{n} dS} .$$

Here, $\mathbf{H}_{\mathrm{Ti}}$ is the tangential magnetic mode field for mode i. This expression is valid for all reciprocal modes (modes that have the same propagation constant for both the incoming and outgoing wave).

For pure TE, TM, and TEM modes, the expression above is reduced to

$$S_{11} = \frac{\int_{\text{port 1}} ((\mathbf{E}_{\mathrm{T}c} - \mathbf{E}_{\mathrm{T}1}) \cdot \mathbf{E}_{\mathrm{T}1}) dA_1}{\int_{\text{port 1}} (\mathbf{E}_{\mathrm{T}1} \cdot \mathbf{E}_{\mathrm{T}1}^*) dA_1}$$

$$S_{21} = \frac{\int_{\text{port 2}} (\mathbf{E}_{\mathrm{T}c} \cdot \mathbf{E}_{\mathrm{T}2}^*) dA_2}{\int_{\text{port 2}} (\mathbf{E}_{\mathrm{T}2} \cdot \mathbf{E}_{\mathrm{T}2}^*) dA_2}$$

$$S_{31} = \frac{\int_{\text{port 3}} (\mathbf{E}_{\mathrm{T}c} \cdot \mathbf{E}_{\mathrm{T}3}^*) dA_3}{\int_{\text{port 3}} (\mathbf{E}_{\mathrm{T}3} \cdot \mathbf{E}_{\mathrm{T}3}^*) dA_3}$$

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,

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

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

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

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

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

where Z_{TE} is the wave impedance

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

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

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

TM Waves

For TM waves it holds that

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

where Z_{TM} is the wave impedance

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

and ε is the permittivity. The power flow then becomes

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

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:

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

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.

The Stratton–Chu formula is based on the assumption that the Green's function for the vector Helmholtz equation for the far-field domain is known. The Green's function used in COMSOL is based on the assumption that the far-field domain material is homogeneous. Thus, if inhomogeneous materials are used in the far-field domain an error will be displayed.

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 $|\mathbf{E}_p|^2$.

For more information about to calculate the far field in certain points, see the blog 2 Methods for Simulating Radiated Fields in COMSOL Multiphysics®

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

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 on how 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:

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

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.

In addition to the S-parameter variables, variables are also added for reflectances, transmittances, and absorptance, as shown in the table below:

NAME	EXPRESSION	DESCRIPTION
`ewfd.Rport_N`	`abs(ewfd.SN1)^2`	Reflectance, port `N`
`ewfd.Tport_N`	`abs(ewfd.SN1)^2`	Transmittance, port `N`
`ewfd.Rtotal`	$\sum_i$`ewfd.Rport_i`	Total reflectance
`ewfd.Ttotal`	$\sum_i$`ewfd.Tport_i`	Total transmittance
`ewfd.RTtotal`	`ewfd.Rtotal+ewfd.Ttotal`	Total reflectance and transmittance
`ewfd.Atotal`	`1-ewfd.RTtotal`	Absorptance

In the table above, `ewfd` is the tag for the physics interface. This tag can be different for different physics interface instances. `N` is the name of the port (a number).

In the expressions for the reflectance and the transmittance, it is assumed that the excitation port has the name `1`.

The definition of `ewfd.Atotal` is actually the sum of all the volume and surface losses in the physics. It is only when **Activate slit condition on interior port** is disabled or **Slit type** is set to **PEC-backed** that `ewfd.Atotal = 1-ewfd.RTtotal`. Otherwise, when **Activate slit condition on interior port** is enabled and **Slit type** is set to **Domain-backed**, some of the radiation that reaches the port might not match any of the port mode fields. Thus, in this case you have the following power conservation relation,

`ewfd.RTtotal + ewfd.Atotal + ewfd.Lsca = 1`,

where `ewfd.Lsca` represents scattering loss — radiation that does not match any of the port mode fields.

For more information about the scattering loss variable `ewfd.Lsca`, see Coupling and Loss Calculations.

Periodic ports and diffraction order ports represent the plane-wave radiation propagating in certain directions, defined by diffraction order mode numbers. In 3D, two mode numbers are used, for example m and n, whereas in 2D only one mode number, n, is needed. Each periodic port (or diffraction order port) represent one of two possible polarization directions. Thus, for 3D ports and diffraction order ports, the following variables are available:

NAME	EXPRESSION	DESCRIPTION
`ewfd.Rorder_0_0`	`ewfd.Rport_K`	Reflectance, order [0,0]
`ewfd.Torder_0_0`	`ewfd.Tport_K`	Transmittance, order [0,0]
`ewfd.Rorder_0_0_orth`	`ewfd.Rport_K`	Reflectance, order [0,0], orthogonal
`ewfd.Torder_0_0_orth`	`ewfd.Tport_K`	Transmittance, order [0,0], orthogonal
`ewfd.Rorder_M_N_ip`	`ewfd.Rport_K`	Reflectance, order [M,N], in-plane
`ewfd.Rorder_M_N_op`	`ewfd.Rport_K`	Reflectance, order [M,N], out-of-plane
`ewfd.Torder_M_N_ip`	`ewfd.Tport_K`	Transmittance, order [M,N], in-plane
`ewfd.Torder_M_N_op`	`ewfd.Tport_K`	Transmittance, order [M,N], out-of-plane

In the table above, the variables with the suffix `0_0` represents the mode of the excitation periodic port. The variables with the suffix `0_0_orth` represent the reflectance/transmittance for the zero-order mode with a polarization that is orthogonal to the exciting periodic port mode. The suffix `M_N_ip` represents the mode

[M,N] with in-plane polarization. Here, M and N are written p1 for the positive +1 mode number and n1 for the negative −1 mode number, and so on. Variables with the suffix op represent diffraction orders with out-of-plane polarization. In-plane modes have the polarization in the plane spanned by the mode wave vector and port normal, whereas out-of-plane modes are orthogonal to this plane.

Notice that the port mode numbers M and N are completely unrelated to the port name K.

For 2D, the corresponding diffraction order variables are

NAME	EXPRESSION	DESCRIPTION
ewfd.Rorder_0	ewfd.Rport_K	Reflectance, order 0
ewfd.Torder_0	ewfd.Tport_K	Transmittance, order 0
ewfd.Rorder_0_orth	ewfd.Rport_K	Reflectance, order 0, orthogonal
ewfd.Torder_0_orth	ewfd.Tport_K	Transmittance, order 0, orthogonal
ewfd.Rorder_M_ip	ewfd.Rport_K	Reflectance, order M, in-plane
ewfd.Rorder_M_op	ewfd.Rport_K	Reflectance, order M, out-of-plane
ewfd.Torder_M_ip	ewfd.Tport_K	Transmittance, order M, in-plane
ewfd.Torder_M_op	ewfd.Tport_K	Transmittance, order M, out-of-plane

Port Mode Field Variables

The Port, Diffraction Order, Orthogonal Polarization, and Periodic Port nodes add the following variables related to port mode fields

Name	Description
ewfd.tEmodeC_K	Port tangential electric mode field, C-component
ewfd.normtEmode_K	Port tangential electric mode field norm
ewfd.Enmode_K	Normal electric boundary mode field
ewfd.EmodeC_K	Electric mode field, C-component
ewfd.normEmode_K	Electric mode field norm
ewfd.tHmodeC_K	Port tangential magnetic mode field, x-component
ewfd.normtHmode_K	Port tangential magnetic mode field norm
ewfd.Hnmode_K	Normal magnetic boundary mode field
ewfd.HmodeC_K	Magnetic mode field, C-component
ewfd.normHmode_K	Magnetic mode field norm

Name	Description
ewfd.beta_K	Propagation constant
ewfd.neff_K	Effective mode index

In the table above, the suffix C denotes the vector component (x, y, and z in 2D and 3D, and r, phi, and z in 2D axisymmetry). The suffix K represents the port name.

The electric mode field is defined from the tangential electric mode field and the normal component of the electric mode field as

ewfd.EmodeC_K = ewfd.tEmodeC_K + ewfd.Enmode_K*ewfd.nC,

where ewfd.nC is the C-component of the normal to the port boundary.

The magnetic mode field is defined from its tangential and normal components in the same way as the electric mode field.

For most port types, the mode fields and the propagation constant are defined from analytical expressions or as a user input. However, for **Numeric** ports the electric mode fields and the propagation constant are solved for using a **Boundary Mode Analysis** study step.

Additional Variables for Periodic Structure Calculations

To describe the unit cell of the periodic structure and to define the wave vectors for the incident and diffracted plane waves, a few different vectors are used.

When the Periodic Structure feature is used, a single axis unit vector $\mathbf{a}$ is defined. This vector is parallel to the normal to the excitation port boundary. Instead, when Port features are used, two axis vectors, $\mathbf{a}_R$ and $\mathbf{a}_T$, are defined, one for each port boundary. The axis vectors are parallel to the normal for each port boundary. The different unit cell axis vectors are summarized in Table 2-8.

TABLE 2-8: UNIT CELL AXIS VECTOR VARIABLES

NAME	DESCRIPTION	DEFINING FEATURE
ewfd.axisC	Unit cell axis, C-component	Periodic Structure
ewfd.axisRC	Unit cell axis on reflection side, C-component	Port
ewfd.axisTC	Unit cell axis on transmission side, C-component	Port

Here, `c` represents any of the Cartesian coordinates, `x`, `y`, or `z`. The reflection side is the side (boundary) where there is an excitation port. At the transmission side, there is no excitation port.

To define the propagation directions for the plane waves, two unit vector reference axes are defined, $\mathbf{a}_1$ and $\mathbf{a}_2$. When a Periodic Structure feature is used, the two reference axes are defined for the whole periodic structure. The unit cell axis vector and the two reference axes vectors are related by

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

Thus, the three vectors are all orthogonal to each other and the cross product of the two reference vectors points in the direction of the unit cell axis vector.

Given the reference axes and the unit cell axis, the wave vector for the excitation side port mode field is given by

$$\mathbf{k}_{00} = k(\sin\alpha_1(\cos\alpha_2\mathbf{a}_1 + \sin\alpha_2\mathbf{a}_2) + \cos\alpha_1\mathbf{a}) = \mathbf{k}_{T00} + \beta_{00}\mathbf{n} .$$

Here, the subscripts 00 indicate that this is the zeroth diffraction order, k is the wave number for the material in the domain adjacent to the port boundary, α_1 is the elevation angle, measured from the unit cell axis, α_2 is the azimuth angle, measured from the first reference axis, $\mathbf{k}_{T00}$ is the wave vector component tangential to the port boundary, β_{00} is the propagation constant, and $\mathbf{n}$ is the port boundary normal. Note that β_{00} is positive and $\mathbf{n}$ points out from the simulation domain. So, $\mathbf{k}_{00}$ is the wave vector for the outgoing wave.

When Port features are used, the reference axes are defined for each port boundary. So,

$$\mathbf{a}_R = \mathbf{a}_{1R} \times \mathbf{a}_{2R}$$

and

$$\mathbf{a}_T = \mathbf{a}_{1T} \times \mathbf{a}_{2T} .$$

The wave vector is defined as for the Periodic Structure case above, but the unit cell axis $\mathbf{a}$ and the reference axes $\mathbf{a}_1$ and $\mathbf{a}_2$ are replaced by $\mathbf{a}_R$, $\mathbf{a}_{1R}$, and $\mathbf{a}_{2R}$ or $\mathbf{a}_T$, $\mathbf{a}_{1T}$,

and $\mathbf{a}_{2T}$, depending on the port side (reflection or transmission). The reference axis variables are summarized in Table 2-9.

TABLE 2-9: REFERENCE AXIS VARIABLES

NAME	DESCRIPTION	DEFINING FEATURE
ewfd.aUnit1C	First reference direction, C-component	Reference Direction
ewfd.aUnit2C	Second reference direction, C-component	Reference Direction
ewfd.aUnit1RC	First reference direction on reflection side, C-component	Port
ewfd.aUnit2RC	Second reference direction on reflection side, C-component	Port
ewfd.aUnit1TC	First reference direction on transmission side, C-component	Port
ewfd.aUnit2TC	Second reference direction on transmission side, C-component	Port

The (primitive) unit cells, the periodic structures, are arranged in an infinite two-dimensional or one-dimensional lattice. So for a point $\mathbf{r}$ in this lattice, the neighborhood to the point $\mathbf{r}$ looks the same as the neighborhood to the translated point $\mathbf{r} + \mathbf{T}$, where $\mathbf{T}$ is a translation vector

$$\mathbf{T} = i\mathbf{b}_1 + j\mathbf{b}_2 .$$

Here, i and j are integers and $\mathbf{b}_1$ and $\mathbf{b}_2$ are the primitive vectors that define the unit cell.

The primitive cell vectors are most often aligned with two of the sides of the unit cell. However, for hexagonal unit cells, the primitive cell is a rhomboid. So, the primitive vectors don't align with the hexagon sides. Note that the primitive cell vectors, $\mathbf{b}_1$ and $\mathbf{b}_2$, don't need to be orthogonal, whereas the reference axes $\mathbf{a}_1$ and $\mathbf{a}_2$ are always orthogonal.

As the problem is periodic in real space, the solution can be found as a Fourier series expansion

$$\mathbf{E}(\mathbf{r}) = \sum_{m,n} \mathbf{E}_{mn} \exp(-i((\mathbf{k}_{T00} + m\mathbf{G}_1 + n\mathbf{G}_2) \cdot \mathbf{r})) ,$$

where $\mathbf{G}_1$ and $\mathbf{G}_2$ are the reciprocal lattice vectors for the unit cell and m and n are integers. The reciprocal lattice vectors are calculated from the primitive cell vectors as

$$\mathbf{G}_1 = \frac{2\pi}{A}(\mathbf{b}_2 \times \mathbf{a})$$

and

$$\mathbf{G}_2 = \frac{2\pi}{A}(\mathbf{a} \times \mathbf{b}_1),$$

where

$$A = \mathbf{a} \cdot (\mathbf{b}_1 \times \mathbf{b}_2).$$

The reciprocal lattice vectors are then used when defining the wave vectors for the mode field for the different diffraction orders,

$$\mathbf{k}_{mn} = k(\sin\alpha_{1mn}(\cos\alpha_{2mn}\mathbf{a}_1 + \sin\alpha_{2mn}\mathbf{a}_2) + \cos\alpha_{1mn}\mathbf{n})$$

$$= \mathbf{k}_{T00} + m\mathbf{G}_1 + n\mathbf{G}_2 + \beta_{mn}\mathbf{n}.$$

The relation above defines the propagation angles α_{1mn} and α_{2mn} for the diffraction order with mode numbers m and n.

The discussion about the primitive cell and reciprocal lattice vectors above describes the procedure when Periodic Structure features are used. When Port features are used, the only difference to the procedure above is that there are different sets of vectors for the reflection and transmission sides of the periodic structure. The primitive cell and reciprocal lattice vector variables are summarized in Table 2-10.

TABLE 2-10: PRIMITIVE CELL AND RECIPROCAL LATTICE VECTOR VARIABLES

NAME	DESCRIPTION	DEFINING FEATURE
`ewfd.b1C`	First primitive vector, C-component	Periodic Structure
`ewfd.b2C`	Second primitive vector, C-component	Periodic Structure
`ewfd.G1C`	First reciprocal lattice vector, C-component	Periodic Structure
`ewfd.G2C`	Second reciprocal lattice vector, C-component	Periodic Structure
`ewfd.b1RC`	First primitive vector on reflection side, C-component	Port
`ewfd.b2RC`	Second primitive vector on reflection side, C-component	Port

TABLE 2-10: PRIMITIVE CELL AND RECIPROCAL LATTICE VECTOR VARIABLES

NAME	DESCRIPTION	DEFINING FEATURE
ewfd.G1RC	First reciprocal lattice vector on reflection side, C-component	Port
ewfd.G2RC	Second reciprocal lattice vector on reflection side, C-component	Port
ewfd.b1TC	First primitive vector on transmission side, C-component	Port
ewfd.b2TC	Second primitive vector on transmission side, C-component	Port
ewfd.G1TC	First reciprocal lattice vector on transmission side, C-component	Port
ewfd.G2TC	Second reciprocal lattice vector on transmission side, C-component	Port

The Port (of Periodic type), Diffraction Order, Orthogonal Polarization, and Periodic Port nodes also add additional global variables that describe the properties of the plane-wave diffraction orders, used in periodic structure simulations.

NAME	DESCRIPTION	DIMENSION
ewfd.kModex_K	Port mode wave vector, port K, x-component	2D, 3D
ewfd.kModey_K	Port mode wave vector, port K, y-component	2D, 3D
ewfd.kModez_K	Port mode wave vector, port K, z-component	2D, 3D
ewfd.Eamplx_K	Electric mode field amplitude, port K, x-component	2D, 3D
ewfd.Eamply_K	Electric mode field amplitude, port K, y-component	2D, 3D
ewfd.Eamplz_K	Electric mode field amplitude, port K, z-component	2D, 3D
ewfd.Emodex_K	Electric mode field port K, x-component	2D, 3D
ewfd.Emodey_K	Electric mode field port K, y-component	2D, 3D
ewfd.Emodez_K	Electric mode field port K, z-component	2D, 3D
ewfd.alpha1Port_K	Elevation angle, port K	2D, 3D
ewfd.alpha1R_M	Elevation angle on reflection side, order M	2D
ewfd.alpha1T_M	Elevation angle on transmission side, order M	2D
ewfd.alpha1R_M_N	Elevation angle on reflection side, order [M,N]	3D

NAME	DESCRIPTION	DIMENSION
ewfd.alpha1T_M_N	Elevation angle on transmission side, order [M,N]	3D
ewfd.alpha2Port_K	Azimuth angle, port K	2D, 3D
ewfd.alpha2R_M	Azimuth angle on reflection side, order M	2D
ewfd.alpha2T_M	Azimuth angle on transmission side, order M	2D
ewfd.alpha2R_M_N	Azimuth angle on reflection side, order [M,N]	3D
ewfd.alpha2T_M_N	Azimuth angle on transmission side, order [M,N]	3D

In the table above, `K` represents the port name and `M` and `N` are diffraction order mode numbers.

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

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 `I` if the mode field represents the input field and for fields representing outgoing waves, `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.JIOOP_0_n1` represents the Jones vector element for the out-of-plane mode in 3D with mode numbers $m = 0$ and $n = -1$ for the exciting Periodic port input field.

There are also variables for the norm of the Jones vector, named as `tag.normJX_Z`, where `X` represents the boundary location and whether it is an input or outgoing wave (as for the variables described above) and `Z` represents the mode number. Thus, the variable `ewfd.normJR_0_n1` represents the Jones vector norm in 3D with mode numbers $m = 0$ and $n = -1$ for an outgoing wave on 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.

Coupling and Loss Calculations

In this section:

Coupling Between Sources and Destinations

Some boundary conditions, like the Port and Scattering Boundary Condition, provide options for well-defined input fields. If such an input condition is used, it defines the input power variable, `ewfd.Pin`, where `ewfd` is the tag of the physics interface.

Currently, the input power variable, `ewfd.Pin`, is only defined by the Port and Scattering Boundary Condition input fields for The Electromagnetic Waves, Frequency Domain Interface, when **Formulation** is set to **Full field**, and for The Electromagnetic Waves, Beam Envelopes Interface.

The input power variable is only defined for driven study steps, like the Frequency Domain and Wavelength Domain study steps, but not for Eigenfrequency, Mode Analysis, and Boundary Mode Analysis study steps.

At boundaries, where the waves leave the simulation domain, such as at the boundaries toward Perfectly Matched Layer domains, or at Port and Scattering Boundary Condition boundaries, the power of the outgoing waves is computed and stored in variables named like `ewfd.sctr1.Pout`, where `sctr1` is the tag of the particular physics feature defined on the boundary. The different output power contributions are then summed up hierarchically with `ewfd.Pout` representing the total power of all

outgoing waves from the physics interface. All output power variables are summarized in Table 2-11.

TABLE 2-11: VARIABLES FOR THE POWER OF OUTGOING WAVES.

NAME	DESCRIPTION	DEFINITION
ewfd.wee1.pml1.Pout	Power, outgoing wave	Power of outgoing wave flowing from the Wave Equation, Electric node wee1 into the Perfectly Matched layer node pml1.
ewfd.wee1.Pout	Power, outgoing wave	Sum of all power of outgoing waves flowing from the Wave Equation, Electric node wee1 into Perfectly Matched layer nodes.
ewfd.sctr1.Pout	Power, outgoing wave	Power of outgoing wave flowing out from the Scattering Boundary Condition node sctr1.
ewfd.port1.Pout	Power, outgoing wave, port 1	Power of outgoing wave flowing out from the Port node with tag port1.
ewfd.PoutMPort_N	Power, outgoing wave, matched part on boundary of port N	Sum of all power of outgoing waves matching any the Port nodes on the boundary of the Port node named N.
ewfd.PoutNMPort_N	Power, outgoing wave, non-matched part on boundary of port N	Power of outgoing waves not matching any of the Port nodes on the boundary of the node named N.
ewfd.PoutPort_N	Power, outgoing wave, on boundary of port N	Sum of ewfd.PoutMPort_N and ewfd.PoutNMPort_N.
ewfd.Pout	Power, outgoing wave	Sum of all power of outgoing waves flowing out from the physics interface.

For the variables ewfd.PoutMPort_N, ewfd.PoutNMPort_N, and ewfd.PoutPort_N, the suffix N represents the value of the **Port name** parameter for the first Port node listed by the physics on the particular boundary.

The variable ewfd.PoutNMPort_N is always equal to zero, unless the **Activate slit condition on interior port** checkbox is enabled and **Slit type** is set to **Domain-backed** in the Port **Settings**.

When The Electromagnetic Waves, Beam Envelopes Interface is used, the power variables related to the Wave Equation, Beam Envelopes node are called ewbe.webe1.pml1.Pout and ewbe.webe1.Pout, representing the power flowing from the Wave Equation, Beam Envelopes node webe1 into the Perfectly Matched

Layer node `pml1` or into all Perfectly Matched Layer nodes overlapping with the Wave Equation, Beam Envelopes node, respectively.

Currently, the output power variables are only defined by Port and Scattering Boundary Condition features. In addition, it is also defined on interior boundaries adjacent to Perfectly Matched Layer domains.

The output power variables are defined for all study steps and both for **Full field** and **Scattered field Formulation**.

When there is a single feature that provides the input power, it is possible to calculate the coupling of power from that input feature to the different output boundaries. These outcoupling variables form a similar hierarchy as the variables for the power of the outgoing waves. All outcoupling variables are summarized in Table 2-12.

TABLE 2-12: VARIABLES FOR THE OUTCOUPLING OF OUTGOING WAVES.

NAME	DESCRIPTION	DEFINITION
`ewfd.wee1.pml1.etaOut`	Outcoupling efficiency	`ewfd.wee1.pml1.Pout/ewfd.Pin`
`ewfd.wee1.etaOut`	Outcoupling efficiency	`ewfd.wee1.Pout/ewfd.Pin`
`ewfd.sctr1.etaOut`	Outcoupling efficiency	`ewfd.sctr1.Pout/ewfd.Pin`
`ewfd.port1.etaOut`	Outcoupling efficiency, port 1	`ewfd.port1.Pout/ewfd.Pin`
`ewfd.etaOutMPort_N`	Outcoupling efficiency, matched part on boundary of port N	`ewfd.PoutMPort_N/ewfd.Pin`
`ewfd.etaOutNMPort_N`	Outcoupling efficiency, non-matched part on boundary of port N	`ewfd.PoutNMPort_N/ewfd.Pin`
`ewfd.etaOutPort_N`	Outcoupling efficiency, on boundary of port N	`ewfd.PoutPort_N/ewfd.Pin`
`ewfd.etaOut`	Outcoupling efficiency	`ewfd.Pout/ewfd.Pin`

See Table 2-11 for the definition of the variables for the power of the outgoing waves. The suffix `N` for the Port-related variables, represents the value of the **Port name** parameter.

The outcoupling variables are only defined when the input power variable is defined.

For convenience, there are also variables representing scattering loss. With scattering loss means outcoupling through boundaries that almost always represents some sort of unwanted leakage of radiation. The scattering loss variables are summarized in Table 2-13.

TABLE 2-13: SCATTERING LOSS VARIABLES.

NAME	DESCRIPTION	DEFINITION
`ewfd.wee1.pml1.Lsca`	Scattering loss	`ewfd.wee1.pml1.etaOut`
`ewfd.wee1.Lsca`	Scattering loss	`ewfd.wee1.etaOut`
`ewfd.LscaPort_N`	Scattering loss, non-matched part on boundary of port `N`	`ewfd.etaOutNMPort_N`
`ewfd.Lsca`	Scattering loss	Sum of all scattering losses from the physics interface.

There is no scattering loss variable defined for the Scattering Boundary Condition, as the outgoing waves through Scattering Boundary Condition boundaries often represent “useful” radiation. As a consequence, the outcoupling from Scattering Boundary Condition nodes is not included in the summation variable `ewfd.Lsca`.

Similarly, for Port nodes, only the part of the outgoing waves that does not match any of the port modes on the boundary is considered as scattering loss.

To account for absorption in domains and at boundaries, loss variables are defined as listed in Table 2-14.

TABLE 2-14: LOSS VARIABLES.

NAME	DESCRIPTION	DEFINITION
ewfd.wee1.Ploss	Power loss	Sum of all power absorbed in the Wave Equation, Electric node wee1.
ewfd.imp1.Ploss	Power loss	Power absorbed in the Impedance Boundary Condition node imp1.
ewfd.libc1.Ploss	Power loss	Power absorbed in the Layered Impedance Boundary Condition node libc1.
ewfd.ltbc1.Ploss	Power loss	Power absorbed in the Layered Transition Boundary Condition node ltbc1.
ewfd.trans1.Ploss	Power loss	Power absorbed in the Transition Boundary Condition node trans1.
ewfd.Ploss	Power loss	Sum of all power absorbed in the physics interface.

When the input power variable is defined, it is possible to define the absorptance for each lossy feature and for the total absorptance. The different absorptance variables are summarized in Table 2-15.

TABLE 2-15: ABSORPTANCE VARIABLES.

NAME	DESCRIPTION	DEFINITION
ewfd.wee1.A	Absorptance	ewfd.wee1.Ploss/ewfd.Pin
ewfd.imp1.A	Absorptance	ewfd.imp1.Ploss/ewfd.Pin
ewfd.libc1.A	Absorptance	ewfd.libc1.Ploss/ewfd.Pin
ewfd.ltbc1.A	Absorptance	ewfd.ltbc1.Ploss/ewfd.Pin
ewfd.trans1.A	Absorptance	ewfd.trans1.Ploss/ewfd.Pin
ewfd.Atotal	Absorptance	ewfd.Ploss/ewfd.Pin

For an example model demonstrating the use of the outcoupling and absorptance variables discussed above, see *Modeling a Scatterer Near an Optical Waveguide*: Application Library path **Wave_Optics_Module/Waveguides/waveguide_with_scatterer**.

Relations to Other Port Variables

For Port nodes, the outcoupling variables are related to S-parameter expressions and reflectance/transmittance variables as shown in Table 2-16 below.

TABLE 2-16: RELATIONS BETWEEN PORT AND OUTCOUPLING VARIABLES.

VARIABLE NAME	RELATED PORT VARIABLE
`ewfd.portM.etaOut`	`ewfd.Rport_N` or `ewfd.Tport_N`, depending on whether the port with tag `portM` is on the reflection or transmission side.
`ewfd.etaOutMPort_N`	`ewfd.Rtotal` or `ewfd.Ttotal`, depending on whether the port with port name `N` is on reflection or transmission side.

In the variables and expressions in Table 2-16, `portM` is the tag of the port and `N` is the value of the **Port name** parameter. The reflection side represents the boundary for the excited port and the transmission side represents the side where there is no excited port.

For more information about Port variables, see S-Parameters and Ports.

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 this is not necessarily the case; 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

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

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

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

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

VARIABLES AFFECTED BY EIGENFREQUENCY ANALYSIS

The following table lists 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 a form other 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 they 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 provides some background to the problem of defining the eigenvalue linearization point. The complication is not only the nonlinearity itself, but also the way it enters the equations. For example, the impedance boundary condition with nonzero boundary conductivity has the term

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

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 specified in the **Search for eigenfrequencies around shift** 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 above equation. For cases other 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 Solver** node (not the Eigenfrequency node) under the **Solver Configurations** node in the **Study** branch of the Model Builder. A solver configuration can be generated first. In the **General** section, select the **Transform eigenvalue linearization point** checkbox and enter a suitable value in the **Value of**

eigenvalue linearization 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 configurations in the **Eigenvalue Solver** 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.

- 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*

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

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, except for 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, while the real part, δ_z, represents the damping along the propagation direction. Thus,

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

where λ is the eigenvalue.

In 2D axisymmetry, the complex propagation constant is given by

$$\alpha = -\frac{\lambda}{r_{\text{ave}}},$$

where r_{ave} is the average radius of curvature for the geometry.

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(alpha)`	No	Propagation constant
`dampz`	`real(alpha)`	No	Attenuation constant
`dampzdB`	`20*log10(exp(1))*dampz`	No	Attenuation per meter in dB
`neff`	`-j*alpha/k0`	Yes	Effective mode index

In the table above, `alpha=-lambda`, `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 labeled `i`.

	For an example of Mode Analysis, see the model *Step-Index Fiber Bend*: Application Library path **Wave_Optics_Module/Waveguides/step_index_fiber_bend**.
	For an example of Boundary Mode Analysis, see the model *Directional Coupler*: Application Library path **Wave_Optics_Module/Couplers_Filters_and_Mirrors/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-FK55, H-FK61, H-FK61B, H-FK71, H-FK95N, H-QK1, H-QK3L, H-K1, H-K2, H-K3, K4A, 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-LaK71, H-LaK72, H-TK9, H-KF6, H-QF1, QF1, H-QF3, QF3, QF5, H-QF6A, QF6, H-QF8, QF8, H-QF14, H-QF50, H-QF50A, QF50, H-QF56, H-F1, F1, H-F2, F2, H-F3, F3, H-F4, F4, H-F5, F5, H-F6, F6, F7, H-F13, F13, H-F51, H-F52, H-BaF2, H-BaF3, H-BaF4, H-BaF5, H-BaF6, H-BaF7, BaF7, H-BaF8, H-ZBaF1, ZBaF2, H-ZBaF3, H-ZBaF4, H-ZBaF5, H-ZBaF16, ZBaF17, H-ZBaF20, H-ZBaF21, H-ZBaF50, ZBaF51, H-ZBaF52, H-ZF1A, H-ZF1, ZF1, H-ZF2, ZF2, H-ZF3, ZF3, H-ZF4A, H-ZF4AGT, ZF4, H-ZF5, ZF5, H-ZF6, ZF6, H-ZF7LA, H-ZF7LAGT, ZF7, ZF7L, ZF7LGT, ZF7LTT, ZF8, H-ZF10, ZF10, H-ZF11, ZF11, H-ZF12, ZF12, H-ZF13, H-ZF13GT, ZF13, H-ZF39, H-ZF50, ZF50, ZF51, H-ZF52, H-ZF52GT, H-ZF52TT, H-ZF52A, ZF52, 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-ZLaF55C, H-ZLaF55D, H-ZLaF56B, H-ZLaF66, H-ZLaF66GT, H-ZLaF68B, H-ZLaF68C, H-ZLaF68N, H-ZLaF69, H-ZLaF69A, H-ZLaF71, H-ZLaF71AGT, H-ZLaF73, H-ZLaF75A, H-ZLaF75B, H-ZLaF75C, H-ZLaF76, H-ZLaF76A, H-ZLaF78B, H-ZLaF89L, H-ZLaF89LA, H-ZLaF90, H-ZLaF90A, H-ZLaF91, H-ZLaF92, H-ZLaF92A, H-ZLaF96, H-TF3L, TF3, H-TF5, D-FK61, D-FK61-25, D-FK61A, D-FK61A-25, D-FK95, D-FK95-25, D-QK3L, D-QK3L-25, D-K9, D-K9-25, D-K9GT, D-K9GT-25, D-K59, D-K59-25, D-PK3, D-PK3-25, D-ZPK1A, D-ZPK1A-25, D-ZPK3, D-ZPK3-25, D-ZPK5, D-ZPK5-25, D-ZPK7, D-ZPK7-25, D-ZK2, D-ZK2-25, D-ZK2A, D-ZK2A-25, D-ZK2L, D-ZK2L-25, D-ZK3, D-ZK3-25, D-ZK3A, D-ZK3A-25, D-ZK3L, D-ZK3L-25, D-ZK21, D-ZK21-25, D-ZK79, D-ZK79-25, D-LaK5, D-LaK5-25, D-LaK6, D-LaK6-25, D-LaK52, D-LaK52-25, D-LaK70, D-LaK70-25, D-ZF10, D-ZF10-25, D-ZF93, D-ZF93-25, D-LaF50, D-LaF50-25, D-LaF050, D-LaF050-25, D-LaF53, D-LaF53-25, D-LaF79, D-LaF79-25, D-ZLaF50, D-ZLaF50-25, D-ZLaF52LA, D-ZLaF52LA-25, D-ZLaF61, D-ZLaF61-25, D-ZLaF67, D-ZLaF67-25, D-ZLaF81, D-ZLaF81-25, D-ZLaF85A, D-ZLaF85A-25, D-ZLaF85LN, D-ZLaF85LN-25, D-ZLaF85L, D-ZLaF85L-25, D-ZLaF85LS, D-ZLaF85LS-25, H-FK61GTi, H-QK3LGTi, H-QK30GTi, H-K90GTi, QF50GTi, F4GTi, 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 cdgmglass.com. For other questions concerning the materials, please contact info@universalphotonics.com.

Product catalog: cdgmglass.com.

CORNING INCORPORATED

Corning® Gorilla® Glass Victus®, 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, ULE, and VICTUS 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, L-LAH94, S-FPL51Y, S-FSL5Y, BSL7Y, BAL15Y, BAL35Y, BSM51Y, PBL1Y, PBL6Y, PBL25Y, PBL26Y, PBL35Y, PBM2Y, PBM8Y, PBM18Y, SK-1300, and SK-1310.

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: oharacorp.com/optical-glass/.

SCHOTT AG

BK7G18, F2, F2G12, F2HT, F2HTi, F5, FK5HTi, K10, K5G20, LAK9G15, K7, LAFN7, LASF35, LF5, LF5G19, 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-LAK28, 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-LASF55, 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-SK5HTi, 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, SF3, SF4, SF5, SF56A, SF57, SF57HTultra, SF6, SF6G05, SF6HT, and ZERODUR®.

The foregoing materials are the property of Schott AG. For product information concerning these materials, see www.schott.com. 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: 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/Modulators_and_Switches/mach_zehnder_modulator**
- *Optical Ring Resonator Notch Filter 3D*: Application Library path **Wave_Optics_Module/Couplers_Filters_and_Mirrors/optical_ring_resonator_3d**
- *Optical Ring Resonator Notch Filter*: Application Library path **Wave_Optics_Module/Couplers_Filters_and_Mirrors/optical_ring_resonator**
- *Tapered Waveguide*: Application Library path **Wave_Optics_Module/Waveguides/tapered_waveguide**

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 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** checkbox 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** checkbox 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** checkbox 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 2-17 shows the symbol and SI unit for most of the physical quantities that are included with this module.

TABLE 2-17: ELECTROMAGNETIC QUANTITIES.

QUANTITY	SYMBOL	UNIT	ABBREVIATION
Angular frequency	ω	radian/second	rad/s
Attenuation constant	α	meter^{-1}	m^{-1}
Capacitance	C	farad	F
Charge	q	coulomb	C
Charge density (surface)	ρ_s	coulomb/meter^2	C/m^2
Charge density (volume)	ρ	coulomb/meter^3	C/m^3
Current	I	ampere	A
Current density (surface)	$\mathbf{J}_s$	ampere/meter	A/m
Current density (volume)	$\mathbf{J}$	ampere/meter^2	A/m^2
Electric displacement	$\mathbf{D}$	coulomb/meter^2	C/m^2
Electric field	$\mathbf{E}$	volt/meter	V/m
Electric potential	V	volt	V
Electric susceptibility	χ_e	(dimensionless)	–
Electric conductivity	σ	siemens/meter	S/m
Energy density	W	joule/meter^3	J/m^3
Force	$\mathbf{F}$	newton	N
Frequency	ν	hertz	Hz
Impedance	Z, η	ohm	Ω
Inductance	L	henry	H
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 2-17: 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, 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: **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 checkbox 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** checkbox 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** checkbox 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, 1/8 in 2D or 1/5 in 3D 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 1/8 in 2D and 1/5 in 3D 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 1/8 in 2D and 1/5 in 3D of the entered wavelength.

The maximum mesh element sizes discussed above are used with quadratic shape functions. When linear shape functions are used, 1/2 of the maximum mesh element size for quadratic shape functions are used. Similarly, when cubic shape functions are used, the maximum mesh element size is 2.25 times the maximum mesh element size for quadratic shape functions.

Furthermore, for Port features, the maximum mesh element size can be slightly finer than what is discussed above.

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.

The material property can be defined by a function, if the function use a single input argument, for example called `freq`, and this variable is also marked as a Frequency Model input in the material property group node where the function is used.

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, Linearly polarized plane wave
3D	User defined (default), Gaussian beam, Linearly polarized plane wave, Electric point dipole

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** $k_{\mathrm{t,max}}$ (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** $k_{\mathrm{t,max}}$ 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** checkbox 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 waist radius** w_0 (SI unit: m). The default is $20\pi/\texttt{ewfd.k}_0$ 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** $\mathbf{E}_{T\mathrm{bg0}}$ (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 **Nonnormalized transverse electric field amplitude, Gaussian beam** $\mathbf{E}_{T\mathrm{bg0}}$ (SI unit: V/m).
- Enter a **Wave number** k (SI unit: rad/m). The default is `ewfd.k0` 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.

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.k0` rad/m. The wave number must evaluate to a value that is the same for the domains the scattered field is applied to.

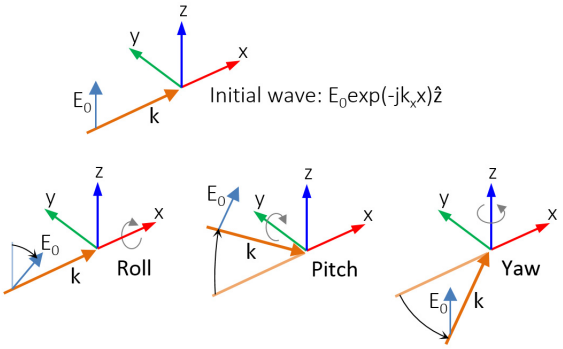


Figure 3-1: Schematic of the directions for the wave vector $\mathbf{k}$*, the electric field* $\mathbf{E}_0$*, and the roll, pitch, and yaw rotations. The top image represents an initial wave propagating in the* x *direction with a polarization along the* z *direction.*

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 φ 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.

Linearly Polarized Plane Wave (2D Axisymmetry)

Linearly polarized plane wave can be written as the sum of an infinite number of azimuthal modes in cylindrical coordinates. Thus, the **Linearly polarized plane wave** option available for 2D axisymmetric components enables fast simulation of scattering problem for body-of-revolution geometries. For a graphic representation of the initial background field and the definition of the three rotations have a look at Figure 3-2 below. To define a linearly polarized plane wave with arbitrary angle of incidence θ and polarization α:

- Enter an **Incident angle with respect to z-axis** θ (SI unit: rad). The default is 0 rad.
- Enter a **Polarization angle** α (SI unit: rad). The default is 0 rad.
- Enter a **Highest mode number**. The default is 10.
- Enter an **Electric field amplitude** E_0 (SI unit: V/m). The default is 1 V/m.
- Enter a **Wave number** k (SI unit: rad/m). The default is `ewfd.k0` rad/m.
- Click the **Set up Sweep** button. This button creates a parameter `modeNum`, which will be used as the azimuthal mode number, and a parameter `highestMode`, which is the highest mode number used in the expansion. Then the auxiliary sweep in the first frequency domain study step under the first study will be enabled and a sweep over `modeNum` will be added.

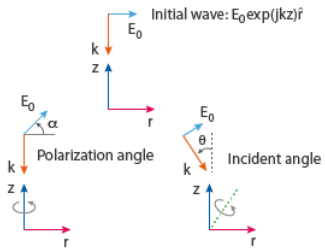


Figure 3-2: Schematic of the directions for the wave vector **k**, *the electric field* $\mathbf{E}_0$, *the polarization angle* α, *and the incident angle* θ. *The top image represents an initial wave propagating in the -z direction with a polarization along the r direction.*

Cloaking of a Cylindrical Scatterer with Graphene: Application Library path **Wave_Optics_Module/Optical_Scattering/cylinder_graphene_cloak** demonstrates how to set up the Linearly polarized plane wave background field.

Electric Point Dipole

For **Electric point dipole**, select the **Electric point dipole type** — **Electric current dipole** (the default) or **Electric charge dipole**. Select the checkbox **Dipole position is inside domain selection** if the dipole is located inside the simulation domain. By default, the checkbox is not selected.

The background wave for the **Electric current dipole** is defined as

$$\mathbf{E}_b(\mathbf{r}) = \frac{1}{j\omega}\frac{1}{4\pi\varepsilon_r\varepsilon_0}\left\{\frac{k^2}{R}(\hat{\mathbf{R}}\times\mathbf{p}_I)\times\hat{\mathbf{R}}+\left(\frac{1}{R^3}+\frac{jk}{R^2}\right)(3\hat{\mathbf{R}}[\hat{\mathbf{R}}\cdot\mathbf{p}_I]-\mathbf{p}_I)\right\}e^{-jkR}e^{j\omega t}$$

Here, $\hat{\mathbf{R}}$ is the unit vector of $\mathbf{R}$, $\mathbf{R}$ is the distance vector from the dipole position to the field position is space, R is the norm of $\mathbf{R}$, ε_r is the relative permittivity of the background medium, k is the wavenumber in the background medium, $\mathbf{p}_I$ is the electric current dipole moment (SI unit: A·m).

The background wave for the **Electric charge dipole** is defined as

$$\mathbf{E}_b(\mathbf{r}) = \frac{1}{4\pi\varepsilon_r\varepsilon_0}\left\{\frac{k^2}{R}(\hat{\mathbf{R}}\times\mathbf{p}_Q)\times\hat{\mathbf{R}}+\left(\frac{1}{R^3}+\frac{jk}{R^2}\right)(3\hat{\mathbf{R}}[\hat{\mathbf{R}}\cdot\mathbf{p}_Q]-\mathbf{p}_Q)\right\}e^{-jkR}e^{j\omega t}$$

Here, $\mathbf{p}_Q$ is the Electric charge dipole moment (SI unit: C·m).

If the **Electric point dipole type** is **Electric current dipole**, define the **Electric point dipole** using the parameters below:

- Enter the component expressions of **Electric current dipole moment** (SI Unit: A·m).
- Enter the **Dipole position** $\mathbf{r}_0$ (SI unit: m).
- Enter a **Wave number** k (SI unit: rad/m). The default is `ewfd`.k0 rad/m.

If the **Electric point dipole type** is **Electric charge dipole**, define the **Electric point dipole** using the parameters below:

- Enter the component expressions of **Electric charge dipole moment** (SI Unit: C·m).
- Enter a **Wave number** k (SI unit: rad/m). The default is `ewfd`.k0 rad/m.
- Enter the **Dipole position** $\mathbf{r}_0$ (SI unit: 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 $\mathbf{E}$.
- **Out-of-plane vector** to solve for the electric field vector component perpendicular to the modeling plane, assuming that there is no electric field in the plane.
- **In-plane vector** to solve for the electric field vector components in the modeling plane assuming that there is no electric field perpendicular to the plane.

OUT-OF-PLANE WAVE NUMBER

This section is available for 2D 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.

When using a non-zero **Out-of-plane wave number**, for most applications the appropriate setting for the **Electric field components solved for** field is **Three-component vector**. One of the few exceptions, for setting **Electric field components solved for** to **In-plane vector**, is if the solution only consists of a plane wave with a non-zero **Out-of-plane wave number** and **In-plane vector** polarization.

When performing a mode analysis study (solving for the out-of-plane wave number), the **Out-of-plane wave number** property is not used, as this is now the variable solved for.

PORT SWEEP SETTINGS

Select the **Use manual port sweep** checkbox 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 filename 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.

The electric field on port boundaries is expanded as

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

where $\mathbf{E}$, $\mathbf{r}$, $\mathbf{E}_{\text{inc}}$, S_i, $\mathbf{E}_i$, α_i, $\mathbf{n}$, and $\mathbf{r}_0$ 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 **Weak formulation** (the default value) from the **Port formulation** list. In this formulation, the expansion coefficients (or S-parameters) are calculated by adding a scalar dependent variable for each coefficient. The S-parameter and the tangential

electric field on the port boundary are solved for by adding the following weak expression for each port:

$$j\omega\mu_0(\mathbf{J}_{s,i}\cdot\text{test}(\mathbf{E}_T)-(\mathbf{n}\times\text{test}(\delta_{ij}-S_i)\mathbf{H}_i^*\cdot(\mathbf{E}_T-\mathbf{E}_{\text{bnd}})))\,,$$

where $\mathbf{J}_{s,i}$ is the surface current density for the port

$$\mathbf{J}_{s,i}=-(\delta_{ij}-S_i)\mathbf{n}\times\mathbf{H}_i\,,$$

$\mathbf{E}_T$ is the tangential electric field (the dependent variable) on the port boundary, and δ_{ij} is the Kronecker delta

$$\delta_{ij}=\begin{cases}1 & i=j\\ 0 & \text{else}\end{cases}.$$

Above, $\mathbf{H}_i$ is the magnetic mode field for the port, $\mathbf{E}_{\text{bnd}}$ is the expansion of the electric field on the port boundary in terms of the electric mode fields, $\mathbf{E}_i$,

$$\mathbf{E}_{\text{bnd}}=\sum_i(\delta_{ij}+S_i)\mathbf{E}_i\,,$$

and test() is the test operator.

In the *COMSOL Multiphysics Reference Manual*, see the Built-In Operators section for more information about the test operator.

When **Constraint-based** is selected from the **Port formulation** list, 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.

SCALING RADIUS

This section is available in 2D axisymmetry. To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog.

Select **Automatic** (the default) or **User defined** for the **Scaling radius type**. When **Automatic** is selected, the scaling radius is defined as the average radial position of the

physics selection bounding box. Select **User defined** to enter a value for the **Scaling radius** r_0 (SI unit: m). The default value is 1 m.

In COMSOL 6.2 and earlier, the scaling radius for the **Automatic Scaling Radius Type** was defined as the average radius of the bounding box of the whole geometry. In COMSOL 6.3 this changed. So, now, the **Automatic** scaling radius is defined as the average radius of the physics selection bounding box.

For models created in COMSOL 6.2 and earlier, with **Scaling Radius Type** originally set to **Automatic**, the **Scaling Radius Type** is changed to be **User defined** and the **Scaling radius** value is set to be the average radius of the geometry bounding box. That is, the value is the same as for the old **Automatic** scaling radius.

The scaling radius is used in Mode analysis in 2D axisymmetry, where the relation between the eigenvalue and the effective mode index and out-of-plane wave number is given by

$$\beta = \text{real}(k_0 n_{\text{eff}}) = -\text{imag}\left(\frac{\lambda}{r_0}\right),$$

respectively. Here, β is the out-of-plane wave number, k_0 is the vacuum wave number, n_{eff} is the effective mode index, and λ is the eigenvalue.

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.

In 2D axisymmetry, for all studies except Mode Analysis and Boundary Mode Analysis, the Covariant formulation is used. Thereby, `Ephi` is not a dependent variable. To get the correct out-of-plane electric field component, the variable `ewfd.Ephi` should be used. Here, it is assumed that the dependent variable has been given the name `E` and that the tag for the physics interface is `ewfd`.

DISCRETIZATION

Select the shape order for the **Electric field** dependent variable — **Linear**, **Linear type 2**, **Quadratic** (the default), **Quadratic type 2**, **Cubic**, **Cubic type 2**, **Quartic**, **Quartic type 2**, **Quintic**, **Quintic type 2**, **Sextic**, **Sextic type 2**, **Septic**, or **Septic 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

- Cross Section Calculation
- External Current Density
- Far-Field Domain
- Far-Field Domain, Inhomogeneous
- Initial Values
- Periodic Structure
- 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 $\mathbf{E}$ 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 following conditions are available (listed in alphabetical order):

- Axial Symmetry
- Diffraction Order
- Electric Field
- Far-Field Calculation (Far-Field Domain)
- Far-Field Calculation (Far-Field Domain, Inhomogeneous)
- Impedance Boundary Condition
- Layered Impedance Boundary Condition
- Layered Transition Boundary Condition
- Magnetic Field
- Matched Boundary Condition
- Orthogonal Polarization
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Condition
- Port
- Scattering Boundary Condition
- Substrate
- Superstrate
- Surface Current Density
- Surface Magnetic Current Density
- Symmetry Plane
- 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 Field
- 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

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

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

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

In 2D axisymmetry, for all studies except Mode Analysis and Boundary Mode Analysis, a covariant formulation for the out-of-plane electric field component is used. For more information, see Covariant formulation.

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

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

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

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

When the equation is written using the refractive index, the assumption is that $\mu_r = 1$ and $\sigma = 0$ and only the constitutive relations for linear materials are available. When

solving for the scattered field the same equations are used but $\mathbf{E} = \mathbf{E}_{sc} + \mathbf{E}_i$ and $\mathbf{E}_{sc}$ is the dependent variable.

ELECTRIC DISPLACEMENT FIELD

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

material parameters **Refractive index** n and **Refractive index, imaginary part** k form the complex relative permittivity $(n - ik)^2$.

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.

Note that the time-harmonic Sign Convention requires a lossy material to have a positive material parameter k (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 a 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 a 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. The material parameters **Relative permittivity** ε' and **Relative permittivity (imaginary part)** ε'' form the complex relative permittivity $\varepsilon_r = \varepsilon' - j\varepsilon''$.

Note that the time-harmonic Sign Convention requires a lossy material to have a positive material parameter ε'' (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.

For the **Drude–Lorentz dispersion model** select **User defined** (default) or **From material** for **Relative permittivity, high frequency** ε_∞ (dimensionless). 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.

For the **Debye dispersion model** select **User defined** (default) or **From material** for **Relative permittivity, high frequency** ε_∞ (dimensionless). 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 **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. The material parameters **relative permeability (real part)** μ' and **Relative permeability (imaginary part)** μ'' form the complex relative permeability $\mu_r = \mu' - \mu''$.

For magnetic losses, note that the time-harmonic Sign Convention requires a lossy material to have a positive material parameter μ'' (see Introducing Losses in the Frequency Domain).

CONDUCTION CURRENT

By default, the **Electric 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: Ω·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** $\boldsymbol{E}$ (SI unit: V/m). The default values are 0 V/m.

Cross Section Calculation

To calculate the scattering, absorption, and extinction cross sections of a scatterer, add a **Cross Section Calculation** node. The scattering cross section is calculated as

$$\sigma_{\text{sca}} = \frac{1}{I_0}\int_S (\mathbf{n} \cdot \mathbf{S}_{\text{sca}})dS$$

Here, $\mathbf{n}$ is the normal vector pointing outward from the scatter, $\mathbf{S}_{\text{sca}}$ is the scattered Poynting vector, and I_0 is the intensity of the incident plane wave. The integral is performed over the surface enclosing the scatterer.

The intensity of the incident wave can be calculated from the electric field amplitude $\boldsymbol{E}_0$ as

$$I_0 = \frac{1}{2}\frac{k}{k_0}c\varepsilon_0|E_0|^2$$

Here, k and k_0 are the wavenumber in the background medium and free space, respectively, c is the free space speed of light and ε_0 is the free space permittivity.

The absorption cross section is calculated as

$$\sigma_{abs} = \frac{1}{I_0}\int_V Q dV$$

Here, Q is the power loss density in the scatterer and the integral is performed over the volume V of it.

The extinction cross section yields to

$$\sigma_{ext} = \sigma_{sca} + \sigma_{abs}$$

The **Cross Section Calculation** node is only available when **Scattered field** is selected from the **Formulation** list in the **Settings** window of the physics interface, and the **Background wave type** is set to

- **User defined** for 2D,
- **User defined** or **Circularly polarized plane wave** for 2D axisymmetry,
- **User defined** or **Linearly polarized plane wave** for 3D.

Add a Cross Section Calculation node from the Physics toolbar or by right-clicking the physics interface and selecting the Cross Section Calculation item from the context menu.

CROSS SECTION CALCULATION

Enter the **Incident plane wave intensity** when the **Background wave type** is set to **User defined** for 2D and 3D. The default value is 1 W/m^2.

	To calculate the cross section area, select the entire domain of the scatterer. When the **Background wave type** is set to **User defined**, enter the electric field expression of a plane wave.

	*Scatterer on Substrate*: Application Library path **Wave_Optics_Module/Optical_Scattering/scatterer_on_substrate**

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

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: $\mathrm{A/m^2}$).

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 (Far-Field Domain) 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 exhibits isotropic material properties, and 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**

Axial Symmetry

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.

When **Electric field components solved for** in the **Components** property for The Electromagnetic Waves, Frequency Domain Interface is set to either **Three-component vector** or **Out-of-plane vector** a constraint is added on the symmetry axis

$$\Psi = 0$$

For more information, see Covariant formulation. This formulation is used for all study steps, except Mode Analysis and Boundary Mode Analysis.

CONSTRAINT SETTINGS

In the *COMSOL Multiphysics Reference Manual*, see the Constraint Settings for more information about this section.

Far-Field Calculation (Far-Field Domain)

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. The selection can be edited, but only boundaries adjacent and exterior to the Far-Field Domain are selectable. 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 the **Symmetry settings** — **From symmetry plane(s)** or **User defined**.

From Symmetry Plane(s)

When a model is reduced with Symmetry Plane features, use **From symmetry plane(s)** option to adjust far-field calculation automatically. The symmetry plane features have to coincide with one of the Cartesian coordinate planes.

User Defined

Select as needed the **Symmetry in the x=0 plane**, **Symmetry in the y=0 plane**, or **Symmetry in the z=0 plane** checkboxes 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 checkbox 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).

ADVANCED SETTINGS

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

Enter an **Average operator integration order**. The default is 2. The average operator is used when calculating the maximum directivity or total radiated power.

Select the **Maximum operator point type** — **Node point**, **Integration points**, or **Lagrange points**.

When selecting the **Integration points**, enter a **Maximum operator integration order**. The default is 2. When selecting the **Lagrange points**, enter a **Maximum operator Lagrange order**. The default is 2. The maximum operator is used when calculating the maximum directivity, maximum gain, or maximum realized gain.

A higher operator order value improves accuracy with a longer time of computation.

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 (and reset **Boundary relative to domain** to **Outside**).

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

Far-Field Domain, Inhomogeneous

To set up a far-field calculation for a structure consisting of a superstrate and a substrate, with different but isotropic material properties, add a **Far-Field Domain, Inhomogeneous** node. Use the Superstrate and the Substrate subnodes to specify the settings needed to define the superstrate and the substrate, respectively. Use the Far-Field Calculation (Far-Field Domain, Inhomogeneous) subnode to specify all other settings needed to define the far-field calculation. The normal to the interface between the substrate and the superstrate should be aligned with one of the Cartesian axes. The interface plane should be a flat surface.

If a **Perfectly Matched Layer** (PML) node has been added before adding the **Far-Field Domain, Inhomogeneous** node, all 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 domain group that is outside of all radiating and scattering objects and which has the material settings of the far-field media.

To reduce the symmetry, use a Symmetry Plane node. Not more than one Symmetry Plane node should be used to set up the far-field calculation using a **Far-Field Domain, Inhomogeneous** node.

The schematic available in the Settings for the **Far-Field Domain, Inhomogeneous** node shows the definition of the elevation and the azimuthal angles used in the calculation of the far field.

The **Far-Field Domain, Inhomogeneous** node is only available for 3D.

	Modeling with Far-Field Calculations (Far-Field Domain, Inhomogeneous)

	*Scatterer on Substrate*: Application Library path **Wave_Optics_Module/ Optical_Scattering/scatterer_on_substrate**

Superstrate

A **Superstrate** subnode is automatically added by default to the Far-Field Domain, Inhomogeneous node to select the superstrate domain or domain group of the structure. All of the domains in the geometry are automatically selected by default and

the selection cannot be modified. The domains that are not in the selection of the Electromagnetic Waves, Frequency Domain interface, they appear as not applicable. The superstrate domain or domain group should must have homogeneous material properties.

Substrate

A **Substrate** subnode is automatically added by default to the Far-Field Domain, Inhomogeneous node to select the substrate domain or domain group of the structure. The default selection is empty. Select the domain or the domain group to define the substrate medium. It determines the normal axis to the interface plane between the superstrate and the substrate. The substrate domain or domain group should must have homogeneous material properties.

Far-Field Calculation (Far-Field Domain, Inhomogeneous)

A **Far-Field Calculation** subnode is automatically added by default to the Far-Field Domain, Inhomogeneous node. If a **Perfectly Matched Layer** has been added before adding the Far-Field Domain, Inhomogeneous node, all the boundaries that enclose the domain or domain group in the Far-Field Domain, Inhomogeneous node, adjacent to the PML are automatically selected by default. If there is no PML, all the exterior boundaries that enclose the domain or domain group in the Electromagnetic Waves, Frequency Domain interface are automatically selected by default. The selection can be edited, but only boundaries adjacent and exterior to the Far-Field Domain, Inhomogeneous are selectable.

To reduce the symmetry, not more than one Symmetry Plane node should be used. If the Symmetry Plane node and a **Perfectly Matched Layer** are added before adding the Far-Field Domain, Inhomogeneous node, all the exterior boundaries of the Far-Field Domain, Inhomogeneous node that are adjacent to the PML and are not part of the Symmetry Plane boundaries, are automatically selected by default. In the absence of a **Perfectly Matched Layer**, if the Symmetry Plane node is added before adding the Far-Field Domain, Inhomogeneous node, all the exterior boundaries of the Far-Field Domain, Inhomogeneous node that are not part of the Symmetry Plane boundaries, are automatically selected by default. The selection can be modified, but only boundaries exterior to the Far-Field Domain, Inhomogeneous excluding the Symmetry Plane boundaries should be selected.

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

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: $\mathrm{C/m^2}$).

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

Periodic Structure

To perform a periodic structure calculation, add a **Periodic Structure** node. The **Periodic Structure** node has the following default subnodes:

- Wave Equation, Electric
- Perfect Electric Conductor
- Periodic Port

- Floquet Periodic Condition
- Reference Direction

By default, two Periodic Port subnodes are added. On the remaining boundaries exterior to the **Periodic Structure** node, for each pair of parallel boundaries a Floquet Periodic Condition subnode is added.

For standard modeling procedures, there are no settings to make on the Periodic Port and the Floquet Periodic Condition subnodes, as all the settings are made on the **Periodic Structure** node.

In addition, the following features can be added as subnodes to the **Periodic Structure** node:

- Impedance Boundary Condition
- Layered Impedance Boundary Condition
- Layered Transition Boundary Condition
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Transition Boundary Condition
- Wave Equation, Electric

These subnodes are available from the context menu (right-click the **Periodic Structure** parent node) or from the **Physics** toolbar, **Attributes** menu. For more information about the subnodes listed above, click the respective link.

The selection of the **Periodic Structure** node defines the periodic unit cell. The reference direction $\mathbf{a}_1$ is defined from the selections in the Reference Direction subnode in 3D and from the **Excited Port Selection** and the Reference Direction subnode selection in 2D. Given the first reference direction $\mathbf{a}_1$, the second reference direction $\mathbf{a}_2$ is calculated from

$$\mathbf{a}_2 = \mathbf{a}_0 \times \mathbf{a}_1 ,$$

where $\mathbf{a}_0$ is the periodic structure axis direction that is pointing in the direction from the passive port toward the excitation port. Thus, the axis direction equals the normal direction for the excited port.

Given the reference directions, the wave vector for the incident plane wave is given by

$$\mathbf{k}_{\text{inc}} = k[\sin\alpha_1(\cos\alpha_2\mathbf{a}_1 + \sin\alpha_2\mathbf{a}_2) - \cos\alpha_1\mathbf{a}_0] ,$$

where k is the material wavenumber at the excitation port and α_1 and α_2 are the elevation and azimuth angles, respectively. The elevation angle α_1 is the angle between the wave vector $\mathbf{k}_{\text{inc}}$ and the periodic structure axis ($\mathbf{k}_{\text{inc}}\cdot\mathbf{a}_0 \le 0$, as α_1 is in the range from 0 to $\pi/2$ radians). For normal incidence, this angle is zero. The azimuth angle α_2 is the angle between the first reference direction $\mathbf{a}_1$ and the projection of the incident wave vector $\mathbf{k}_{\text{inc}}$ on the plane spanned by $\mathbf{a}_1$ and $\mathbf{a}_2$ (the port planes).

In 2D, the elevation angle is simply called the angle of incidence and is denoted α. Furthermore, in the Out-of-Plane Wave Number section of The Electromagnetic Waves, Frequency Domain Interface it is possible to specify an out-of-plane wave vector component k_z. Thus, the wave vector of the incident plane wave is given by

$$\mathbf{k}_{inc} = k[\sin\alpha(\cos\alpha_2\mathbf{a}_1 + \sin\alpha_2\mathbf{a}_2) - \cos\alpha\mathbf{a}_0] = k\sin\alpha\cos\alpha_2\mathbf{a}_1 + k_z\mathbf{z} - \beta\mathbf{a}_0$$

From this relation we deduce that

$$\cos\alpha = \beta/k$$

and

$$\sin\alpha_2 = \frac{k_z(\mathbf{z}\cdot\mathbf{a}_2)}{k\sin\alpha} = \frac{k_z(\mathbf{z}\cdot\mathbf{a}_2)}{\sqrt{k^2-\beta^2}} .$$

Here, β is the wave vector component in the axis direction.

The first reference direction $\mathbf{a}_1$ also defines the direction of the first primitive vector of the periodic unit cell. That is,

$$\mathbf{b}_1 = b_1\mathbf{a}_1 .$$

The length b_1 is obtained from the excited port edges that are aligned with the **Excited Port Selection**. In 3D, for parallelogram-like unit cells, the second primitive vector $\mathbf{b}_2$ is defined from the cell edges that are not parallel to $\mathbf{b}_1$. Furthermore, $\mathbf{b}_2$ is defined such that

$$\mathbf{b}_1\times\mathbf{b}_2 = c\mathbf{a}_0, c>0 . \tag{3-1}$$

For hexagonal unit cells, three edge vectors are defined from three consecutive hexagon edges:

- $\mathbf{v}_1$ is defined from the hexagon edge that intersects the Reference Direction node edge and point selections.
- $\mathbf{v}_2$ is defined by the hexagon edge that starts where the edge that defines $\mathbf{v}_1$ ends.
- $\mathbf{v}_3$ is defined by the hexagon edge that starts where the edge that defines $\mathbf{v}_2$ ends.

From these three vectors, the first primitive cell vector is defined by

$$\mathbf{b}_1 = \mathbf{v}_1 - \mathbf{v}_3$$

and the second primitive cell vector is defined from

$$\mathbf{b}_2 = \mathbf{v}_1 + \mathbf{v}_2 ,$$

assuming that Equation 3-1 applies. If the cross product does not fulfill Equation 3-1, the expressions for $\mathbf{b}_1$ and $\mathbf{b}_2$ are swapped.

DOMAIN SELECTION

The selection is locked to **All domains**, meaning that the **Periodic Structure** has the same selection as the The Electromagnetic Waves, Frequency Domain Interface.

EXCITED PORT SELECTION

Select a boundary that represents or is part of the excited port. The excited port boundary defines the axis $\mathbf{a}_0$ of the periodic structure, as the excited port boundary normal and the axis point in the same direction.

To deduce the primitive vectors, the **Periodic Structure** axis direction should be aligned with one of the Cartesian axes.

PORT HANDLING

This section provides settings to add or remove the passive listener port and to add Diffraction Order subnodes.

- Clear the **Add listener port** checkbox to remove the passive listener port.
- Define the **Diffraction order specification** — **From current parameters** or **All angles** (the default). When **From current parameters** is selected, clicking the **Add Diffraction Orders** button, creates **Diffraction Order** ports that represents propagating waves for the present parameter values. When **All angles** is selected, clicking the **Add Diffraction**

Orders button, creates **Diffraction Order** ports that represents propagating waves at least for some angles of incidence. Here, the angle of incidence could be any angle from the incident hemisphere.

- Define the **Refractive index, real part, excited port** n_1 — **From adjacent domain** (the default) or **User defined**. When **From adjacent domain** is selected, the **Refractive index, real part** is taken from the domain adjacent to the port. For **User defined**, enter the **Refractive index, real part, excited port** n_1 (SI unit: 1). The default value is 1.
- If **Add listener port** is selected, define the **Refractive index, real part, listener port** n_2 as was done for **Refractive index, real part, excited port** n_1.

When all parameters are defined, click the **Add Diffraction Orders** button to automatically create **Diffraction Order** ports as subnodes to all **Periodic Port** subnodes.

Perform the same action as when clicking the **Add Diffraction Orders** button, using the COMSOL API, with the Java code

```
model.component("comp1").physics("ewfd").feature("ps1").r
unCommand("addDiffractionOrders");
```

where `"comp1"`, `"ewfd"`, and `"ps1"` are the tags for the model component, the physics interface, and the periodic structure, respectively, and `model` is a model object.

PORT MODE SETTINGS

Select a **Polarization** — **Linear polarization** (default value), **Circular polarization**, or **User defined**.

For **Linear polarization**, select the polarization type — **S** (default value), **P**, or **Mixed**, where **S** and **P** represent a polarization orthogonal or parallel to the plane of incidence (spanned by the wave vector for the incident plane wave and the port normal), respectively, and **Mixed** represents a mixture of s- and p-polarizations. For **Mixed Linear polarization** also specify the **P-polarization power fraction** η_p. The default value is `0`, meaning that the mode field will be s-polarized.

Fresnel Equations: Application Library path **Wave_Optics_Module/Verification_Examples/fresnel_equations**

For **Circular polarization**, select the type of circular polarization — **Right-handed** (default value) or **Left-handed**. The convention used here is from the point of view of the

source. That is, for right-handed circular polarization, when the thumb points in the direction of wave propagation, the temporal field polarization curls in the direction the right hand fingers. For left-handed circular polarization, the polarization curls temporally along the left hand fingers, when the thumb is pointing in the wave direction.

Reflection of a Circularly Polarized Plane Wave: Application Library path **Wave_Optics_Module/Verification_Examples/circular_polarization**

For **User defined Polarization**, select an **Input quantity** — **Electric field** (default value) or **Magnetic field** — and define the mode field amplitude for the outgoing wave at the port.

- For 2D components and if the **Input quantity** is set to **Electric field**, define the **Electric mode field amplitude**. For example, for s-polarization 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 p-polarization, set the x, y, and z components to 0, 0, and 1. The default value is the tangent vector `t1`. However, if **Electric field components solved for**, in the **Settings** for the physics interface, is set to **Out-of-plane vector**, only the out-of-plane component is nonzero for the **Electric mode field amplitude**. If **Electric field components solved for** is set to **In-plane vector**, only the out-of-plane component is nonzero for the **Magnetic mode field amplitude**.

In the *COMSOL Multiphysics Reference Manual*, see the Tangent Variables for more information about the tangent vector `t1`.

In 2D, define the **Angle of incidence** α (SI unit: radians), and in 3D define the **Elevation angle of incidence** α_1 (SI unit: radians) and the **Azimuth angle of incidence** α_2 (SI unit: radians).

Modeling Periodic Structures

Hexagonal Plasmonic Color Filter: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/hexagonal_plasmonic_color_filter**

Reference Direction

The **Reference Direction** node defines the first reference direction vector $\mathbf{a}_1$, discussed in the Periodic Structure documentation.

EDGE SELECTION

In 3D, select an edge. The direction $\mathbf{a}_1$ will be aligned with the selected edge. In 2D, this selection is not available. Instead, the direction $\mathbf{a}_1$ will be aligned with the parent Periodic Structure Excited Port Selection,

ORIGIN

This selection is only available in 3D. The direction $\mathbf{a}_1$ will point away from the selected point.

POINT SELECTION

This selection is only available in 2D. The direction $\mathbf{a}_1$ will point away from the selected point.

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.

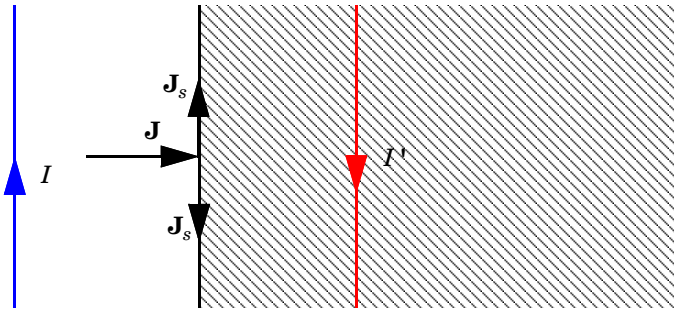


Figure 3-3: The perfect electric conductor boundary condition is used on exterior and interior boundaries representing the surface of a lossless metallic conductor or (on exterior boundaries) representing a symmetry cut. The shaded (metallic) region is not part of the model but still carries effective 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.

For a better option to work with symmetries, use Symmetry Plane.

CONSTRAINT SETTINGS

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

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 exterior 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.

The Surface Current Density subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

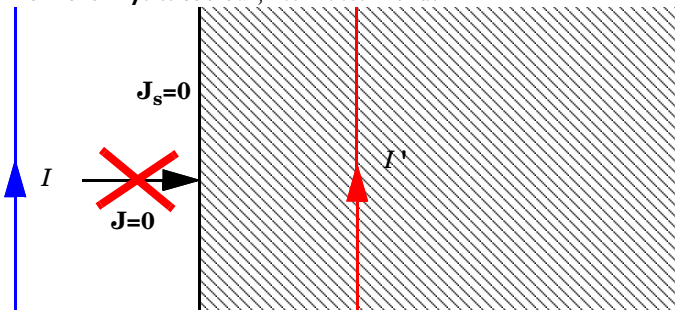


Figure 3-4: The perfect magnetic conductor boundary condition is used on exterior boundaries representing the surface of a high impedance region or a symmetry cut. The shaded (high impedance) region is not part of the model but nevertheless carries effective mirror images of the sources. Note also that 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.

For a better option to work with symmetries, use Symmetry Plane.

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, reflectance and transmittance variable names, 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

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

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** checkbox to use the **Port** boundary condition on interior boundaries.

Then select a **Slit type** — **Domain-backed** (the default) or **PEC-backed**. 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. The **PEC-backed** type makes the port on interior boundaries perform as it does on exterior boundaries.

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.

Furthermore, it should be the same material on both sides of the port,

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. The

default value is the tangent vector `t1`. However, if **Electric field components solved for**, in the **Settings** for the physics interface, is set to **Out-of-plane vector**, only the out-of-plane component is nonzero for $\mathbf{E}_0$. If **Electric field components solved for** is set to **In-plane vector**, only the out-of-plane component is nonzero for $\mathbf{H}_0$.

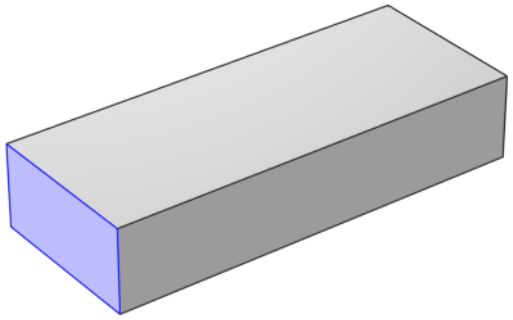
- Enter the **Propagation constant** β (SI unit: rad/m). The default value is `ewfd.k`. This parameter is frequency dependent for all but TEM modes and a correct frequency-dependent expression must be used.

	In the *COMSOL Multiphysics Reference Manual*, see the Tangent Variables for more information about the tangent vector `t1`.

	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.

Rectangular

The following figure shows an example of a boundary selection for a **Rectangular** waveguide port in a 3D model. The mode field is assigned to this selection.



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

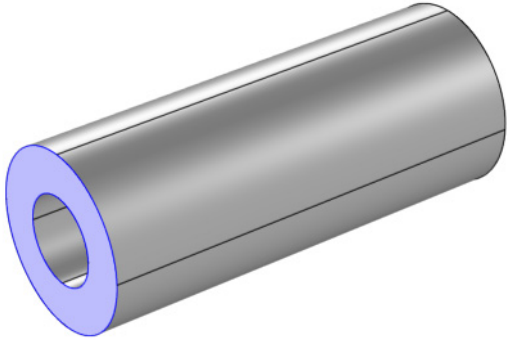
For 3D components, select a **Mode type** — **Transverse electric (TE)** or **Transverse magnetic (TM)**. Enter the **Mode number**, for example, 10 for a TE_{10} mode, or 11 for a TM_{11} mode. When the port boundaries are parallel to one of the Cartesian coordinate

planes, click the **Plot Analytical Port Mode Field** button to inspect the mode field instantly before running a simulation.

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

The following figure shows an example of a boundary selection for a **Coaxial** waveguide port in a 3D model. The mode field is assigned to this selection.

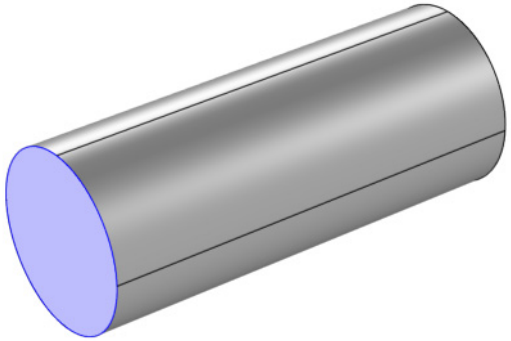


Coaxial only supports **Transverse electromagnetic (TEM)** mode type. When the port boundaries are parallel to one of the Cartesian coordinate planes, click the **Plot Analytical Port Mode Field** button to inspect the mode field instantly before running a simulation.

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

The following figure shows an example of a boundary selection for a **Circular** waveguide port in a 3D model. The mode field is assigned to this selection.



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. When the port boundaries are parallel to one of the Cartesian coordinate planes, click the **Plot Analytical Port Mode Field** button to inspect the mode field instantly before running a simulation.

Periodic

For **Periodic**, specify parameters for the mode field. When **Periodic** is selected, the Diffraction Order, Orthogonal Polarization, and Periodic Port Reference Point subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Select a **Polarization** — **Linear polarization**, **Circular polarization**, or **User defined** (default value).

For **Linear polarization**, select the polarization type — **S** (default value), **P**, or **Mixed**, where **S** and **P** represent a polarization orthogonal or parallel to the plane of incidence (spanned by the wave vector for the incident plane wave and the port normal), respectively, and **Mixed** represents a mixture of s- and p-polarization. For **Mixed Linear**

polarization also specify the **P-polarization power fraction** η_P. The default value is 0, meaning that the mode field will be s-polarized.

Fresnel Equations: Application Library path **Wave_Optics_Module/Verification_Examples/fresnel_equations**

For **Circular polarization**, select the type of circular polarization — **Right-handed** (default value) or **Left-handed**. The convention used here is from the point of view of the source. That is, for right-handed circular polarization, when the thumb points in the direction of wave propagation, the temporal field polarization curls in the direction the right hand fingers. For left-handed circular polarization, the polarization curls temporally also the left hand fingers, when the thumb is pointing in the wave direction.

Reflection of a Circularly Polarized Plane Wave: Application Library path **Wave_Optics_Module/Verification_Examples/circular_polarization**

For **User defined Polarization**, select an **Input quantity** — **Electric field** (default value) 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. The default value is the tangent vector `t1`. However, if **Electric field components solved for**, in the **Settings** for the physics interface, is set to **Out-of-plane vector**, only the out-of-plane component is nonzero for the **Electric mode field amplitude**. If **Electric field components solved for** is set to **In-plane vector**, only the out-of-plane component is nonzero for the **Magnetic mode field amplitude**.
- Define the **Angle of incidence**, if **Wave excitation at this port** is **On**.

In the *COMSOL Multiphysics Reference Manual*, see the Tangent Variables for more information about the tangent vector `t1`.

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

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

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

where $\mathbf{k}$ is the wave vector, $\mathbf{k}_{\text{parallel}}$ is the projection of $\mathbf{k}$ onto the port, $\mathbf{k}_{\text{F}}$ is the k-vector for Floquet periodicity, $\mathbf{n}$ is the outward unit normal vector to the boundary, and $\hat{\mathbf{a}}_1$ 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.

For more details about the periodic port mode fields, see Periodic Port Mode Fields

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** checkbox 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** — **From adjacent domain** (the default) or **User defined**. When **From adjacent domain** is selected, the **Refractive index, real part** is taken from the domain adjacent to the port. For **User defined**, enter the **Refractive index, real part** n (SI unit: 1). The default value is 1.
- Define the **Diffraction order specification** — **From current parameters** (the default) or **All angles**. When **From current parameters** is selected, clicking the **Add Diffraction Orders** button, creates **Diffraction Order** ports that represents propagating waves for the present parameter values. When **All angles** is selected, clicking the **Add Diffraction Orders** button, creates **Diffraction Order** ports that represents propagating waves at least for some angles of incidence. Here, the angle of incidence could be any angle from the incident hemisphere.
- 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** checkbox selected.

Perform the same action as when clicking the **Add Diffraction Orders** button, using the COMSOL API, with the Java code

```
model.component("comp1").physics("ewfd").feature("port1")
.runCommand("addDiffractionOrders");
```

where “`comp1`”, “`ewfd`”, and “`port1`” are the tags for the model component, the physics interface, and the excited port, respectively, and `model` is a model object.

CUTOFF FREQUENCY 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.

DE-EMBEDDING PORT

Set the **Port Offset** d_{offset} (SI unit: m) for the de-embedded S-parameter calculation. The default is 0 m. The phase of the de-embedded S-parameters is adjusted from the calculated S-parameters with the propagation constant and the value of d_{offset}. The full expression for the de-embedded S-parameter is given by

$$dS_{nm} = S_{nm}\exp(j\beta_n d_n)\exp(j\beta_m d_m)$$

where m is the source port name, n is the listener port name, β is the propagation constant, and d is the offset distance from the port boundary. The generated variables are typically denoted by `ewfd.dS11` and `ewfd.dS21`.

The de-embedding functionality is triggered when d_{offset} is set to a nonzero value. It is assumed that the domain between the port boundary and the boundary projected by the d_{offset} is straight, while maintaining a constant cross-sectional shape.

CONSTRAINT SETTINGS

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

- S-Parameters and Ports
- S-Parameter Variables

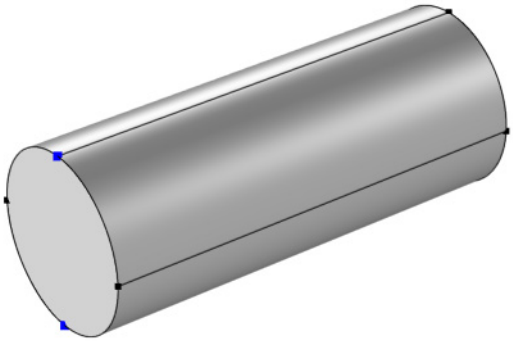
2D model with periodic ports — *Plasmonic Wire Grating*: Application Library path **Wave_Optics_Module/Gratings_and_Metamaterials/plasmonic_wire_grating**

3D model with numeric ports — *Directional Coupler*: Application Library path **Wave_Optics_Module/Couplers_Filters_and_Mirrors/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.

The following figure shows an example of a selection of two points for defining the **Circular Port Reference Axis**.



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**.

Notice that the **Components** setting is automatically updated when the **Electric field components solved for** parameter in the **Settings** for the physics interface is changed. This happens only if **Electric field components solved for** is changed to either **Out-of-plane vector** or **In-plane vector**. Furthermore, no change occurs, if appropriate **Diffraction Order** nodes already exist with **Components** set to both **Out-of-plane vector** and **In-plane vector** for each mode number. Additionally, no change occurs if the **Out-of-plane wave number**, in the **Settings** for the physics, is non-zero.

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

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

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

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

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 Periodic port assumes homogeneous isotropic material properties in the domain adjacent to the selected port boundary.

For more details about the periodic port mode fields, see Periodic Port Mode Fields

- 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 Periodic port assumes homogeneous isotropic material properties in the domain adjacent to the selected port boundary.

For more details about the periodic port mode fields, see Periodic Port Mode Fields

- 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

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

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

where $\mathbf{n}$ is the outward unit normal vector to the port boundary.

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.

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.

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**

Periodic Port

The **Periodic Port** node is a default subnode to the Periodic Structure node. These nodes are added automatically when the Periodic Structure node is added.

The Diffraction Order and Orthogonal Polarization subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. However, it is simpler to add these subnodes by clicking the **Add diffraction Orders** button in the parent Periodic Structure **Settings**.

BOUNDARY SELECTION

This selection is locked to **All boundaries** and filtered to include all boundaries in the plane defined by the parent Periodic Structure Excited Port Selection.

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, reflectance and transmittance variable names, and numeric port names are also required for port sweeps and Touchstone file export.

The remaining settings are only displayed here. The editable settings are found in the **Settings** for the parent Periodic Structure node.

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. As the equation suggests, the boundary condition only guarantees that the tangential components of $\mathbf{E}$ and $\mathbf{E}_0$ are equal. Their normal components might deviate depending on specific situations. 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. 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).

Matched Boundary Condition

Use the **Matched Boundary Condition** to make a boundary transparent to a wave with a known scattered wave direction. Since the **Scattered wave direction** setting is taken into account, this boundary condition is low reflecting also for a wave propagating in a direction at a large angle to the normal of the boundary. This is in contrast to the Scattering Boundary Condition, where the scattered beam should propagate in a direction that is almost parallel to the boundary normal to be efficiently absorbed. The boundary is also transparent to an incoming wave.

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. Taking into account the Reference Point subnode, the total electric field, including the incident and scattered waves, can be written as

$$\mathbf{E} = \mathbf{E}_0 e^{-jk(\mathbf{k}_{\mathrm{i,\,dir}} \cdot (\mathbf{r} - \mathbf{r}_{ref}))} + \mathbf{E}_s e^{-jk(\mathbf{k}_{\mathrm{s,\,dir}} \cdot (\mathbf{r} - \mathbf{r}_{ref}))} .$$

Here, $\mathbf{r}_{\mathrm{ref}}$ is a reference point determined by the Reference Point subnode, the field $\mathbf{E}_0$ is the incident wave that travels in the direction $\mathbf{k}_{\mathrm{i,dir}}$ in a medium with wave number k, and the field $\mathbf{E}_s$ is the scattered wave that travels in the direction $\mathbf{k}_{\mathrm{s,dir}}$.

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 Matched boundary condition's boundary selection.

MATCHED 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 of the **Incident electric field amplitude** $\mathbf{E}_0$ or **Incident magnetic field amplitude** $\mathbf{H}_0$, depending on the **Incident field** selected.

If the **Incident field** is set to **Gaussian beam**, edit the **Beam waist 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. 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 nonnormalized electric field amplitude** $\mathbf{E}_{g0}$ (SI unit: V/m). The optical axis for the Gaussian beam is defined by a line including a point which is the average position for the feature selection and a direction specified by the **Incident wave direction** (see below).

If the **Incident field** is not set to **No incident field**, edit the **Incident wave direction** $\mathbf{k}_{i,dir}$ vector components. The default direction is the inward normal to the boundary. For 2D axisymmetry, the direction should be parallel or anti–parallel to the symmetry axis. If no scattered field is expected, select the **No scattered field** checkbox. This prevents COMSOL from returning spurious solutions that otherwise could appear between boundaries with unconstrained scattered fields. Edit the **Scattered wave direction** $\mathbf{k}_{s,dir}$ vector components. The default direction is the outward normal to the boundary.

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

Gaussian Beam Propagation Through an Optical Prism: Application Library path **Wave_Optics_Module/Beam_Propagation/gaussian_beam_propagation_optical_prism**

Scattering Boundary Condition

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

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

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

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

The field $\mathbf{E}_0$ is the incident plane wave that travels in the direction $\mathbf{k}$. The boundary condition is transparent for incoming (but not outgoing) plane waves with any angle of incidence. 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 $\mathbf{r}_{\text{ref}}$ 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, assuming that the material properties adjacent to the boundary are isotropic. 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.

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

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

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

In the scattered field formulation, the background field must be a solution to the wave equation in the domain adjacent to the Scattering Boundary Condition feature. If the background field is not a solution to the wave equation in the adjacent domain, there will be reflections at the boundary.

So, if the scattering problem consists of a scatterer embedded in a top superstrate and a bottom substrate, a structure similar to what is used in the tutorial model *Plasmonic Wire Grating*, the background field should be either a numerical or analytical solution to the two-layer superstrate-substrate problem. The tutorial model *Scatterer on Substrate* demonstrates how to compute a numerical background field that is used in a following scattered field formulation study.

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 nonnormalized electric field amplitude** $\mathbf{E}_{g0}$ (SI unit: V/m). Also edit the **Beam waist 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** $\mathbf{k}_{\text{dir}}$ for the vector coordinates. The default direction is in the opposite direction to the boundary normal. For 2D axisymmetry, the **Incident wave direction** $\mathbf{k}_{\text{dir}}$ 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** $\mathbf{r}_{\text{axis}}$ (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** checkbox 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 checkbox is cleared (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/Beam_Propagation/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** $\mathbf{A}_0$ (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.

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 $\mathbf{r}_{\text{ref}}$ 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 $\mathbf{r}_{\text{ref}}$ at the intersection point between the parent node's boundary selection and the symmetry axis. If the parent selection does not 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, $|N| >> 1$. Furthermore, the exterior material properties are assumed to be isotropic.

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.

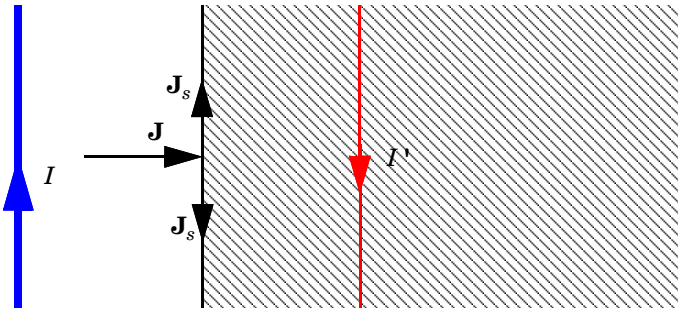


Figure 3-5: The impedance boundary condition is used on exterior boundaries representing the surface of a lossy domain. The shaded (lossy) region is not part of the model. The effective induced image currents are of reduced magnitude due to losses. Any current flowing into the boundary is perfectly balanced 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** — **From wave vector** (the default) or **Normal direction**. 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 a **Surface impedance definition** — **From material properties** (the default) or **User defined**.

For **From material properties**, 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. However, notice that only isotropic (scalar) material parameters are supported for this boundary condition.

Enter a **Surface impedance** for **User defined** (SI unit: Ω). The default is `Z0_const` that is the predefined COMSOL constant for the wave impedance of free space.

SOURCE ELECTRIC FIELD

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

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

See *Skin Depth Calculator* to evaluate the skin depth of a homogeneous material.

	Fresnel Equations: Application Library path **Wave_Optics_Module/ Verification_Examples/fresnel_equations**

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 Perfect Magnetic Conductor, Surface Magnetic Current Density, andTransition Boundary Condition, the **Surface Current Density** subnode as an one-sided surface current density applied to interior boundaries is available from the context menu (right-click the 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.

The Surface Current Density subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

SURFACE MAGNETIC CURRENT DENSITY

Enter values or expressions for the components of the **Surface magnetic current density** $\mathbf{J}_{\mathrm{ms0}}$ (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. 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 and the material properties in the thin layer are assumed to be isotropic.

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.

The Surface Current Density subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

PROPAGATION DIRECTION

This section is only available for the Electromagnetic Waves, Beam Envelopes interface. Select a **Propagation direction** — **From wave vector** (the default) or **Normal direction**. 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. However, notice that only isotropic (scalar) material parameters are supported for this boundary condition.

Select a **Type** — **Electrically thin layer**, **Electrically thick layer**, or (for the Electromagnetic Waves, Frequency Domain interface only) **Electrically very thin layer**. The **Electrically thick layer** option decouples the two domains adjacent to the boundary. This setting is suitable, for example, when the thickness exceeds three times the skin depth. If the thickness is much smaller than the skin depth, select **Electrically**

very thin layer. For both the **Electrically thin layer** and **Electrically very thin layer** options, specify a **Thickness** d (SI unit: m). The default is 0.01 m.

- *Beam Splitter*: Application Library path **Wave_Optics_Module/Beam_Propagation/beam_splitter**
- *Focusing Lens*: Application Library path **Wave_Optics_Module/Verification_Examples/focusing_lens**

RESONANCE CONSTRAINT

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. Select the **Activate resonance constraint** checkbox to apply the constraint to address resonance conditions, for lossless materials when the entered **Thickness** d is an integer number of half wavelengths. It is selected by default.

SKIN DEPTH CALCULATOR

Select a **Defined by** option — **Electric conductivity** (default) or **Resistivity**. Enter a **Electric conductivity** σ (SI unit: S/m) or **Resistivity** ρ (SI unit: Ω·m), **Relative permittivity** ε_r, **Relative permeability** μ_r of the material to be evaluated, and **Frequency** f_0 (SI unit: Hz). Then click the **Compute Skin Depth** button ( ) to compute the skin depth for the particular material specified by the above input values. The result is displayed in the settings window below the **Compute Skin Depth** button.

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:

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

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

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

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

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

where the index $i = 1, 2, \ldots, n$ 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.

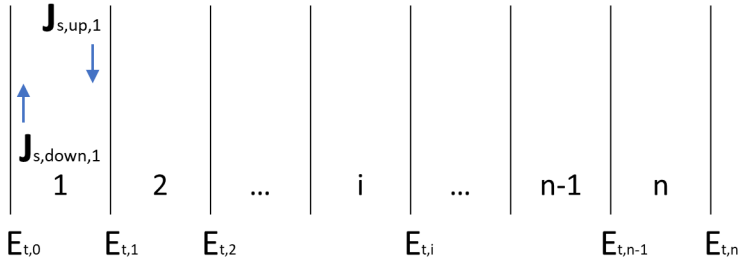


Figure 3-6: The layered material is composed of n layers. The surface currents on the up and downside of each layer are determined from the transfer and surface impedances and are functions of the tangential electric fields.

- 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 and the material properties are assumed to be isotropic.

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.

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 Layered Transition Boundary Condition node, the phase specified for the Electromagnetic Waves, Beam Envelopes interface can be discontinuous.

SHELL PROPERTIES

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

Clear the **Use all layers** checkbox 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 can 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.

PROPAGATION DIRECTION

This section is only available for the Electromagnetic Waves, Beam Envelopes interface. Select a **Propagation direction** — **From wave vector** (the default) or **Normal direction**. 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.

LAYERED 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. However, notice that only isotropic (scalar) material parameters are supported for this boundary condition.

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.

See *Skin Depth Calculator* to evaluate the skin depth of a homogeneous material.

Layered Impedance Boundary Condition

The **Layered Impedance Boundary Condition** is an extension of the **Impedance Boundary Condition** that allows to model a sequence of geometrically thin layers on top of a substrate. It is used on exterior boundaries where the field is known to penetrate only a short distance outside the boundary. In brief, this feature combines a **Layered Transition Boundary Condition** with an **Impedance Boundary Condition** of the type:

$$\sqrt{\frac{\mu_0\mu_r}{\varepsilon_c}}\mathbf{n}\times\mathbf{H}+\mathbf{E}-(\mathbf{n}\cdot\mathbf{E})\mathbf{n}\ =\ 0\quad .$$

The layer stack is built using a **Layered Material**. For each layer in the **Layered Material**, the transfer and surface impedances are obtained from the layer thickness and the 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. The mathematical details relative to the field propagation in the layer stack can be found in the Layered Transition Boundary Condition section.

- 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 Impedance Boundary Condition** is based on the assumption that in the thin layers and in the substrate the wave propagates essentially in the normal direction. 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 and the material properties are assumed to be isotropic.

SHELL PROPERTIES

The **Shell Properties** section displays which **Layered Material** is coupled to the **Layered Impedance Boundary Condition**.

Clear the **Use all layers** checkbox in order to select a specific **Layered Material** from the list. The **Layered Impedance Boundary Condition** feature is then applicable only on the boundaries where the chosen **Layered 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:

- 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 can 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.
- 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.
- 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**.

LAYER PROPERTIES

The **Layer Properties** section specifies the material properties of the thin layers constituting the stack located on top of the substrate. 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. However, notice that only isotropic (scalar) material parameters are supported for this boundary condition.

The defaults use the values **From material**. In this case, the material properties are taken layer by layer from the **Layered Material** existing on those boundaries where the **Layered Impedance Boundary Condition** feature is enabled. Otherwise, choose **User defined** and enter a value or an expression. In the latter case all layers constituting the chosen **Layered Material** will take on the same given value for the selected material property.

SUBSTRATE PROPERTIES

The **Substrate Properties** section specifies the material properties of the thick domain that is not included in the model, being approximated by an **Impedance Boundary Condition**.

Select a **Surface impedance definition** — **From material properties** (the default) or **User defined**.

Select a **Substrate Material** from the list of materials that have been introduced in the model previously. 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. The defaults use the values **From material**. In this case, the material properties are taken from the specified **Substrate Material**. Otherwise, choose **User defined** and enter a value or an expression.

Enter a **Surface impedance** for **User defined** (SI unit: Ω) in the **Surface impedance definition**. The default is `Z0_const` that is the predefined COMSOL constant for the wave impedance of free space.

See *Skin Depth Calculator* to evaluate the skin depth of a homogeneous material.

Enhanced Coating for a Microelectromechanical Mirror: Application Library path **Wave_Optics_Module/Couplers_Filters_and_Mirrors/enhanced_mems_mirror_coating** demonstrates how to set up the Layered Impedance Boundary Condition to represent a metal mirror with a thin-film coating.

Periodic Condition

The **Periodic Condition** sets up a periodicity between the selected boundaries.

BOUNDARY SELECTION

The software usually automatically identifies the boundaries as either source boundaries or destination boundaries, as indicated in the selection list. This works fine for cases like opposing parallel boundaries. In other cases, right-click **Periodic Condition** and select **Manual Destination Selection** to control the destination. By default it contains the selection that COMSOL Multiphysics identifies.

DESTINATION SELECTION

This section is available for specifying the destination boundaries, if needed, when the **Manual Destination Selection** option is selected in the context menu for the **Periodic Condition** node. You can only select destination boundaries from the union of all source and destination boundaries.

PERIODICITY SETTINGS

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

- **Continuity** to make the electric field periodic (equal on the source and destination),
- **Antiperiodicity** to make it antiperiodic,
- **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.

  The phase shift between the fields on the parallel source and destination boundaries is defined as

$$e^{-j(\mathbf{k}_F \cdot (\mathbf{r}_{dst} - \mathbf{r}_{src}))}$$

- **Cyclic symmetry** for azimuthal periodicity in a structure consisting of a number of identical sectors.
  - When **Sector angle** is set to **Automatic** (the default), the sector angle is automatically computed from the geometry. Set **Sector angle** to **User defined** to enter a manual value for the **Sector angle** θ_S (SI unit: rad).
  - Enter an integer number for the **Azimuthal mode number** m.

  The phase shift between the fields on the source and destination boundaries is defined as

$$e^{-jm\theta_S}$$

CONSTRAINT SETTINGS

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

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

ORIENTATION OF DESTINATION

This section appears if the setting for **Transform to intermediate map** in the **Orientation of Source** section is changed from the default value, **Automatic**, and **Advanced Physics Options** is selected in the **Show More Options** dialog. For information about the **Orientation of Destination** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. For information about the **Mapping**

Between Source and Destination section, see Mapping Between 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**

Floquet Periodic Condition

The **Floquet Periodic Condition** sets up a Floquet periodicity condition between the selected boundaries. It is a default subnode to the Periodic Structure node. These nodes are added automatically when the Periodic Structure node is added. First two Periodic Port subnodes are added to the parent Periodic Structure node on a pair of opposing parallel boundaries. For the remaining exterior boundaries to the parent Periodic Structure node, a **Floquet Periodic Condition** subnode is added for each pair of parallel opposing boundaries.

The phase shift between the fields on the parallel source and destination boundaries is defined as

$$e^{-j(\mathbf{k}_F \cdot (\mathbf{r}_{dst} - \mathbf{r}_{src}))},$$

where $\mathbf{k}_F$ is the Floquet wave vector (the wave vector component tangential to the Periodic Port boundaries) and $\mathbf{r}_{dst}$ and $\mathbf{r}_{src}$ are the position vectors for the **Floquet Periodic Condition** destination and source boundaries, respectively.

BOUNDARY SELECTION

The selection is locked to **All boundaries**, but filters out a pair of opposing parallel boundaries.

The remaining sections are described in the documentation for the Periodic Condition.

	Modeling Periodic Structures 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).

Symmetry Plane

The **Symmetry Plane** node adds a boundary condition that represents symmetry in the electric or magnetic field, depending on which option is chosen.

SYMMETRY TYPE

Choose between **Zero tangential electric field (PEC)** and **Zero tangential magnetic field (PMC)** for the electric field. If **Zero tangential electric field** is chosen, the boundary condition is

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

which states that the tangential components of the electric field are zero. It behaves as Perfect Electric Conductor. Use this symmetry type when there is only the normal

component of the electric field on the symmetry boundaries. If **Zero tangential magnetic field** is chosen, the boundary condition becomes

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

which states that the tangential components of the magnetic field and the surface current density are zero. It behaves as Perfect Magnetic Conductor. Use this symmetry type when there are only the tangential components of the electric field on the symmetry boundaries.

It is important to inspect the polarization on the plane where a symmetry plane feature is applied. The symmetry plane enforces a PEC or PMC condition and thereby suppresses the tangential field component of electric field or magnetic field on the plane, respectively. Some waveguides modes cannot be simplified using symmetry plane features. For instance, a **Zero tangential magnetic field (PMC)** symmetry plane cut at the center of a TE20 rectangular waveguide mode can satisfy the boundary condition of the transverse electric field on a port boundary. However, the longitudinal magnetic field component in the waveguide is incorrectly suppressed by the symmetry plane.

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

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

Electric Point Dipole 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).

For 2D and 2D Axisymmetry, the **Electric Point Dipole** node is only available when the **Electric field components solved for** is set to **Three-component vector** and **In-plane vector**. For 2D, the **Out-of-plane wave number** should be 0. For 2D Axisymmetry, the **Azimuthal mode number** should be 0.

In 2D axisymmetry, enter the **Electric current dipole moment in the z direction** $\mathbf{p}$ (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 Ia.

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

For 2D and 2D Axisymmetry, the **Magnetic Point Dipole** node is only available when the **Electric field components solved for** is set to **Three-component vector** and **Out-of-plane vector**. For 2D, the **Out-of-plane wave number** should be 0. For 2D Axisymmetry, the **Azimuthal mode number** should be 0.

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 nonsinusoidal 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.

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 checkbox 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** checkbox 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** checkbox 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 — **User defined** (the default), **Frequency**, or **Wavelength**. 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 1/8 in 2D and 1/5 in 3D 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 1/8 in 2D and 1/5 in 3D of the entered wavelength.

The maximum mesh element sizes discussed above are used with quadratic shape functions. When linear shape functions are used, 1/2 of the maximum mesh element size for quadratic shape functions are used. Similarly, when cubic shape functions are used, the maximum mesh element size is 2.25 times the maximum mesh element size for quadratic shape functions.

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.

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), **Cubic**, **Quartic**, **Quintic**, **Sextic**, or **Septic**. 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 (Far-Field Domain)

BOUNDARY CONDITIONS

With no surface currents present the boundary conditions

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

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

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

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

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

and is therefore also fulfilled.

These nodes and subnodes are available and described for the Electromagnetic Waves, Frequency Domain:

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

EDGE, POINT, AND PAIR

These edge, point, and pair nodes are available and described for the Electromagnetic Waves, Frequency Domain (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** $\mathbf{P}$ (SI unit: $\mathrm{C/m^2}$).

Remanent Electric Displacement

For **Remanent electric displacement** enter coordinates for the **Remanent electric displacement** $\mathbf{D}_r$ (SI unit: $\mathrm{C/m^2}$). 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**, **Remanent electric displacement**, or **Drude–Lorentz dispersion model** are chosen as the **Electric displacement field model**.

By default, the **Electric 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 $\mathbf{A}$ (SI unit: Wb/m) and its time derivative $\partial\mathbf{A}/\partial t$ (SI unit: V/m). The default values are 0 Wb/m and 0 V/m, respectively.

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 $\mathbf{D}$, 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

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

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. 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 in 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.

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 checkbox 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** checkbox 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** checkbox 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** — **User defined** (the default), **Frequency**, or **Wavelength**. 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 1/8 in 2D and 1/5 in 3D 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 1/8 in 2D and 1/5 in 3D of the entered wavelength.

The maximum mesh element sizes discussed above are used with quadratic shape functions. When linear shape functions are used, 1/2 of the maximum mesh element size for quadratic shape functions are used. Similarly, when cubic shape functions are used, the maximum mesh element size is 2.25 times the maximum mesh element size for quadratic shape functions.

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.

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 $\mathbf{E}$ and the magnetic field $\mathbf{H}$.

- **E in plane (TM wave)** to solve for the electric field vector components in the modeling plane and one magnetic field vector component perpendicular to the plane, assuming that there is no electric field perpendicular to the plane and no magnetic field components in the plane.
- **H in plane (TE wave)** to solve for the magnetic field vector components in the modeling plane and one electric field vector component perpendicular to the plane.

FILTER PARAMETERS FOR ABSORBING LAYERS

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog. In the **Filter Parameters for Absorbing Layers** section you can change and control the values set for the filter used in the Absorbing Layers. The values of the filter parameters defined here are used in all absorbing layers added to the model and they override the value of filter parameters enabled in the Wave Equations node. The default values of the filter parameters α, η_c, and s are set to 0.5, 0.1, and 4, respectively. Inside the absorbing layer, it is important to use a filter that is not too aggressive since this will result in spurious reflections.

For general information about the filter see the Filter Parameters section under Wave Form PDE in the *COMSOL Multiphysics Reference Manual*.

DISCRETIZATION

Select the shape order for the **Electric and magnetic fields** dependent variables (the same order for both fields) — **Linear**, **Quadratic**, **Cubic** (the default), or **Quartic**. 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 vector** $\mathbf{E}$ and for the **Magnetic field vector** $\mathbf{H}$. The name can be changed but the names of fields and dependent variables must be unique within a model.

- Domain, Boundary, and Pair Nodes for the Electromagnetic Waves, Time Explicit 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
- Far-Field Calculation
- Far-Field Domain
- 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 **Electric 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

- Enter a value or an expression for **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`.

Select the **Flux type** — **Lax–Friedrichs** or **Upwind flux**.

Lax–Friedrichs

- Enter an expression for **Lax–Friedrichs flux parameter for E field** τ_E (SI unit: S), the default is $0.5/Z$ for Ampère's law.
- Enter an expression for **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.

Upwind Flux

Enter a **Scaling factor** α_{upwind}, the default is 1 (dimensionless). When it is set to 0, the flux type becomes the central flux.

FILTER PARAMETERS

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

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.

The exponential filter can be described by the matrix formula

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

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

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

where

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

and N_p is the basis function and i_m the polynomial order for coefficient m. Furthermore, α, η_c, and s (for default values, see below) 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. Maximal damping is obtained 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.

Set **Filter settings** to **Automatic** (the default), **Only use filter on absorbing layers**, or **User defined**. When **Automatic** is selected, $\alpha = 0.5$ in **Absorbing Layer** domains and $\alpha = 0.05$ in all other domains, and $\eta_c = 0.01$ and $s = 4$ in all domains. When **Only use filter on absorbing layers** is selected, the filter parameters in **Absorbing Layer** domains are taken from the settings in the Filter Parameters for Absorbing Layers section for The Electromagnetic Waves, Time Explicit Interface, whereas there is no filtering in non-**Absorbing Layer** domains. Finally, when **User defined** is selected, set values for α (default value: 0.05), η_c (default value: 0.1), and s (default value: 4) for non-**Absorbing Layer** domains, whereas the filter parameters in **Absorbing Layer** domains are taken from the settings in the Filter Parameters for Absorbing Layers section.

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

Electric Current Density

The **Electric Current Density** node adds an external current density to the specified domains, which appears on the right-hand side of Ampère'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** $\mathbf{E}_0$ (SI unit: V/m).

Perfect Electric Conductor

The **Perfect Electric Conductor** boundary condition

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

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

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

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** $\mathbf{H}_0$ (SI unit: A/m).

Perfect Magnetic Conductor

The **Perfect Magnetic Conductor** boundary condition

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

is a special case of the surface current density boundary condition that sets the tangential component of the magnetic field and thus also the surface current density to zero. On exterior 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.

SCATTERING BOUNDARY CONDITION

Enter the expressions for the components for the **Incident electric field** $\mathbf{E}_0$ (SI unit: V/m), if there is an incoming wave from the boundary.

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 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** $\mathbf{E}_{\text{b}}$ (SI unit: V/m) and **Background magnetic field** $\mathbf{H}_{\text{b}}$ (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

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

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** checkboxes 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 checkbox 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 checkbox 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** checkbox 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, and so on, 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/Couplers_Filters_and_Mirrors/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/Beam_Propagation/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:

- Layered Transition Boundary Condition
- Transition Boundary Condition
- Field Continuity

These 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 interior and/or exterior 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.

The requirement that the phase is everywhere continuous is not applicable for the **Bidirectional** formulation, when the phase discontinuities occur at boundaries between materials having different optical properties.

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/Beam_Propagation/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.

For an example using the **User Defined Wave Vector** section with a user defined wave vector setting, see *Tapered Waveguide*: Application Library path **Wave_Optics_Module/Waveguides/tapered_waveguide**

PORT SWEEP SETTINGS

Select the **Activate port sweep** checkbox 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.

Select **Weak formulation** (the default) or **Constraint-based** from the **Port formulation** list. For a more detailed discussion of these options, see Port Options in the documentation of The Electromagnetic Waves, Frequency Domain Interface.

LOSS CALCULATION

Select the **Use averaged loss calculation** checkbox 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*.

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

DEPENDENT VARIABLES

The dependent variables (field variables) are for the:

- **Electric field envelope, first wave** **E1** and its components (in the **Electric field envelope components, first wave** fields).
- **Electric field envelope, second wave** **E2** and its components (in the **Electric field envelope components, second wave** fields). The second wave is applicable if the Wave Vectors are bidirectional.

The name can be changed but the names of fields and dependent variables must be unique within a model.

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

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

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

For the unidirectional formulation, the electric field is given by the product of the electric field envelope $\mathbf{E}_1$ and the phase function (see Wave Vectors). Because $\mathbf{E}_1$ is being solved for and the phase function is continuous across boundaries, the tangential component of the electric field is always continuous, and thus the first condition is automatically fulfilled. The second condition is equivalent to the natural boundary condition for the unidirectional formulation

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

and is therefore also fulfilled.

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

The following boundary conditions are available for this 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
- Layered Transition Boundary Condition
- Perfect Electric Conductor
- Perfect Magnetic Conductor
- Periodic Condition
- Port
- Reference Point
- Symmetry Axis Reference Point
- Symmetry Plane
- 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 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

$$(\nabla - j\mathbf{k}_2)\times((\nabla - j\mathbf{k}_2)\times\mathbf{E}_2) - k^2\mathbf{E}_2 = \mathbf{0} \quad ,$$

where

$$\mathbf{k}_2 = \nabla\varphi_2 .$$

When solving the equations as an eigenfrequency problem the eigenvalue is the complex-valued 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: $\mathrm{C/m^2}$) and, for the bidirectional formulation, the **Polarization, second wave** $\mathbf{P}_{i2}$ (SI unit: $\mathrm{C/m^2}$).

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.

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.

CONSTRAINT SETTINGS

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

	Optical Ring Resonator Notch Filter 3D: Application Library path **Wave_Optics_Module/Couplers_Filters_and_Mirrors/ optical_ring_resonator_3d**
	Optical Ring Resonator Notch Filter: Application Library path **Wave_Optics_Module/Couplers_Filters_and_Mirrors/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 there is **No incident field** (the default), the input wave is specified by an **Electric field**, a **Magnetic field**, or a **Gaussian Beam**.

When **No incident field** is selected, 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** $\mathbf{E}_{g0}$ (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 nonnormalized electric field amplitude** $\mathbf{E}_{g0}$ (SI unit: V/m). Also edit the **Beam waist 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).

If the **Incident field** is set to **Gaussian beam** and **Type of phase specification**, in the Wave Vectors section for The Electromagnetic Waves, Beam Envelopes Interface, is set to **User defined**, edit the **Incident wave direction** $\mathbf{k}_{\text{dir}}$ for the vector coordinates. The default direction is in the opposite direction to the boundary normal.

When Wave Vectors is set to bidirectional, if no scattered field is expected, select the **No scattered field** checkbox. This prevents COMSOL from returning spurious

solutions that otherwise could appear between boundaries with unconstrained scattered fields.

	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/Beam_Propagation/brewster_interface**. *Total Internal Reflection*: Application Library path **Wave_Optics_Module/Waveguides/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

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

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

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

The field $\mathbf{E}_0$ is the incident plane wave that travels in the direction $\mathbf{k}$. 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, assuming that the material properties adjacent to the boundary are isotropic. 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 $\mathbf{r}_{\text{ref}}$ 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.

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

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

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

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 there is **No incident field** (the default), 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 nonnormalized electric field amplitude** $\mathbf{E}_{g0}$ (SI unit: V/m). Also edit the **Beam waist 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**.

If the **Incident field** is set to **Gaussian beam** and **Type of phase specification**, in the Wave Vectors section for The Electromagnetic Waves, Beam Envelopes Interface, is set to **User defined**, edit the **Incident wave direction** $\mathbf{k}_{dir}$ for the vector coordinates. The default direction is in the opposite direction to the boundary normal.

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** $\mathbf{r}_o$ (SI unit: m) and **Source axis direction** $\mathbf{r}_{axis}$ (dimensionless).
- For **Spherical wave** enter coordinates for the **Source point** $\mathbf{r}_o$ (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

When **Number of directions** is set to **Unidirectional**, in the Wave Vectors section for The Electromagnetic Waves, Beam Envelopes Interface, enter values or expressions for the components of the **Surface current density** J_{s0} (SI unit: A/m). When **Number of directions** is set to **Bidirectional**, enter values or expressions for the components of the **Surface current density, first wave** $\mathbf{J}_{s01}$ (SI unit: A/m) and 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 checkbox 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** checkbox 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** checkbox 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. 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 in 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 $\mathbf{E}$.
- **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.

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, Electric point dipole

The scattered field formulation supports both metallic PEC scatterers and dielectric scatterers. Notice that a Wave Equation, Electric can be active on multiple domains provided that each material parameter (**Relative permittivity**, **Relative permeability**, and **Electric conductivity**) has the same constant value in all the selected domains. In brief, there needs to be a Wave Equation, Electric node for each dielectric material.

When multiple Wave Equation, Electric nodes exist, the Infinite void selection needs to correspond to the first Wave Equation, Electric node and cannot coexist with other domain selections.

Optical Yagi–Uda Antenna: Application Library path **Wave_Optics_Module/Optical_Scattering/optical_yagi_uda_antenna** demonstrates how to use the **Scattered field** formulation with domain scatterers.

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** checkbox 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, 2·13·13 = 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 waist radius** w_0 (SI unit: m). The default is $20\pi/\texttt{ebem}.k_0$ 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** $\mathbf{E}_{T\text{bg}0}$ (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 **Nonnormalized transverse electric field amplitude, Gaussian beam** $\mathbf{E}_{T\text{bg}0}$ (SI unit: V/m).
- Enter a **Wave number** k (SI unit: rad/m). The default is $\texttt{ebem}.k_0$ 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.

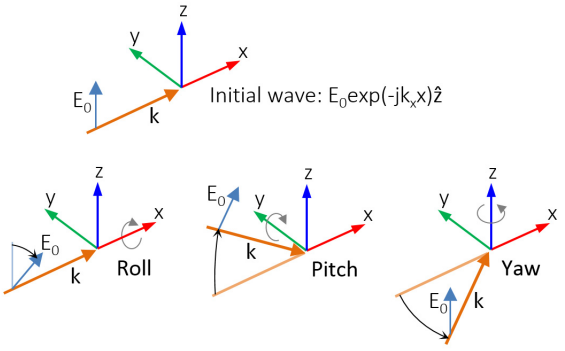


Figure 3-7: Schematic of the directions for the wave vector $\mathbf{k}$*, the electric field* $\mathbf{E}_0$*, and the roll, pitch, and yaw rotations. The top image represents an initial wave propagating in the* x *direction with a polarization along the* z *direction.*

ELECTRIC POINT DIPOLE

For **Electric point dipole**, select the **Electric point dipole type** — **Electric current dipole** (the default) or **Electric charge dipole**.

The background wave for the **Electric current dipole** is defined as

$$\mathbf{E}_b(\mathbf{r}) = \frac{1}{j\omega}\frac{1}{4\pi\varepsilon_r\varepsilon_0}\left\{\frac{k^2}{R}(\hat{\mathbf{R}}\times\mathbf{p}_I)\times\hat{\mathbf{R}}+\left(\frac{1}{R^3}+\frac{jk}{R^2}\right)(3\hat{\mathbf{R}}[\hat{\mathbf{R}}\cdot\mathbf{p}_I]-\mathbf{p}_I)\right\}e^{-jkR}e^{j\omega t}$$

Here, $\hat{\mathbf{R}}$ is the unit vector of $\mathbf{R}$, $\mathbf{R}$ is the distance vector from the dipole position to the field position is space, R is the norm of $\mathbf{R}$, ε_r is the relative permittivity of the background medium, k is the wavenumber in the background medium, $\mathbf{p}_I$ is the electric current dipole moment (SI unit: A·m).

The background wave for the **Electric charge dipole** is defined as

$$\mathbf{E}_b(\mathbf{r}) = \frac{1}{4\pi\varepsilon_r\varepsilon_0}\left\{\frac{k^2}{R}(\hat{\mathbf{R}}\times\mathbf{p}_Q)\times\hat{\mathbf{R}}+\left(\frac{1}{R^3}+\frac{jk}{R^2}\right)(3\hat{\mathbf{R}}[\hat{\mathbf{R}}\cdot\mathbf{p}_Q]-\mathbf{p}_Q)\right\}e^{-jkR}e^{j\omega t}$$

Here, $\mathbf{p}_Q$ is the Electric charge dipole moment (SI unit: C·m).

If the **Electric point dipole type** is **Electric current dipole**, define the **Electric point dipole** using the parameters below:

- Enter the component expressions of **Electric current dipole moment** (SI Unit: A·m).
- Enter the **Dipole position** $\mathbf{r}_0$ (SI unit: m).
- Enter a **Wave number** k (SI unit: rad/m). The default is `ebem`.k0 rad/m.

If the **Electric point dipole type** is **Electric charge dipole**, define the **Electric point dipole** using the parameters below:

- Enter the component expressions of **Electric charge dipole moment** (SI Unit: C·m).
- Enter a **Wave number** k (SI unit: rad/m). The default is `ebem`.k0 rad/m.
- Enter the **Dipole position** $\mathbf{r}_0$ (SI unit: m).

Optical Yagi–Uda Antenna: Application Library path **Wave_Optics_Module/Optical_Scattering/optical_yagi_uda_antenna** demonstrates how to set up the Electric point dipole background field.

SYMMETRY

Symmetry planes are normal to the Cartesian coordinate axes. For **Condition for the $x=x_0$ plane**, **Condition for the $y=y_0$ plane**, and **Condition for the $z=z_0$ plane**, select **Off** (the default), **Zero tangential magnetic field (PMC)**, or **Zero tangential electric field (PEC)**, respectively. For the option **Zero tangential magnetic field (PMC)** or **Zero tangential electric field (PEC)**, enter an axis coordinate value of the symmetry plane (SI unit: m).

STABILIZATION

To display this section, click the **Show More Options** button ( ) and select **Stabilization** in the **Show More Options** dialog.

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.

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.

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** $\mathbf{E}$ 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.

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
- Far-Field Calculation
- Impedance Boundary Condition
- Layered Impedance Boundary Condition
- Perfect Electric Conductor
- Perfect Magnetic 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.

WAVE EQUATION, ELECTRIC

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 Electric Displacement Field section for the Wave Equation, Electric node in The Electromagnetic Waves, Frequency Domain Interface for all settings.

The default **Relative permeability** μ_r, and **Electric 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 **Electric 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** $\boldsymbol{E}$ (SI unit: V/m). The default values are 0 V/m.

Far-Field Calculation

Use a **Far-Field Calculation** to select all boundaries between, on one side, the **Infinite void**, and on the other side, different material domains, voids, or boundaries that scatter or radiate. By default, all boundaries within the simulation domain are selected, while those interior to a material domain are disregarded, marked as **not applicable**.

FAR-FIELD CALCULATION

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

ADVANCED SETTINGS

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

For a discussion about the settings in this section, see Advanced Settings in the documentation for The Electromagnetic Waves, Frequency Domain Interface.

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.

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**.

The Electromagnetic Waves, FEM-BEM Interface

The **Electromagnetic Waves, FEM-BEM** interface ( ) makes it possible to build hybrid FEM-BEM models, where the boundary element method (BEM) is used to compute the electric fields outside the finite element method (FEM) domains. This multiphysics interface adds an Electromagnetic Waves, Frequency Domain interface and an Electromagnetic Waves, Boundary Elements interface. The multiphysics coupling assures continuity of the tangential electric fields across boundaries between the two interfaces.

Frequency-domain and Wavelength-domain modeling are supported in 2D and 3D.

When a predefined **Electromagnetic Waves, FEM-BEM** interface is added from the **Optics** > **Wave Optics** branch ( ) of the **Model Wizard** or **Add Physics** window, the **Electromagnetic Waves, Frequency Domain** and **Electromagnetic Waves, Boundary Elements** interfaces are added to the Model Builder.

In addition, a **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Electric Field Coupling**.

On the Constituent Physics Interfaces

The Electromagnetic Waves, Frequency Domain interface computes time-harmonic electromagnetic field distributions. To use this physics interface, the maximum mesh element size should be limited to a fraction of the wavelength. Thus, the domain size that can be simulated scales with the amount of available computer memory and the wavelength. The physics interface solves the time-harmonic wave equation for the electric field.

The Electromagnetic Waves, Boundary Elements interface is used to solve for time-harmonic electromagnetic field distributions. This interface is especially well suited for radiation and scattering problems since only boundaries need to be meshed and the degrees of freedom (DOFs) solved for are restricted to the boundaries. To use this physics interface, the maximum mesh element size should well resolve the complex electric field on the boundaries. The physics interface solves the vector Helmholtz equation for piecewise-constant material properties and uses the electric field as dependent variable.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURE

When physics interfaces are added using the predefined couplings — for example, **Electromagnetic Waves, FEM-BEM** — 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, Frequency Domain** and **Electromagnetic Waves, Boundary Elements** interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. You can add the **Electromagnetic Waves, FEM-BEM** 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 3-3: MODIFIED SETTINGS FOR AN ELECTROMAGNETIC WAVES, FEM-BEM INTERFACE.

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS (IF ANY)
Electromagnetic Waves, Frequency Domain	No changes.
Electromagnetic Waves, Boundary Elements	No changes.
Electromagnetic Waves, FEM-BEM	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, Frequency Domain** and **Electromagnetic Waves, Boundary Elements** interfaces are preselected in the Coupled Interfaces section (described in the *COMSOL Multiphysics Reference Manual*).

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 Electric Field Coupling feature is described in the documentation for The Electromagnetic Waves, Boundary Elements Interface.

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, Frequency Domain Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Frequency Domain Interface.
- The available physics features for The Electromagnetic Waves, Boundary Elements Interface are listed in the section Domain, Boundary, Edge, Point, and Pair Nodes for the Electromagnetic Waves, Boundary Elements 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
- Linearly Polarized Plane Wave as Background Field in 2D Axisymmetry
- Periodic Port Mode 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 is 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

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

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 3-2, the following wave equation for the electric field envelope $\mathbf{E}_1$ is obtained

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

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 3-3 than when solving Equation 3-2. 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 3-3. Thus, the solution of Equation 3-3 is as exact as the solution of Equation 3-2.

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

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

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

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

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

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

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

Covariant formulation

In the 2D axisymmetric formulation, it is beneficial to formulate the out-of-plane dependent variable as

$$\Psi = rE_\varphi,$$

referred to as the covariant formulation. Here, Ψ is the dependent variable and the out-of-plane electric field component is calculated as

$$E_\varphi = \Psi / r.$$

The out-of-plane dependent variable is constrained to be zero on the symmetry axis,

$$\Psi = 0.$$

The covariant formulation has better performance in terms of numerical stability and accuracy. For eigenfrequency simulations, it also removes spurious solutions.

This formulation is used for all study types, except Mode Analysis and Boundary Mode Analysis.

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 $\boldsymbol{E}_z$,

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

where

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

To be able to write the fields in this form, it is also required that ε_r, σ, and μ_r are 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 $\boldsymbol{E}_\varphi$. 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 (material parameters ε'' and κ) 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 waist 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

$$\eta(z) = \operatorname{atan}\left(\frac{z - p_0}{z_0}\right).$$

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.

Linearly Polarized Plane Wave as Background Field in 2D Axisymmetry

When solving for the scattered field in 2D axisymmetry, the background wave type can be set to a linearly polarized plane wave propagating in arbitrary direction in the **Settings** of The Electromagnetic Waves, Frequency Domain Interface. The linearly polarized plane wave is of the form $\mathbf{E}_\mathrm{b} = \mathbf{E}_0 e^{j(\omega t - (\mathbf{k}\cdot\mathbf{r}))}$. In Cartesian coordinates, $\mathbf{E}_0 = (E_x, E_y, E_z)$, $\mathbf{k} = (k_x, k_y, k_z)$, and $\mathbf{r} = (x, y, z)$. To express a linearly polarized plane wave with arbitrary incident angle and polarization angle in cylindrical coordinates for a 2D axisymmetric simulation, use the following expansions:

$$e^{-jkr\sin\theta\cos\phi} = \sum_{m=-\infty}^{\infty}(-j)^m J_m(kr\sin\theta)e^{-jm\phi},$$

where θ is the angle with respect to the positive z-axis, ϕ is the azimuthal angle, m is the azimuthal mode number, and J_m is the Bessel function of the first kind of order

m. Furthermore, the basis vectors $\hat{\mathbf{x}}$ and $\hat{\mathbf{y}}$ in Cartesian coordinates can be expressed with basis vectors $\hat{\mathbf{r}}$ and $\hat{\phi}$ as

$$\hat{\mathbf{x}} = \frac{1}{2}[e^{j\phi}(\hat{\mathbf{r}} + j\hat{\phi}) + e^{-j\phi}(\hat{\mathbf{r}} - j\hat{\phi})]$$

and

$$\hat{\mathbf{y}} = \frac{1}{2}[e^{j\phi}(\hat{\phi} - j\hat{\mathbf{r}}) + e^{-j\phi}(\hat{\phi} + j\hat{\mathbf{r}})]\,.$$

Consequently, a plane wave background field with amplitude E_0, incident angle θ, and polarization angle α can be written as (E_r, E_ϕ, E_z), where

$$E_r = \frac{E_0}{2}e^{jkz\cos\theta}\sum_{m=-\infty}^{\infty}[(\cos\alpha\cos\theta - j\sin\alpha)(-j)^{m+1}J_{m+1}(kr\sin\theta) + (\cos\alpha\cos\theta + j\sin\alpha)(-j)^{m-1}J_{m-1}(kr\sin\theta)]e^{-jm\phi},$$

$$E_\phi = \frac{E_0}{2}e^{jkz\cos\theta}\sum_{m=-\infty}^{\infty}[(\sin\alpha + j\cos\alpha\cos\theta)(-j)^{m+1}J_{m+1}(kr\sin\theta) + (\sin\alpha - j\cos\alpha\cos\theta)(-j)^{m-1}J_{m-1}(kr\sin\theta)]e^{-jm\phi},$$

and

$$E_z = E_0\cos\alpha\sin\theta e^{jkz\cos\theta}\sum_{m=-\infty}^{\infty}(-j)^m J_m(kr\sin\theta)e^{-jm\phi}\,.$$

Here, θ and α are defined as the schematic shown in Figure 3-2. Once the linearly polarized plane wave is used as the background field, an auxiliary sweep over the azimuthal mode number will be added. After the simulation, the total scattered field is given by the sum of all the azimuthal modes. The z-components of the scattered field and the background field will be plotted by default. Other components of the field can be computed in a similar way with the help of the `sum` and `withsol` operators.

Periodic Port Mode Fields

Periodic, Diffraction Order, and Orthogonal Polarization Port features all use plane-wave electric mode fields of the form

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

where $\mathbf{E}_m$, $\mathbf{k}_m$, and $\mathbf{r}$ are the amplitude, the wave vector, and the position vector, respectively. Here, m is the mode index. Since this field represents a plane wave, the amplitude must be orthogonal to the wave vector,

$$\mathbf{E}_m \cdot \mathbf{k}_m = 0.$$

As plane-wave mode fields are assumed, the material properties in the domain adjacent to the port boundary must be homogeneous and isotropic.

As discussed in Additional Variables for Periodic Structure Calculations, the wave vector for the plane wave for mode m is defined by

$$\mathbf{k}_m = k(\sin\alpha_{1m}(\cos\alpha_{2m}\mathbf{a}_1 + \sin\alpha_{2m}\mathbf{a}_2) + \cos\alpha_{1m}\mathbf{n}),$$

where k is the wave number for the domain adjacent to the port boundary, α_{1m} and α_{2m} are the elevation and azimuth angles for mode m, $\mathbf{a}_1$ and $\mathbf{a}_2$ are the reference axes, and $\mathbf{n}$ is the port boundary normal.

A linearly polarized plane wave can have one of two polarization states — either s or p polarization. For s polarization, the electric field is polarized in a direction orthogonal to the plane of incidence, spanned by the wave vector and the port boundary normal. Thus, the unit polarization vector for s polarization can be written

$$\mathbf{u}_{sm} = \frac{\mathbf{k}_m \times \mathbf{n}}{|\mathbf{k}_m \times \mathbf{n}|} = \cos\alpha_{2m}\mathbf{a}_1 \times \mathbf{n} + \sin\alpha_{2m}\mathbf{a}_2 \times \mathbf{n}.$$

For p polarization, the electric field must be orthogonal both to s polarization and the wave vector. Thus, the p unit polarization vector can be written as

$$\mathbf{u}_{pm} = \frac{\mathbf{u}_{sm} \times \mathbf{k}_m}{|\mathbf{u}_{sm} \times \mathbf{k}_m|} = -\cos\alpha_{1m}(\cos\alpha_{2m}\mathbf{a}_1 + \sin\alpha_{2m}\mathbf{a}_2) + \sin\alpha_{1m}\mathbf{n}.$$

From the definitions of the two polarization vectors, it is clear that the cross product of the p and s polarization vectors gives a vector in the direction of the wave vector.

For right-handed circularly polarized (RHCP) light, if you align the right hand thumb with the wave vector, the remaining fingers should point in the rotation direction when time changes, but the position is fixed. Thus, defining the RHCP polarization vector as

$$\mathbf{u}_{rm} = \mathbf{u}_{sm} + i\mathbf{u}_{pm},$$

will give the following temporal evolution of the polarization vector

$$\text{real}\{\mathbf{u}_{rm}\exp(j\omega t)\} = \mathbf{u}_{sm}\cos\omega t - \mathbf{u}_{pm}\sin\omega t.$$

As seen from this expression, at t = 0, polarization is along the s direction. One quarter of a period later, the polarization is along the negative p direction. This means that the rotation direction is from (positive) p to (positive) s direction. So, the definition for the RHCP polarization vector above satisfies the RHCP definition.

For left-handed circularly polarized light (LHCP), the polarization vector is defined by

$$\mathbf{u}_{lm} = \mathbf{u}_{sm} - i\mathbf{u}_{pm}.$$

For the Periodic Port, when **Polarization** is set to **User defined**, the amplitude $\mathbf{E}_0$ is provided by the user (we set $m = 0$ here, as the Periodic Port represents the lowest diffraction order). The amplitude $\mathbf{E}_0$ can be expressed in terms of the two linear polarization vectors,

$$\mathbf{E}_0 = E_{0s}\mathbf{u}_{sm} + E_{0p}\mathbf{u}_{pm}.$$

The amplitude for the Orthogonal Polarization Port is orthogonal to the amplitude of the Periodic Port. That is,

$$\mathbf{E}_{\text{orth}} \cdot (\mathbf{E}_0)^* = E_{\text{orth}_s}(E_{0s})^* + E_{\text{orth}_p}(E_{0p})^* = 0,$$

where $E_{\text{orth}s}$ and $E_{\text{orth}p}$ are the expansion coefficients for the orthogonal polarization mode field $\mathbf{E}_{\text{orth}}$ and $(\mathbf{E}_0)^*$ means conjugation of the amplitude $\mathbf{E}_0$. To satisfy the equation above, $\mathbf{E}_{\text{orth}}$ can be defined as

$$\mathbf{E}_{\text{orth}} = (E_{0p})^*\mathbf{u}_{sm} - (E_{0s})^*\mathbf{u}_{pm}.$$

For Diffraction Order ports, the amplitude for out-of-plane modes represents s polarization and the amplitude for in-plane modes represents p polarization.

The mode fields described above are the unnormalized mode fields. The normalized mode fields are scaled to produce a mode power that equals the specified (port) input power.

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

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

$$\begin{aligned} \nabla \times \mathbf{H} &= \mathbf{J} + \frac{\partial \mathbf{D}}{\partial t} \\ \nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t} \\ \nabla \cdot \mathbf{D} &= \rho \\ \nabla \cdot \mathbf{B} &= 0 \end{aligned} \tag{3-4}$$

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

$$\begin{aligned} \mathbf{D} &= \varepsilon_0 \mathbf{E} + \mathbf{P} \\ \mathbf{B} &= \mu_0(\mathbf{H} + \mathbf{M}) \\ \mathbf{J} &= \sigma \mathbf{E} \end{aligned} \tag{3-5}$$

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

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

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

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

To get a wave equation for the $\mathbf{E}$ field, for example, take the curl of the second equation in Equation 3-4 (previously divided by μ_0), and insert it into the time derivative of the first row in Equation 3-4

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

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 3-5 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 3-4 can be simplified to Maxwell–Ampère's law and Faraday's law:

$$\nabla \times \mathbf{H} = \sigma \mathbf{E} + \varepsilon_0 \varepsilon_r \frac{\partial \mathbf{E}}{\partial t}$$
$$\nabla \times \mathbf{E} = -\mu_0 \mu_r \frac{\partial \mathbf{H}}{\partial t} \tag{3-6}$$

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

$$\nabla \times \mathbf{u} = \nabla \cdot \begin{bmatrix} 0 & u_3 & -u_2 \\ -u_3 & 0 & u_1 \\ u_2 & -u_1 & 0 \end{bmatrix} \tag{3-7}$$

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

$$\begin{aligned} \varepsilon_0\varepsilon_r\frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma \cdot \mathbf{E} \\ \mu_0\mu_r\frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= 0 \end{aligned} \tag{3-8}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} 0 & -h_3 \\ h_3 & 0 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} e_2 & -e_1 \end{bmatrix} \tag{3-9}$$

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

$$\begin{aligned} \varepsilon_0\varepsilon_r\frac{\partial \mathbf{E}}{\partial t} + \nabla \cdot \Gamma_E(\mathbf{H}) &= -\sigma\mathbf{E} \\ \mu_0\mu_r\frac{\partial \mathbf{H}}{\partial t} + \nabla \cdot \Gamma_H(\mathbf{E}) &= 0 \end{aligned} \tag{3-10}$$

with the flux terms

$$\Gamma_E(\mathbf{H}) = \begin{bmatrix} -h_2 & h_1 \end{bmatrix} \text{ and } \Gamma_H(\mathbf{E}) = \begin{bmatrix} 0 & e_3 \\ -e_3 & 0 \end{bmatrix} \tag{3-11}$$

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 exterior 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 exterior 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** checkbox 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 Heat Transfer 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**.

Thermal Heating of a Semiconductor Saturable Absorber Mirror (SESAM): Application Library path **Wave_Optics_Module/Couplers_Filters_and_Mirrors/sesam_laser_heating** demonstrates how to use the **Laser Heating** interface.

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.
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 Heat Transfer 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 $\mathbf{D}$ and $\mathbf{E}$ fields and between the $\mathbf{B}$ and $\mathbf{H}$ 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 electric conductivity, modeled as a boundary where the electric field is zero.

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

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

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