

INTRODUCTION TO Wave Optics Module

Introduction to the Wave Optics Module

© 1998–2018 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

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

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

Version: COMSOL 5.4

Contact Information

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

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

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM023502

Contents

| | |
|---|----|
| Introduction | 5 |
| The Use of the Wave Optics Module | 6 |
| The Wave Optics Module Physics Interfaces. | 13 |
| Physics Interface Guide by Space Dimension and Study Type . . . | 17 |
| Introduction. | 17 |
| Model Definition. | 18 |
| Results and Discussion | 19 |
| Reference. | 25 |

Introduction

The Wave Optics Module is used by engineers and scientists to understand, predict, and design electromagnetic wave propagation and resonance effects in optical applications. Simulations of this kind result in more powerful and efficient products and engineering methods. It allows its users to quickly and accurately predict electromagnetic field distributions, transmission and reflection coefficients, and power dissipation in a proposed design. Compared to traditional prototyping, it offers the benefits of lower cost and the ability to evaluate and predict entities that are not directly measurable in experiments. It also allows the exploration of operating conditions that would destroy a real prototype or be hazardous.

This module covers electromagnetic fields and waves in two-dimensional and three-dimensional spaces. All modeling formulations are based on Maxwell's equations together with material laws for propagation in various media. The modeling capabilities are accessed via predefined physics interfaces, collectively referred to as Wave Optics interfaces, which allow you to set up and solve electromagnetic models. The Wave Optics interfaces cover the modeling of electromagnetic fields and waves in frequency domain, time domain, eigenfrequency, and mode analysis.

Under the hood, the Wave Optics interfaces formulate and solve the differential form of Maxwell's equations together with the initial and boundary conditions. The equations are solved using the finite element method with numerically stable edge element discretization in combination with state-of-the-art algorithms for preconditioning and solution of the resulting sparse equation systems. The results are presented using predefined plots of electric and magnetic fields, reflectances, transmittances, diffraction efficiencies, S-parameters, power flow, and dissipation. You can also display your results as plots of expressions of the physical quantities that you define freely, or as tabulated derived values obtained from the simulation.

The work flow is straightforward and can be described by the following steps: define the geometry, select materials, select a suitable Wave Optics interface, define boundary and initial conditions, define the finite element mesh, select a solver, and visualize the results. All these steps are accessed from the COMSOL Desktop. The solver selection step is usually carried out automatically using default settings, which are tuned for each specific Wave Optics interface.

The Wave Optics Module's application library describes the physics interfaces and their different features through tutorial and benchmark examples for the different formulations. The library includes examples addressing gratings and metamaterials, nonlinear optics, optical scattering, waveguides and couplers, and benchmark models for verification and validation of the Wave Optics interfaces.

This introduction is intended to give you a jump start in your modeling work. It has examples of the typical use of the Wave Optics Module, a list of the physics interfaces with a short description, and a tutorial example that introduces the modeling workflow.

The Use of the Wave Optics Module

The Wave Optics interfaces are used to model electromagnetic fields and waves in optical applications. Typical wavelengths for optical applications are in the nanometer to micrometer range, corresponding to frequencies of the order of hundreds of THz. A characteristic of optical applications is also that the structures are frequently much larger than the wavelength.

Wave Optics simulations are often used for determining propagation and coupling properties for different types of waveguide structures. Figure 1 shows the electric field distribution in a directional coupler. A wave is launched into the left waveguide. The wave couples over to the right waveguide after 2 mm propagation. The waveguides consist of ion-bombarded GaAs, surrounded by GaAs.

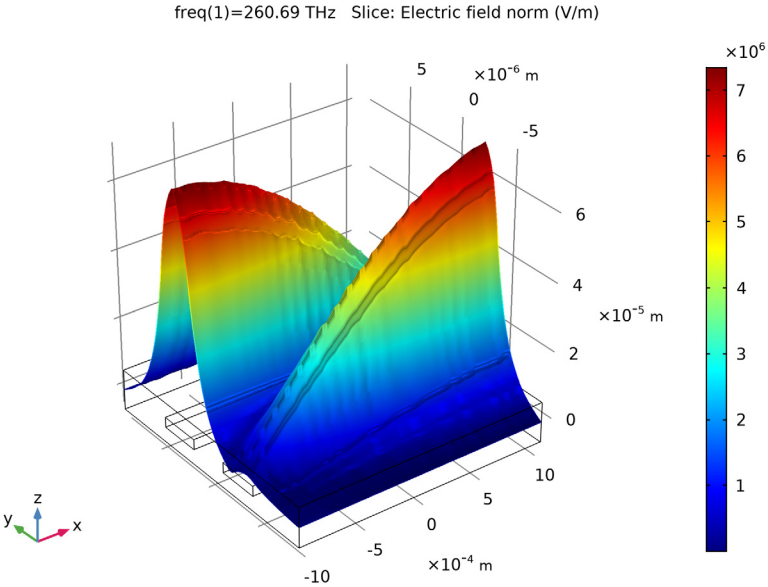


Figure 1: Electric field distribution in a directional coupler. Notice that the propagation length is 2 mm, whereas the cross-sectional area is $12 \mu\text{m}$ by $18 \mu\text{m}$. From Directional Coupler in the Wave Optics Module application library.

Figure 2 shows an example of transient propagation in a nonlinear crystal. The figure shows the total electric field, the sum of the incoming fundamental wave and the generated second harmonic wave, when the wave is located in the middle of the crystal.

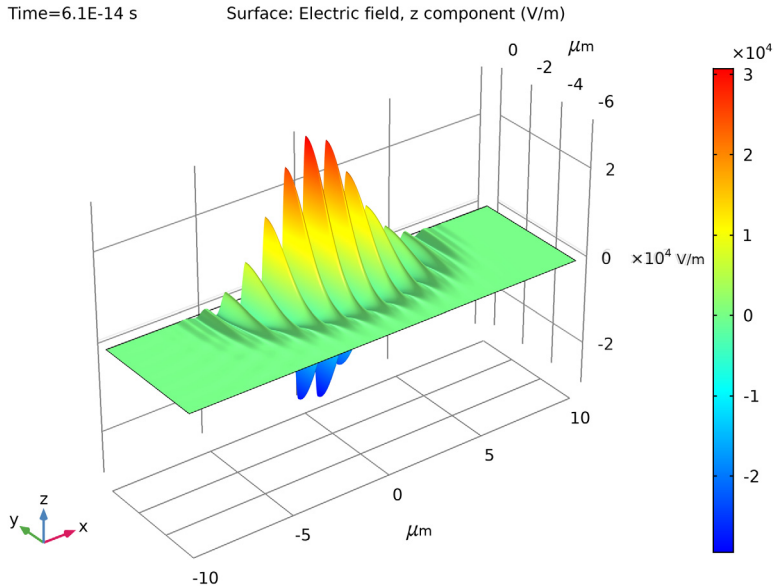


Figure 2: The z-component of the electric field after 61 fs propagation in a nonlinear crystal. From Second Harmonic Generation of a Gaussian Beam in the Wave Optics Module application library.

Transient simulations are useful for modeling nonlinear optical processes involving short optical pulses. Another example of nonlinear optical propagation is the effect of self-focusing, as shown in Figure 3. Here, the Gaussian beam and

the intensity-dependent refractive index forms a self-induced lens in the material that counteracts the spreading effect of diffraction.

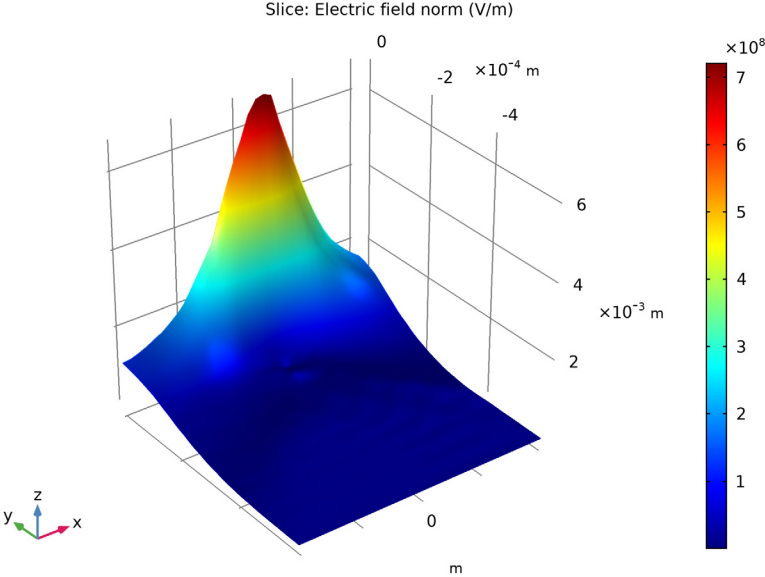


Figure 3: The electric field distribution for a Gaussian beam propagating in a medium with an intensity-dependent refractive index. From Self-Focusing in the Wave Optics Module application library.

In [Figure 4](#) and [Figure 5](#), a model from the Wave Optics Module application library shows the scattering of an incoming plane wave by a small gold sphere. The model is setup using the scattered field formulation, where the incoming plane wave is entered as a background field. The scattered wave is absorbed by a Perfectly

Matched Layer (PML). Figure 4 shows the volume resistive losses in the gold sphere.

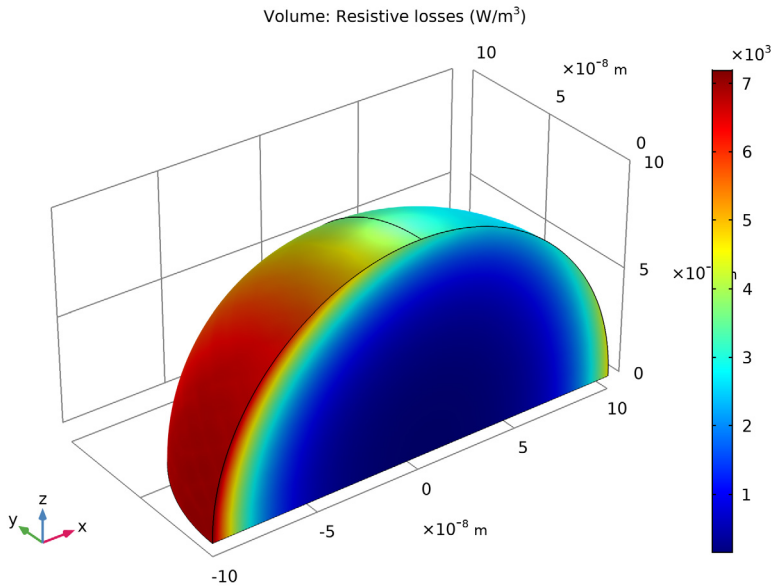


Figure 4: The volume resistive losses in a small gold sphere, when excited by an incoming plane wave. From *Optical Scattering Off a Gold Nanosphere* in the Wave Optics Module application library.

A Far-Field Domain is used in the model to calculate the far-field pattern of the scattered waves, as shown in [Figure 5](#).

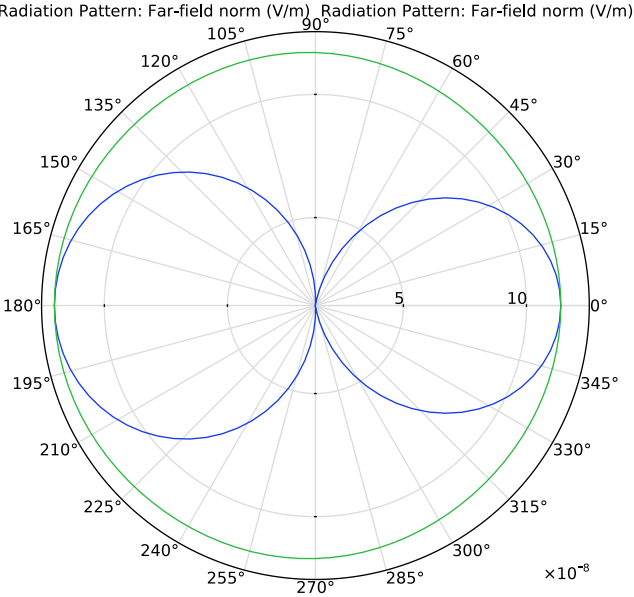


Figure 5: The far-field radiation pattern in the E-plane (blue) and H-plane (green) when wavelength is 700 nm.

The Wave Optics Module also offers a comprehensive set of features for 2D modeling including both source driven wave propagation and mode analysis. [Figure 6](#) shows mode analysis of a step-index profile optical fiber.

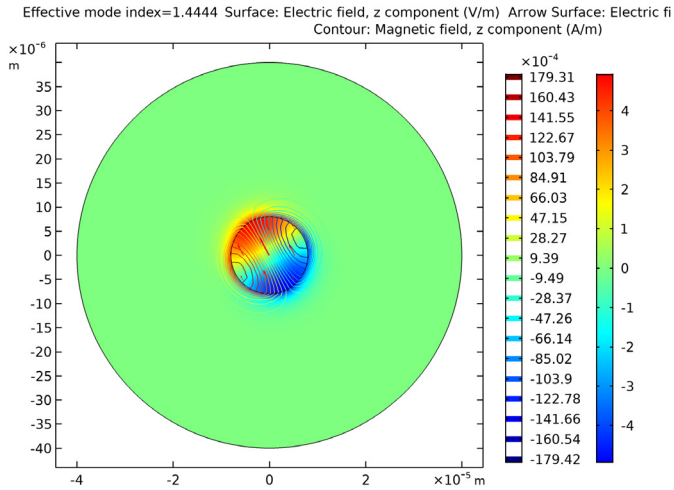


Figure 6: The surface plot visualizes the longitudinal component of the electric field in the fiber core. From Step Index Fiber Bend in the Wave Optics Module application library.

Both in 2D and 3D, the analysis of periodic structures is popular. [Figure 7](#) is an example of a plane wave incident on a metallic wire grating with a dielectric substrate.

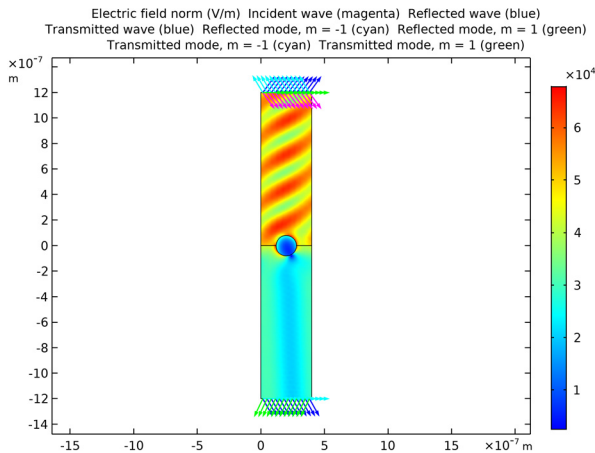


Figure 7: Electric field norm for TE incidence at $\pi/5$ radians. From Plasmonic Wire Grating in the Wave Optics Module application library.

Figure 8 shows the resulting plot of the reflectances, transmittances and diffraction efficiencies.

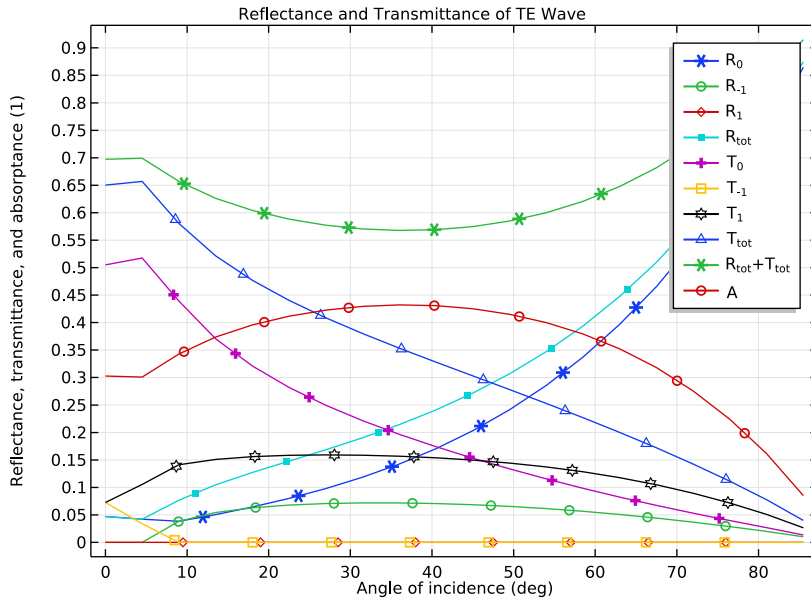


Figure 8: Reflectance, transmittance and diffraction efficiencies as a function of angle of incidence. From *Plasmonic Wire Grating in the Wave Optics Module application library*.

The Wave Optics Module has a vast range of tools to evaluate and export the results, for example, evaluation of far-field and scattering matrices (S-parameters). S-parameters can be exported in the Touchstone file format.

The Wave Optics Module Physics Interfaces

The Wave Optics interfaces are based upon Maxwell's equations together with material laws. In the module, these laws of physics are translated by the Wave Optics interfaces to sets of partial differential equations with corresponding initial and boundary conditions.

The Wave Optics interfaces define a number of features. Each feature represents a term or condition in the underlying Maxwell-based formulation and may be defined in a geometric entity of the model, such as a domain, boundary, edge (for 3D components), or point.

[Figure 9](#) uses the Plasmonic Wire Grating application from the Wave Optics Module application library to show the Model Builder window and the Settings window for the selected Wave Equation, Electric 1 feature node. The Wave Equation, Electric 1 node adds terms representing the Helmholtz wave equation to the model equations for, in this case, all domains in the model.

Furthermore, the Wave Equation, Electric 1 feature node may link to the Materials feature node to obtain physical properties such as relative permittivity and/or refractive index—in this case the refractive index of a user-defined dielectric. The properties, defined by the Dielectric material, can be functions of the modeled physical quantities, such as temperature. Also the first material, Air, is in this case a user-defined material, whereas the third material, the gold wire (labeled Au (Rakic)), is taken from the Optical Materials Database. Since all three materials define the refractive index, only one wave equation feature is needed for this application.

The simulation domain is delimited by boundary condition feature nodes. The default boundary condition feature for the physics interfaces in the Wave Optics Module is the Perfect Electric Conductor feature. For the example in [Figure 9](#) the Perfect Electric Conductor 1 feature node is overridden by two Port feature nodes and a Periodic Condition feature node. The Port features are used for exciting and absorbing waves and the Periodic Condition relates boundaries with periodicity

conditions (for instance continuity, antiperiodicity, and general Floquet phase relationships).

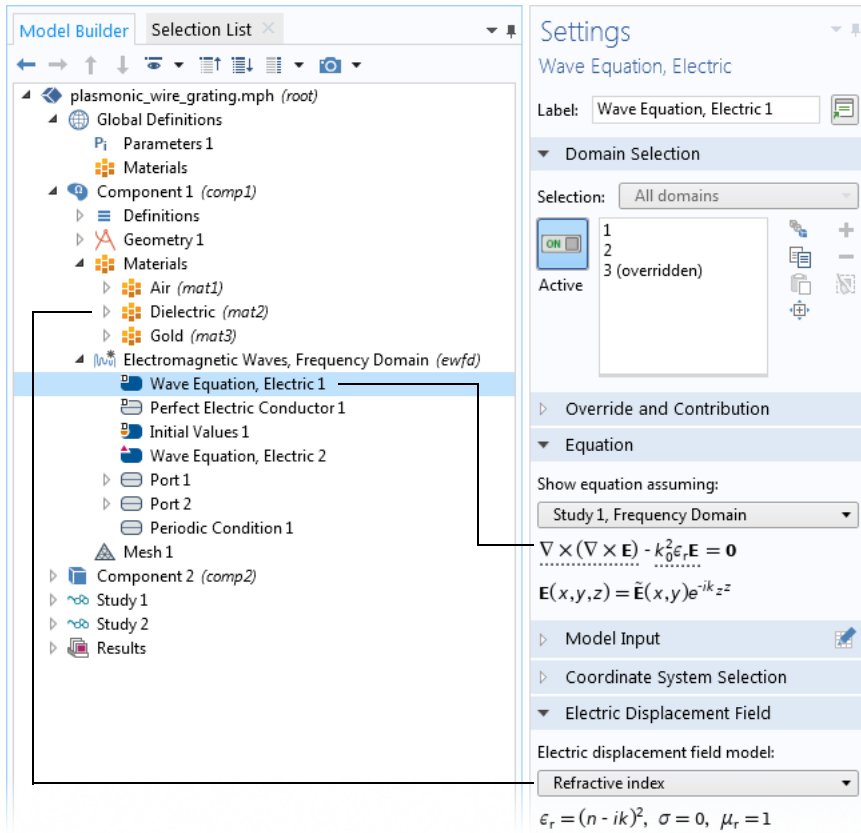


Figure 9: The Model Builder (left), and the Settings window for Wave Equation, Electric (right). The Equation section shows the model equations and the terms added by the Wave Equation, Electric 1 node to the model equations. The added terms are underlined with a dotted line. The text also explains the link between the Dielectric node and the values for the refractive index.

Figure 10 shows the Wave Optics interfaces as displayed in the Model Wizard for this module.

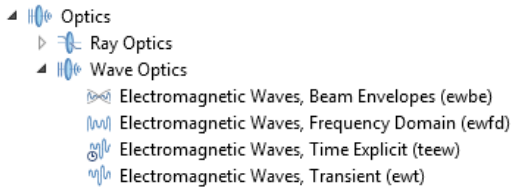




Figure 10: The Wave Optics Module physics interfaces as displayed in the Model Wizard.

This module includes Wave Optics interfaces () for frequency-domain modeling and time-domain modeling, respectively. It also includes the Laser Heating interface, available under Heat Transfer. Also see [Physics Interface Guide by Space Dimension and Study Type](#).


With the addition of the Semiconductor Module license, you also get the Semiconductor Optoelectronics, Beam Envelopes and The Semiconductor Optoelectronics, Frequency Domain interfaces that model the interaction of electromagnetic waves with semiconductors.

A brief overview of the Wave Optics interfaces follows.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN


The Electromagnetic Waves, Frequency Domain interface () solves a frequency-domain wave equation for the electric field. The sources can be in the form of point dipoles, line currents, or incident fields on boundaries or domains. It is used primarily to model electromagnetic wave propagation in different media and structures. Variants of the formulation solves an eigenvalue problem to find the eigenfrequencies of a structure or, at a prescribed frequency, solves an eigenvalue problem to find the propagating modes in waveguides and transmission lines. Some typical applications that are simulated with the interface are waveguides, gratings, and scattering from small particles.

ELECTROMAGNETIC WAVES, BEAM ENVELOPES


The Electromagnetic Waves, Beam Envelopes interface () solves one or two frequency-domain wave equations for the electric field envelope(s). The electric field is represented as the product of the solved for electric field envelope and a rapidly varying prescribed phase function. As the electric field envelopes has a slower spatial variation than the electric field, a coarse mesh can be used. Thus, the Electromagnetic Waves, Beam Envelopes interface is suitable for simulations of optically large structures (structures that are much larger than the wavelength). The sources can be in the form of incident fields on boundaries, surface current,

electric or magnetic fields on boundaries. The interface can be used for propagation problems at a fixed frequency and for finding eigenfrequencies in a resonant structure. Some typical applications that are simulated with the interface are waveguide structures, like directional couplers, nonlinear optical phenomena, and laser beam propagation.


ELECTROMAGNETIC WAVES, TIME EXPLICIT

The Electromagnetic Waves, Time Explicit interface () solves a system of 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. The sources can be in the form of volumetric electric or magnetic currents or electric surface currents or fields on boundaries. It is used primarily to model electromagnetic wave propagation in linear media. Typical applications involve the transient propagation of electromagnetic pulses.

ELECTROMAGNETIC WAVES, TRANSIENT

The Electromagnetic Waves, Transient interface () solves a time-domain wave equation for the electric field. The sources can be in the form of point dipoles, line currents, or incident fields on boundaries or domains. It is used primarily to model electromagnetic wave propagation in different media and structures when a time-domain solution is required—for example, for non-sinusoidal waveforms or for nonlinear media. Typical applications involve the propagation of electromagnetic pulses and the generation of harmonics in nonlinear optical media.

LASER HEATING

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.

Physics Interface Guide by Space Dimension and Study Type

The table below lists the physics interfaces available specifically with this module in addition to the COMSOL Multiphysics basic license. Tutorial Example: Directional Coupler

Introduction

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

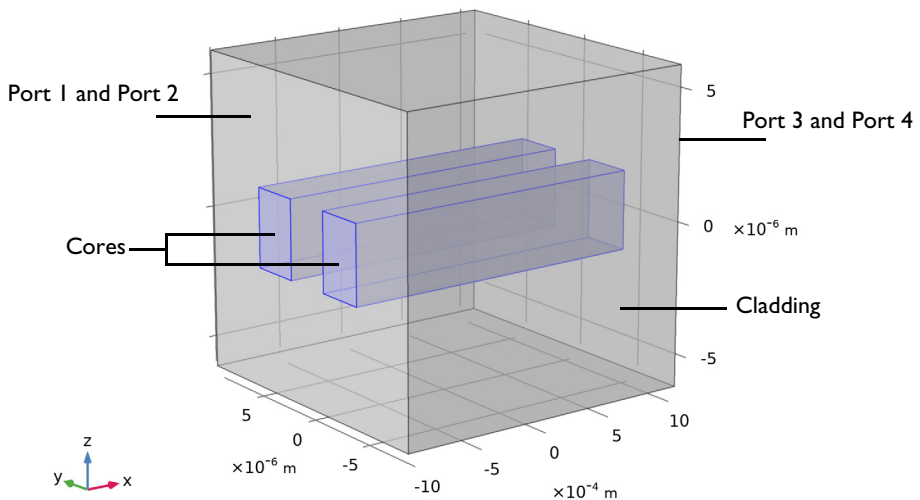


Figure 11: Schematic drawing of the waveguide structure. The structure consists of the two waveguide cores and the surrounding cladding. Port 1 and 2 are used for exciting the waveguides and Port 3 and 4 absorb the waves. Notice that the waveguide structure is not drawn to scale. The length of the waveguides is around 2 mm, whereas the waveguide cross-section is square with a side length of $3 \mu\text{m}$. The waveguide separation is $3 \mu\text{m}$.

Light that propagates through a dielectric waveguide has most of the power concentrated within the central core of the waveguide. Outside the waveguide core, in the cladding, the electric field decays exponentially with the distance from the core. However, if you put another waveguide core close to the first waveguide (see Figure 12), that second waveguide will perturb the mode of the first waveguide (and vice versa). Thus, instead of having two modes with the same effective index, one localized in the first waveguide and the second mode in the

second waveguide, the modes and their respective effective indices split and you get a symmetric supermode (see [Figure 12](#) and [Figure 14](#) below), with an effective index that is slightly larger than the effective index of the unperturbed waveguide mode, and an antisymmetric supermode (see [Figure 13](#) and [Figure 15](#)), with an effective index that is slightly lower than the effective index of the unperturbed waveguide mode.

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

Model Definition

The directional coupler, as shown in [Figure 11](#), consists of two waveguide cores embedded in a cladding material. The cladding material is GaAs, with ion-implanted GaAs for the waveguide cores. The structure is modeled after [Ref. 1](#). The core cross-section is square, with a side length of 3 μm . The two waveguides are separated 3 μm . The length of the waveguide structure is 2 mm. Thus, given the narrow cross-section, compared to the length, it is advantageous to use a view that don't preserve the aspect ratio for the geometry.

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

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

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

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

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

or

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

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

The model uses two numeric ports per input and exit boundary (see [Figure 11](#)). The two ports define the lowest symmetric and antisymmetric modes of the waveguide structure.

In the last part of the modeling procedure, the bidirectional formulation is used. In this case, the two wave vectors are codirectional — they point in the same direction. However, the magnitude of the wave vectors are given by the propagation constants of the two beating modes. Thus, you expect the two waves to have almost constant amplitudes, so the mesh can be very coarse in the propagation direction.

Results and Discussion

[Figure 12](#) to [Figure 15](#) show the results of the initial boundary mode analysis. The first two modes (those with the largest effective mode index) are both symmetric. [Figure 12](#) shows the first mode. This mode has the transverse polarization

component along the z -direction. The second mode, shown in Figure 14, has transverse polarization along the y -direction.

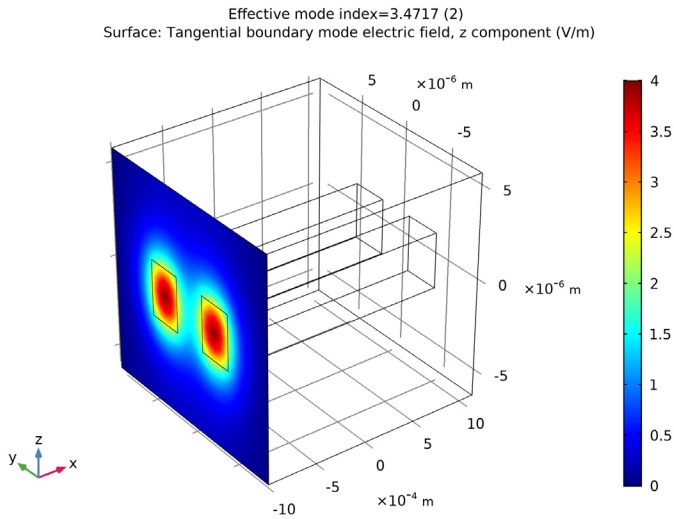


Figure 12: The symmetric mode for z -polarization. Notice that the returned solution can also show the electric field as positive values in the peaks at the cores.

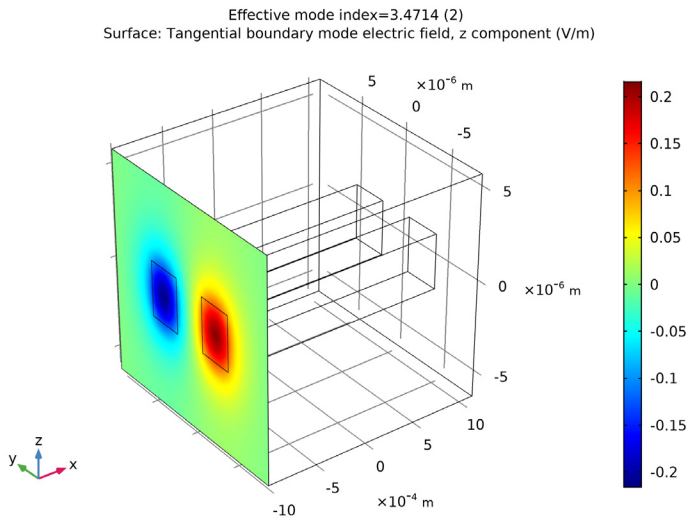


Figure 13: The antisymmetric mode for z -polarization.

Figure 13 and Figure 15 show the antisymmetric modes. Those have effective indices that are slightly smaller than those of the symmetric modes. Figure 13 shows the mode for z -polarization and Figure 15 shows the mode for y -polarization.

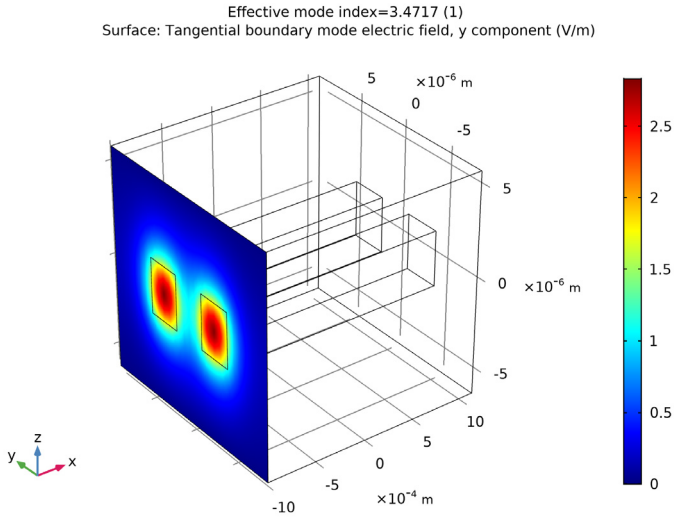


Figure 14: The symmetric mode for y -polarization. Notice that the returned solution can also show the electric field as positive values in the peaks at the cores.

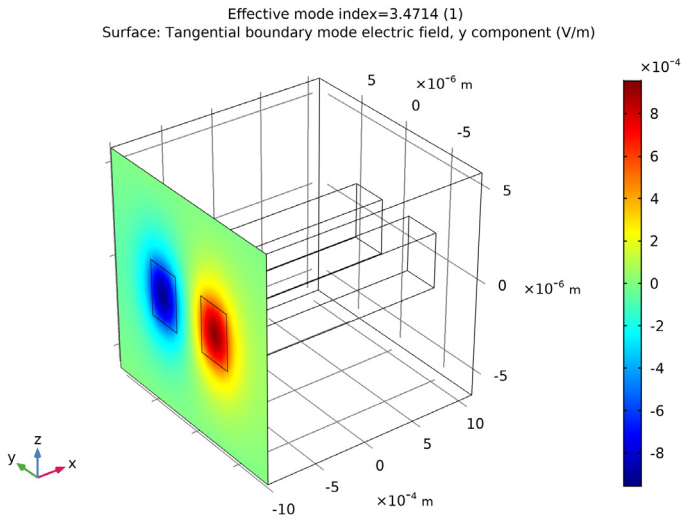


Figure 15: The antisymmetric mode for y -polarization.

Figure 16 shows how the electric field increases in the receiving waveguide and decreases in the exciting waveguide. If the waveguide had been longer, the waves would switch back and forth between the waveguides.

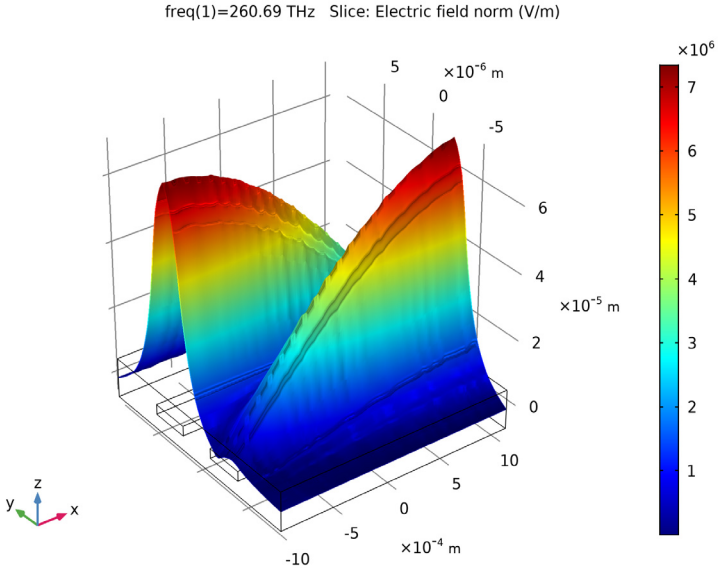


Figure 16: Excitation of the symmetric and the antisymmetric modes. The wave couples from the input waveguide to the output waveguide. Notice that your result may show that the wave is excited in the other waveguide core, if your mode fields have different signs than what is displayed in Figure 12 to Figure 15.

Figure 17 shows the result, when there is a π phase difference between the fields of the exciting ports. In this case, the superposition of the two modes results in excitation of the other waveguides (as compared to the case in Figure 16).

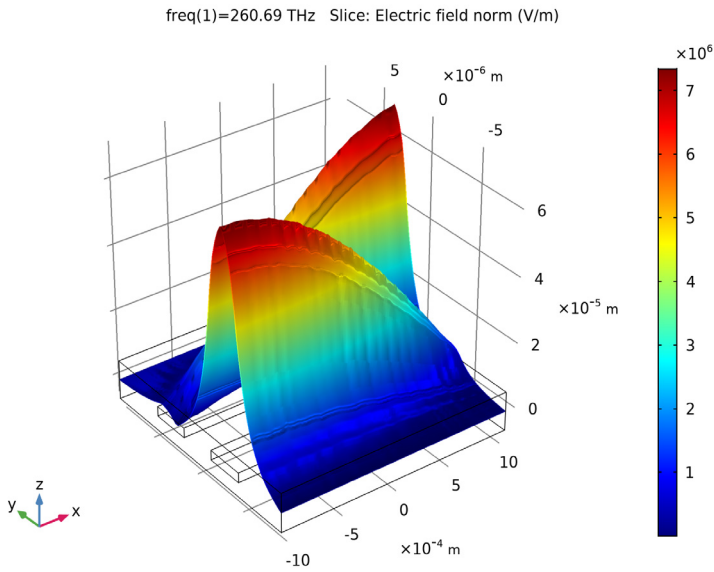


Figure 17: The same excitation conditions as in Figure 16, except that there is a phase difference between the two ports of π radians. Notice that your result may show that the wave is excited in the other waveguide core, if your mode fields have different signs than what is displayed in Figure 12 to Figure 15.

Figure 18 and Figure 19 display the amplitudes of the first and second wave, respectively, when the bidirectional formulation is used. As expected, the amplitudes are almost constant.

freq(1)=260.69 THz

Slice: Electric field norm, first wave (V/m)

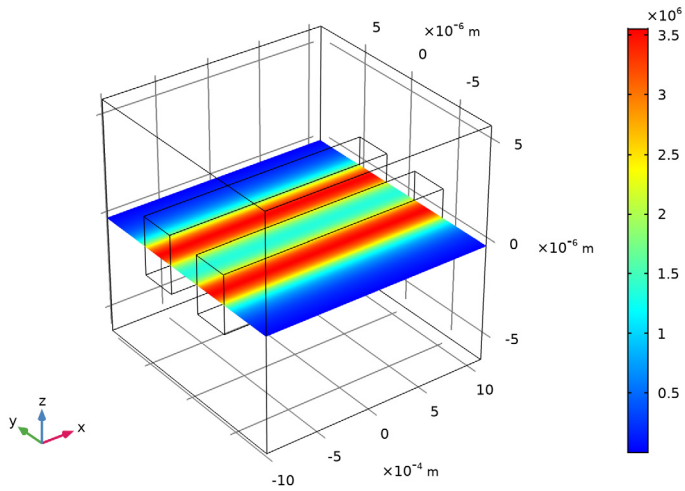


Figure 18: The amplitude of the first wave, when the bidirectional formulation is used.

freq(1)=260.69 THz

Slice: Electric field norm, second wave (V/m)

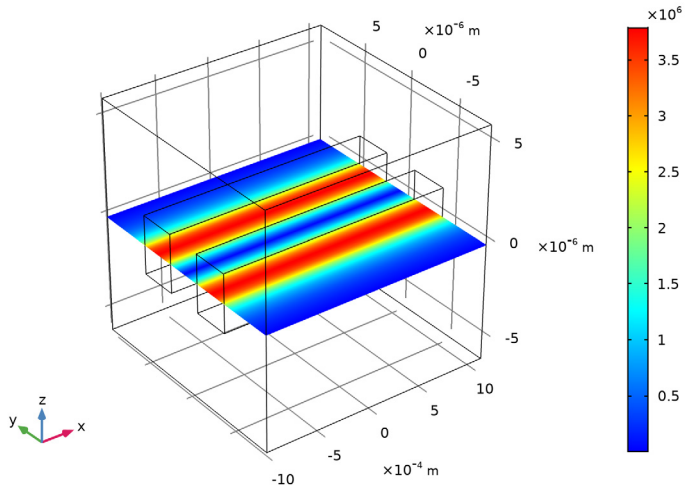


Figure 19: The amplitude of the second wave, when the bidirectional formulation is used.

Reference



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

Model Wizard






These step-by-step instructions guide you through the design and modeling of the directional coupler in 3D. First the simple geometry and the materials are defined. Then the lowest order modes are determined. Those modes (the symmetric and the antisymmetric modes combined) are then used to excite the waveguide structure. In the next step, it is shown how you can change the sign of one of the modes to excite the other waveguide. Finally, the last instructions use the bidirectional formulation for the Electromagnetic Waves, Beam Envelopes to model the two beating modes. Since in this case each mode will have a constant amplitude when propagating along the waveguide, a much coarser mesh can be used.

Note: These instructions are for the user interface on Windows but apply, with minor differences, also to Linux and Mac.

- 1 To start the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the Model Wizard to create a new COMSOL model or Blank Model to create one manually. For this tutorial, click the Model Wizard button.

If COMSOL is already open, you can start the Model Wizard by selecting New  from the File menu and then click Model Wizard .

The Model Wizard guides you through the first steps of setting up a model. The next window lets you select the dimension of the modeling space.

- 2 In the Select Space Dimension window click 3D .
- 3 In the Select Physics tree under Optics>Wave Optics, click Electromagnetic Waves, Beam Envelopes (ewbe) .
- 4 Click Add and then click Study .
- 5 In the Studies tree under Preset Studies for Selected Physics Interfaces, click Boundary Mode Analysis .
- 6 Click Done .

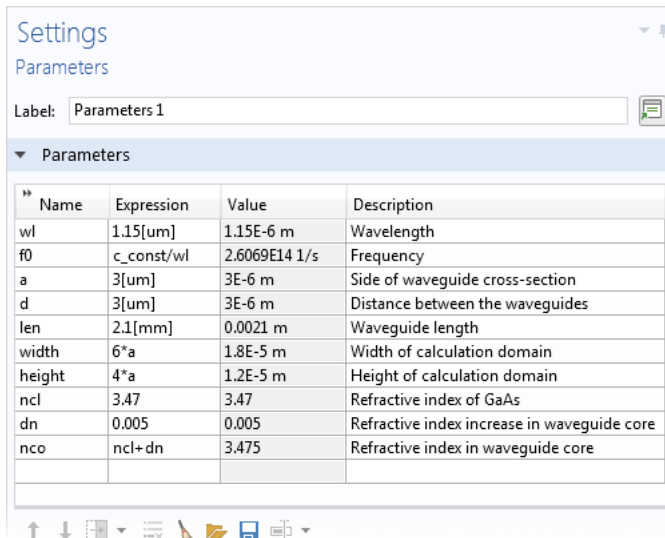
Global Definitions - Parameters

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

1 On the Home toolbar click Parameters P_i and select Parameters 1 P_i .

Note: On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.

2 In the Settings window for Parameters enter these settings in the Parameters table




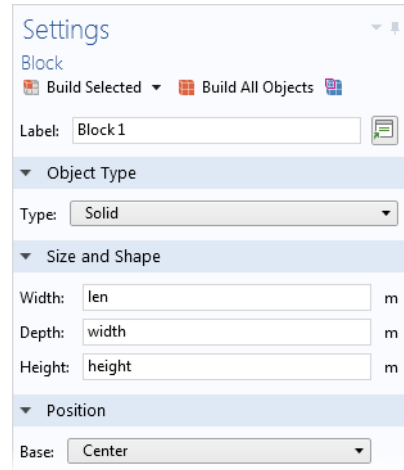
Geometry 1

In section [Global Definitions - Parameters](#) the parameters for the geometry were defined. Using a parametrized geometry it is easy to experiment with different dimensions for your waveguide structure.

To build the geometry, start by defining the simulation domain and then add the two embedded waveguides.

Block 1

- 1 On the Geometry toolbar click Block .
- 2 Go to the Settings window for Block. Under the Size and Shape section in the:
 - Width text field enter len .
 - Depth text field enter $width$.
 - Height text field enter $height$
- 3 Under Position choose Center from the Base list.



Settings

Block

Build Selected Build All Objects

Label: Block 1

Object Type

Type: Solid

Size and Shape

Width: len m

Depth: width m


Height: height m

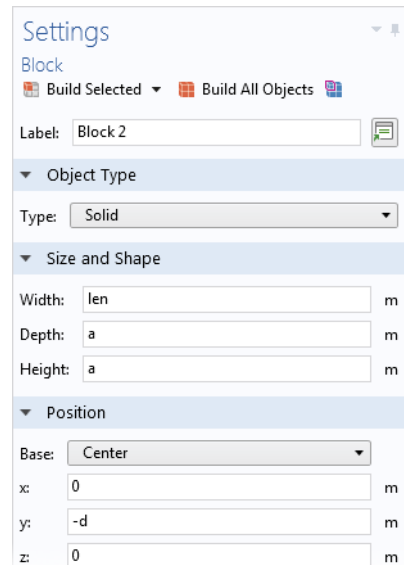
Position

Base: Center

Block 2

Now add the first embedded waveguide.

- 1 On the Geometry toolbar click Block .
- 2 Go to the Settings window for Block. Under the Size and Shape section in the:
 - Width text field enter len .
 - Depth text field enter a .
 - Height text field enter a .
- 3 Under Position choose Center from the Base list.
- 4 In the y text field enter $-d$.



Settings

Block

Build Selected Build All Objects

Label: Block 2

Object Type

Type: Solid

Size and Shape

Width: len m

Depth: a m

Height: a m

Position

Base: Center



x: 0 m

y: -d m

z: 0 m

Block 3

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

- 1 Right-click Block 2  and choose Duplicate .

- 2 Go to the Settings window for Block. In the Position section in the y text field enter d.

Settings

Block

Build Selected Build All Objects

Label: Block 3

Object Type

Type: Solid

Size and Shape

Width: len m

Depth: a m

Height: a m


Position

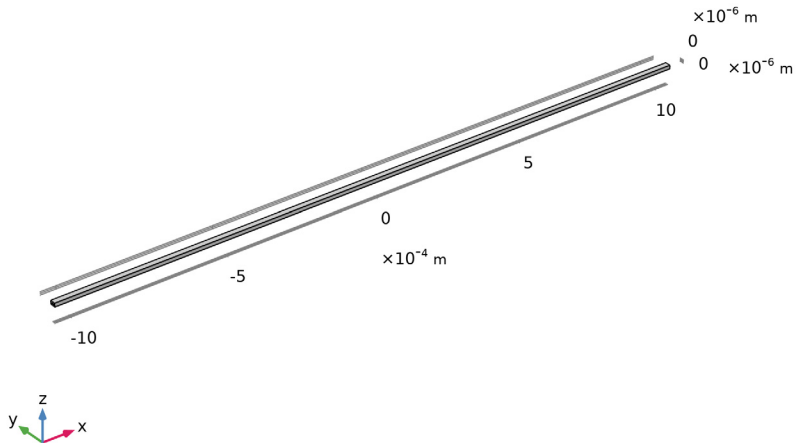
Base: Center

x: 0 m

y: d m





z: 0 m

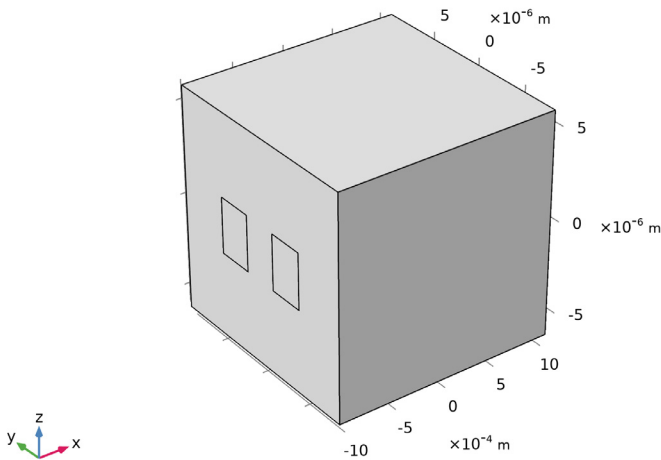
- 3 Click the Build All Objects button  .



Definitions

Since the geometry is so long and narrow, it is better to change the aspect ratio in the view.

- 1 In the Model Builder expand the Component 1>Definitions>View 1 node , then click Camera .
- 2 Go to the Settings window for Camera. In the Camera section, choose Automatic from the View scale list, Anisotropic from the Automatic list, and select the Automatic update check box.
- 3 Click the Update  button at the top of the Camera settings.
- 4 Click the Zoom Extents button  to produce the figure below.




Materials

Now, add materials for the cladding and the core of the waveguides. First add the waveguide cladding material. By default, the first material will be assigned to all domains. In the second step, you will define the waveguide core material.

For this waveguide structure, GaAs is used as cladding material and ion-implanted GaAs is used as core material.

Material 1

- 1 From the Materials toolbar click Blank Material  .
- 2 Go to the Settings window for Material. Type GaAs Cladding in the Label text field.




- 3 Under the Material Contents section enter the following settings in the table:

| Material Contents | | | | | |
|-------------------------------------|----------------------------------|-------------|-------|------|------------------|
| | Property | Variable | Value | Unit | Property group |
| <input checked="" type="checkbox"/> | Refractive index, real part | n_iso ;... | ncl | 1 | Refractive index |
| <input checked="" type="checkbox"/> | Refractive index, imaginary part | ki_iso ;... | 0 | 1 | Refractive index |

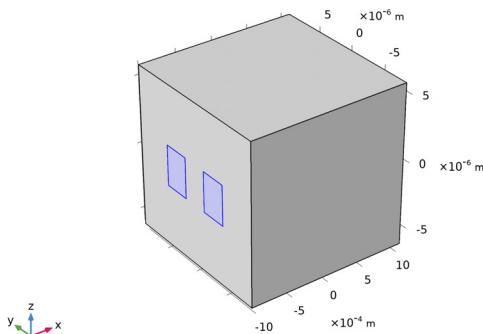
The Stop sign for Refractive index, imaginary part will disappear when you move away from this settings panel.

Material 2

- 1 From the Materials toolbar click Blank Material  .
- 2 Go to the Settings window for Material. Type Implanted GaAs Core in the Label text field.



- 3 Select only the waveguide cores, domains 2 and 3.



4 Under the Material Contents section enter the following settings in the table:

| Material Contents | | | | | |
|--|------------|-------|------|------------------|--|
| Property | Variable | Value | Unit | Property group | |
| <input checked="" type="checkbox"/> Refractive index, real part | n_iso ;... | nco | 1 | Refractive index | |
| <input checked="" type="checkbox"/> Refractive index, imaginary part | ki_iso... | 0 | 1 | Refractive index | |

Electromagnetic Waves, Beam Envelopes

For many simulation problems you have some boundaries between different materials, where reflected waves propagating in the backward direction are created. Thus, the default setting for the Electromagnetic Waves, Beam Envelopes interface is to do bidirectional simulations.

However, for the directional coupler problem, we don't expect any reflected waves. So it is best to select unidirectional propagation in this case.

As will be demonstrated in the final part of this tutorial, the bidirectional formulation can also be used for solving for two modes propagating in the same direction. In this case, this will further reduce the requirement on the mesh size.

- 1 On the Home toolbar select Electromagnetic Waves, Beam Envelopes.
- 2 In the Settings window under the Wave Vectors section, choose Unidirectional from the Number of directions list.
- 3 Enter `ewbe.beta_1` in the x text field of the \mathbf{k}_1 table, as shown in the figure.

| Wave Vectors | | |
|------------------------------|--------------------------|---|
| Number of directions: | | |
| Unidirectional | | |
| Type of phase specification: | | |
| Wave vector | | |
| Wave vector, first wave: | | |
| | <code>ewbe.beta_1</code> | x |
| \mathbf{k}_1 | 0 | y |
| | 0 | z |
| rad/m | | |

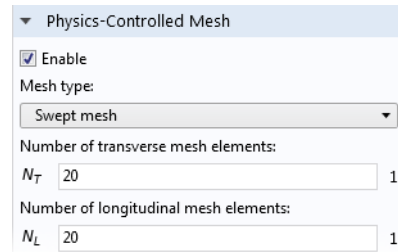
This sets the wave vector to be that of the lowest waveguide mode. With this wave vector setting, the phase factor of the electric field perfectly matches the

lowest mode, but it is slightly mismatched to the also excited higher-order mode, as discussed in the [Model Definition](#) section.

The expression `ewbe.beta_1` is colored orange, as a warning that this variable has not yet been defined. We will define it, when adding the first port below.

By default, the Electromagnetic Waves, Beam Envelopes interface will use a swept mesh consisting of ten mesh elements per transverse dimension and ten mesh elements in the longitudinal direction (the propagation direction). To sufficiently resolve the modes and the mode coupling, 20 mesh elements will be used in this model for both the transverse and longitudinal dimensions.



- 1 Go to the Physics-Controlled Mesh section and enter 20 in the Number of transverse mesh elements field
- 2 Enter 20 in the Number of longitudinal mesh elements field.



Port 1

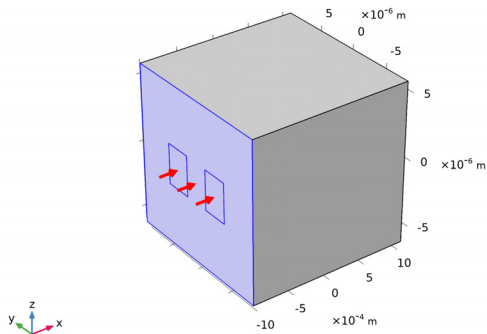
To excite the wave at the input boundary and to absorb the wave at the output boundary, two numeric ports per boundary are used. The first two ports excite the waveguides.

Each numeric port has a Boundary Mode Analysis study step associated with it. The Boundary Mode Analysis study performs an Eigenvalue study to find the mode fields and the propagation constant associated to the port. The Boundary Mode Analysis steps will be defined first for Port 1 in section [Study 1](#) and later for the remaining ports in section [Study 1](#) (same name but links to a different section).


- 1 On the Physics toolbar click Boundaries  and choose Port .
- 2 Select Boundaries 1, 5, and 10, as shown in the figure to the right.
- 3 Go to the Settings window for Port. In the Port Properties section choose Numeric from the Type of port list.

Since this is the first added Port, the Wave excitation at this port list is automatically set to On.

Now duplicate the first port.




Port 2

1 Right-click Port 1 and choose Duplicate .

Next, create the absorbing ports at the other end of the waveguide structure.

Port 3

1 On the Physics toolbar click Boundaries  and choose Port .

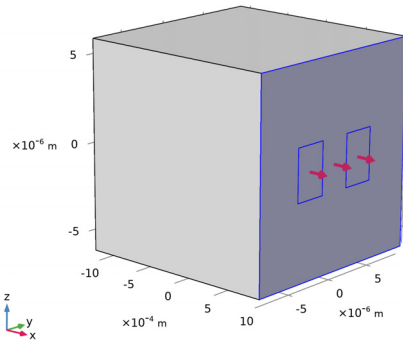
2 Select Boundaries 16–18 only, as shown in the figure to the right.

3 Go to the Settings window for Port. In the Port Properties section choose Numeric from the Type of port list.

Duplicate this port.


Port 4

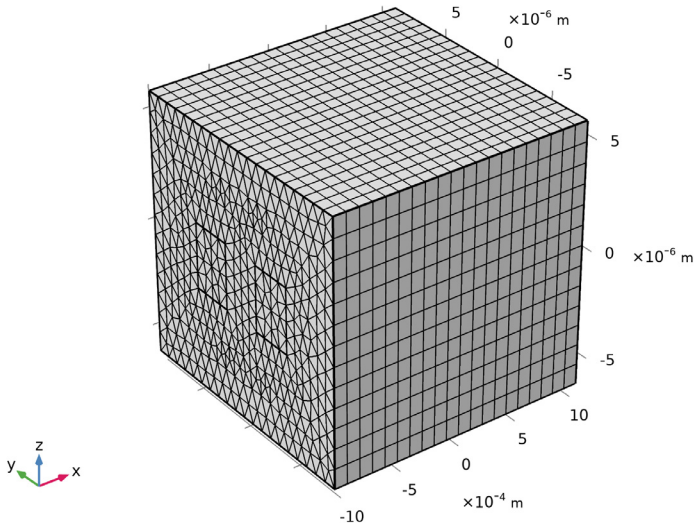
1 Right-click Port 3 and choose Duplicate .



Mesh


Since the mesh was defined in the Physics-controlled mesh section on the physics interface, there are no more actions required to define the mesh. Just inspect it.

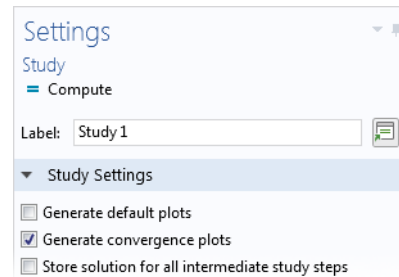
1 In the settings for the Mesh node, click the Build All button .



Study 1


For this model, special plots are generated. Start by clearing the check box for the generation of the default plots.



- 1 In the Model Builder click Study 1 .
- 2 In the Settings window for Study locate the Study Settings section and clear the Generate default plots check box.

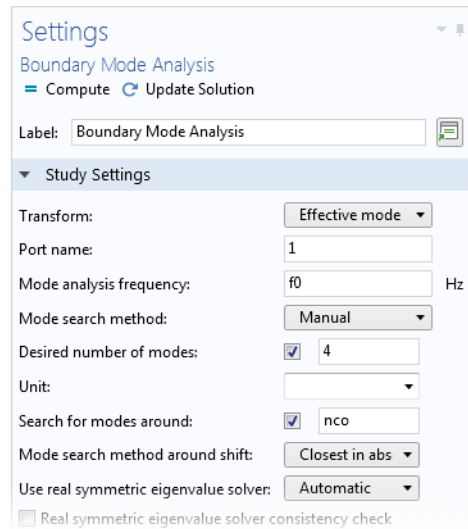


Step 1: Boundary Mode Analysis

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

- 1 In the Model Builder under Study 1, click Step 1: Boundary Mode Analysis .




- 2 In the Settings window for Boundary Mode Analysis locate the Study Settings section, mark the check box for the Desired number of modes text field and enter 4 in the associated text field. This finds the four lowest modes.
- 3 In the Search for modes around setting, mark the check box and enter nc0 in the associated text field, to search for the modes with effective index close to that of the waveguide cores.
- 4 In the Mode analysis frequency text field enter $f0$.
- 5 To compute only the boundary mode analysis step, right-click Step 1: Boundary Mode Analysis  and choose Compute Selected Step .






Results

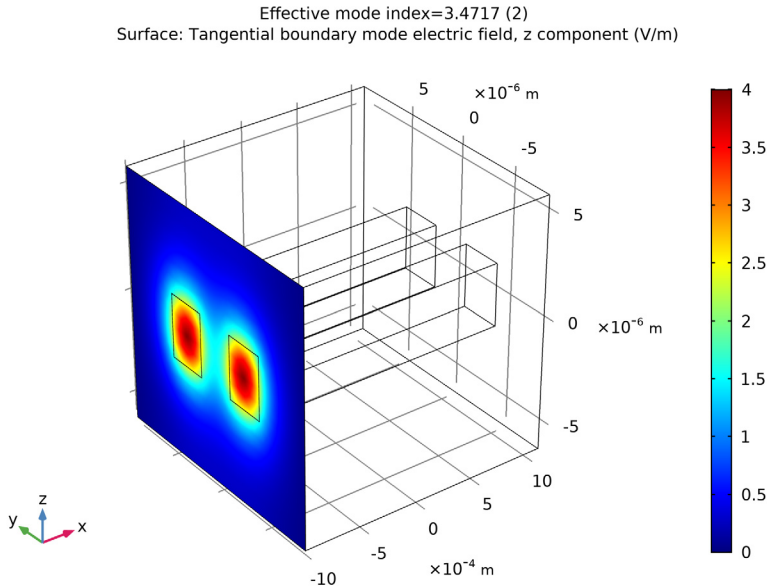
Create a 3D surface plot to view the different modes.

3D Plot Group 1

- 1 On the Home toolbar, click Add Plot Group  and select 3D Plot Group .
- 2 On the 3D Plot Group 1 toolbar, select Surface .

Now, first look at the modes polarized in the z -direction.
- 3 In the Settings window for Surface click Replace Expression  in the upper-right corner of the Expression section. From the menu, navigate to and select the variable Model>Component 1>Electromagnetic Waves, Beam Envelopes>Boundary mode analysis>Tangential boundary mode electric field>Tangential boundary mode electric field, z component (ewbe.tEbmIz).
- 4 In the Model Builder click 3D Plot Group 1 .
- 5 In the Settings window for 3D Plot Group 1 locate the Data section. From the Effective mode index list, choose the largest effective index. This is the last value in the list. Since the modes are pairwise (almost) degenerate, the value to pick is 3.4717.


- 6 Click the Plot button . The plot below shows the symmetric mode polarized in the z-direction (same as Figure 12 in Results and Discussion).

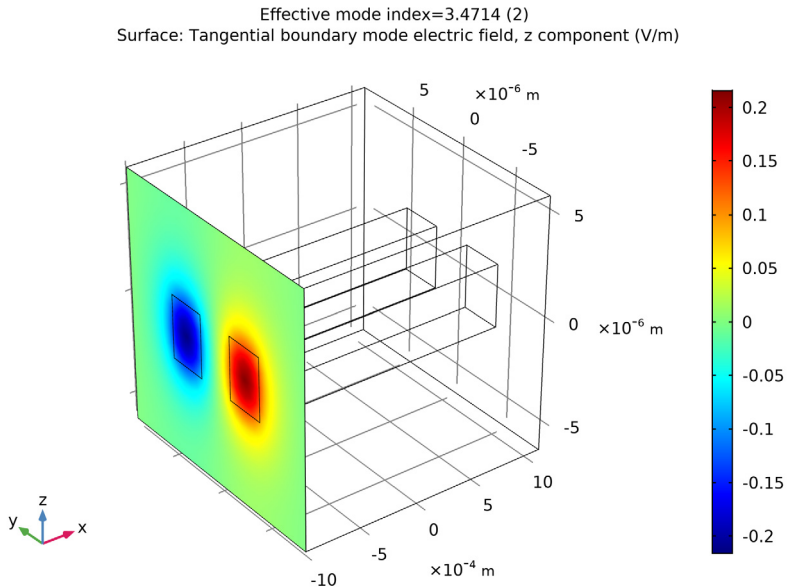




Notice that your plot may look different from the plot above, as the plot shows the real part of the boundary mode electric field. The computed complex electric field can have a different phase factor than for the plot above. Thus, the field can either show minima (a blue color) or maxima (a red color) at the locations for the waveguide cores. However, since this is a symmetric mode, it will have the same field values for both waveguide cores.


Another consequence of the arbitrary phase factor is that the magnitude for the displayed real part of the electric field in your plot can be different from what is shown in the plot above.

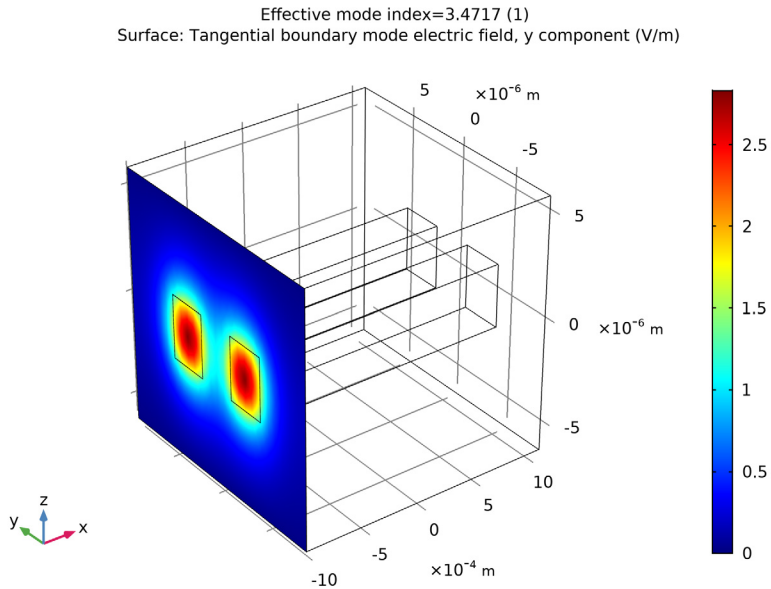
- 7 From the Effective mode index list, choose the third largest effective index. This is the second value in the list (3.4714).

- 8 Click the Plot  button. The plot below shows the anti-symmetric mode polarized in the z-direction (same as Figure 13 in Results and Discussion).




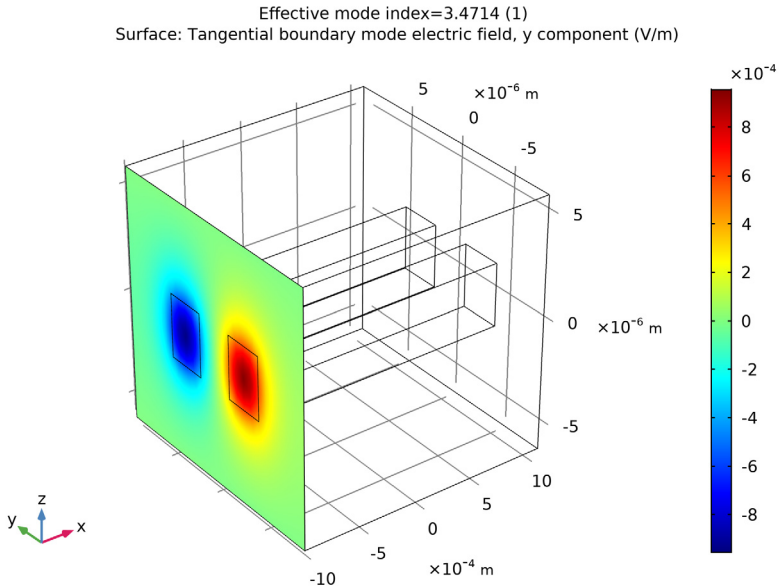
- 9 In the Model Builder under 3D Plot Group 1, click Surface 1 .
- 10 From the Settings window for Surface locate the Expression section and enter $\text{ewbe.tEb}m1y$ in the Expression text field, to plot the mode field polarized in the y-direction.
- 11 In the Model Builder click 3D Plot Group 1 .
- 12 In the Settings window for 3D Plot Group locate the Data section. From the Effective mode index list, choose the second largest effective index. This is the third value in the list (3.4717).

13 Click the Plot button . The plot below shows the symmetric mode polarized in the y-direction (same as [Figure 14](#) in [Results and Discussion](#)).





14 From the Effective mode index list, choose the smallest effective index (the first value in the list, 3.4714)

- 15 Click the Plot button . The plot below shows the anti-symmetric mode polarized in the y -direction (same as Figure 15 in Results and Discussion).



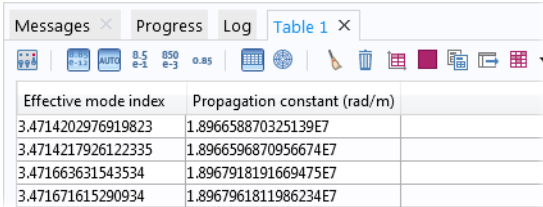
Derived Values

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


- 1 On the Results toolbar select Global Evaluation .
- 2 Locate the Expression section, in the Settings window for Global Evaluation and enter `ewbe.beta_1` in the Expression text field. The variable `ewbe.beta_1` is the propagation constant related to the first port.
- 3 Click the Evaluate button .

Now, copy all information in the table to the clipboard. Then paste that information into your favorite text editor, so you easily can enter the values later in the boundary mode analysis steps.

4 In the Table window click Full Precision 8.85×10^{-12} .




| Effective mode index | Propagation constant (rad/m) |
|----------------------|------------------------------|
| 3.4714202976919823 | 1.896658870325139E7 |
| 3.4714217926122335 | 1.8966596870956674E7 |
| 3.471663631543534 | 1.8967918191669475E7 |
| 3.471671615290934 | 1.8967961811986234E7 |

5 In the Table window, click Copy Table and Headers to Clipboard .

6 Paste the contents of the clipboard into your text editor, for later reference.

Study 1

Step 1: Boundary Mode Analysis

1 In the Model Builder under Study 1, click Step 1: Boundary Mode Analysis .

2 Go to the Settings window for Boundary Mode Analysis locate the Study Settings section and enter 1 in the Desired number of modes text field.

3 In the Search for modes around text field enter the largest effective mode index, 3.471671615290934. You can select the value in your text editor and then copy and paste it here. The last figures could be different from what is written here.



Step 3: Boundary Mode Analysis 1

1 Right-click Step 1: Boundary Mode Analysis  and choose Duplicate .

2 In the Settings window for Boundary Mode Analysis locate the Study Settings section. In the Search for modes around text field enter (or copy and paste) 3.4714217926122335. This should be the third largest effective index. The last figures could be different from what is written here.

3 Enter 2 in the Port name text field.


Step 4: Boundary Mode Analysis 2

1 Select the two boundary mode analyses, Step 1: Boundary Mode Analysis  and Step 3: Boundary Mode Analysis 3 .




2 In the Model Builder right-click Step 1: Boundary Mode Analysis  and choose Duplicate .

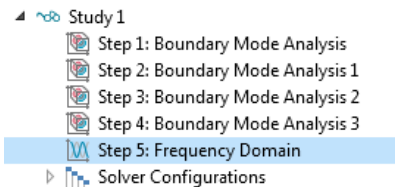
3 For the node Step 4: Boundary Mode Analysis 2, go to the Settings window for Boundary Mode Analysis locate the Study Settings section and enter 3 in the Port name text field.

Step 5: Boundary Mode Analysis 3

- 1 In the Model Builder under Study 1, click Step 5: Boundary Mode Analysis 3 .
- 2 Go to the Settings window for Boundary Mode Analysis locate the Study Settings section and enter 4 in the Port name text field.

Step 2: Frequency Domain

- 1 In the Model Builder under Study 1, click Step 2: Frequency Domain .
- 2 Go to the Settings window for Frequency Domain locate the Study Settings section and enter f0 in the Frequencies text field.
- 3 Finally, move Step2: Frequency Domain to be the last study step, by right-clicking Step 2: Frequency Domain  and choose Move Down  (three times) or by dragging Step2: Frequency Domain to the last study step. After the move, the study sequence should look like this sequence:










- 4 Right-click Study 1  and choose Compute .

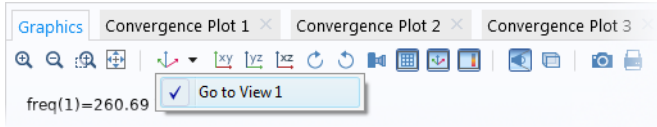
Results


3D Plot Group 1

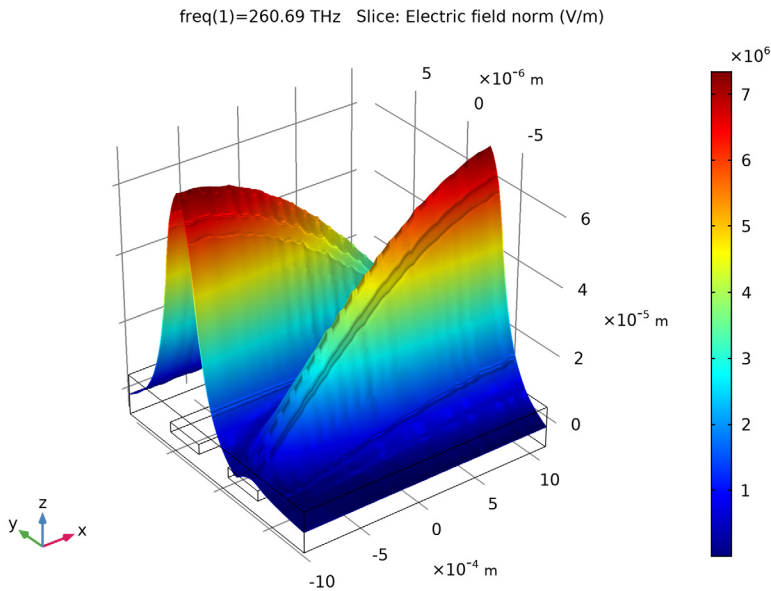
Remove the surface plot and replace it with a slice plot of the norm of the electric field.

- 1 In the Model Builder under 3D Plot Group 1, right-click Surface 1  and choose Delete . Click Yes in the Confirm Delete dialog.
- 2 On the 3D Plot Group 1 toolbar select Slice .
- 3 From the Settings window for Slice locate the Plane Data section and from the Plane list, choose XY-planes.
- 4 In the Planes text field enter 1.
- 5 Click to expand the Quality section. From the Smoothing list select Everywhere. This will make the plot smooth across the core-cladding interfaces.
- 6 Right-click Slice 1  and choose Deformation .

- 7 In the Settings window for Deformation locate the Expression section and in the z component text field enter `ewbe.normE`.
- 8 Click the Plot button .
- 9 Click the little down arrow next to the View button  on the Graphics toolbar and select the Go to View 1 button from the drop-down menu.




- 10 Click the Zoom Extents button  on the Graphics toolbar. The plot below shows how the light couples from the excited waveguide to the unexcited one (same as [Figure 16](#) in [Results and Discussion](#)).

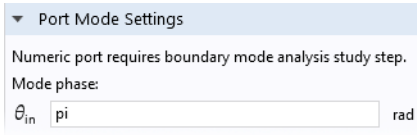


Electromagnetic Waves, Beam Envelopes


Port 2

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

- 1 In the Model Builder under Electromagnetic Waves, Beam Envelopes, click Port 2 .
- 2 In the Settings window for Port locate the Port Mode Settings section and enter π in the Mode phase (θ_{in}) text field.



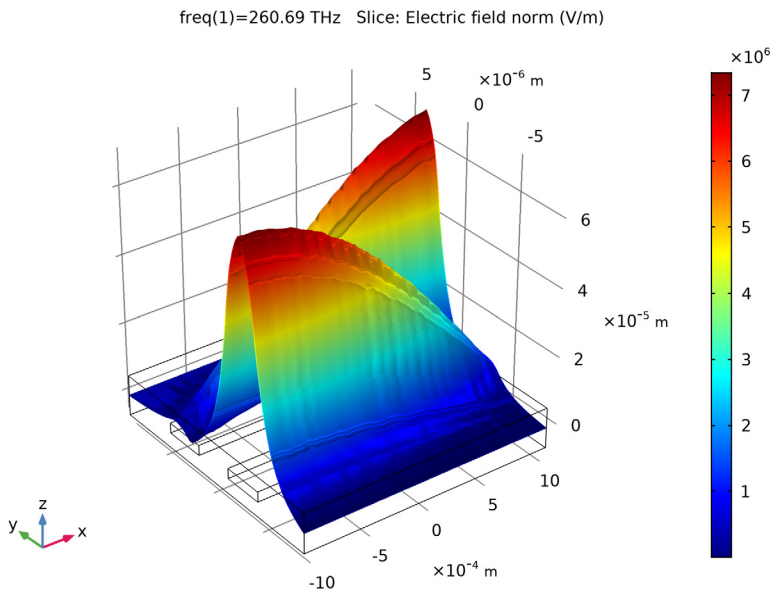
Study 1

On the Home toolbar, select Compute .

Results

3D Plot Group 1

Now the other waveguide is excited and the coupling occurs in reverse direction, compared to the previous case. The figure below reproduces [Figure 17](#) in [Results and Discussion](#).

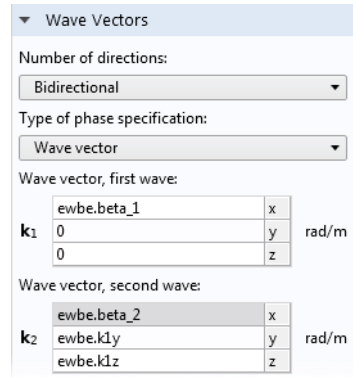
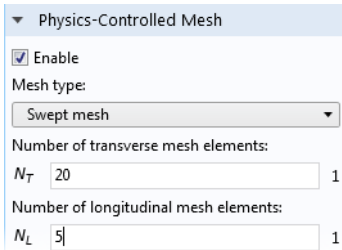


Electromagnetic Waves, Beam Envelopes (ewbe)

Finally, modify the Electromagnetic Waves, Beam Envelopes physics interface to make a bidirectional simulation, where the two wave vectors are codirectional but use the propagation constants of the two beating modes. In this case, each wave will have an almost constant amplitude, so a very coarse mesh can be used in the propagation direction.


- 1 In the Model Builder window, under Component 1 (comp1) click Electromagnetic Waves, Beam Envelopes (ewbe).

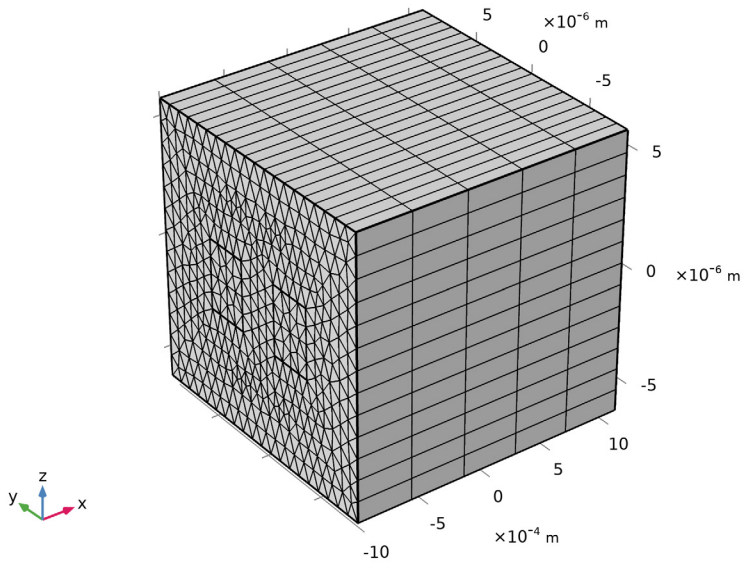
- 2 In the Settings window for Electromagnetic Waves, Beam Envelopes, locate the Wave Vectors section.
- 3 From the Number of directions list, choose Bidirectional.
- 4 Enter `ewbe.beta_2` in the x text field of the \mathbf{k}_2 table, as shown in the figure.
- 5 Locate the Physics-Controlled Mesh section. In the N_L text field, type 5, as shown below.




Mesh 1

- 1 Now just inspect the mesh, by In the Model Builder window, under Component 1 (comp1), click Mesh1.

2 In the settings for the Mesh node, click the Build All button  .

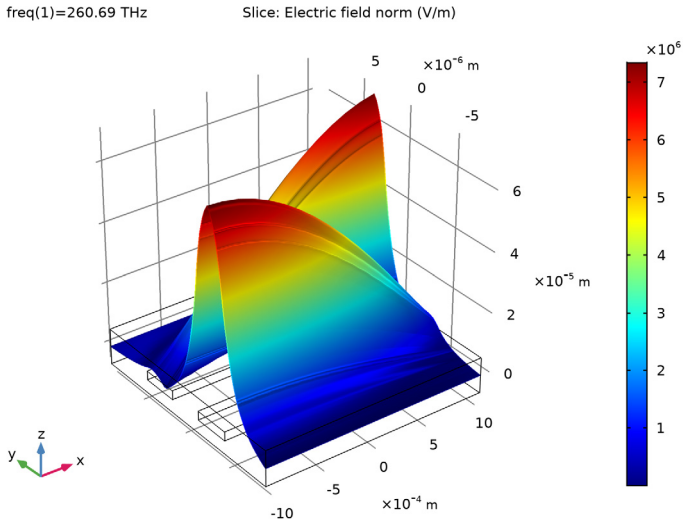


Study I

On the Home toolbar, select Compute  .

Results





3D PLOT GROUP 1




The resulting plot should look similar to [Figure 17](#).

SLICE 1

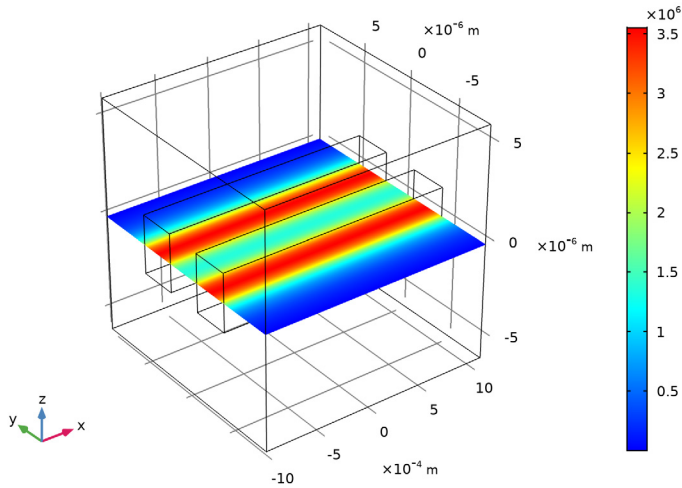
Finally, add a plot to verify that the field amplitudes indeed are constant.

- 1 On the Home toolbar, click Add Plot Group  and select 3D Plot Group .
- 2 In the Model Builder window, right-click 3D Plot Group 2 and choose Slice .
- 3 In the Settings window for Slice, locate the Expression section.
- 4 In the Expression text field, type `ewbe.normE1`.
- 5 Locate the Plane Data section. From the Plane list, choose XY-planes.
- 6 In the Planes text field, type `1`.
- 7 Locate the Coloring and Style section. From the Color table list, choose `RainbowLight`.
- 8 In the 3D Plot Group 2 toolbar, click the Plot button .


9 Click the Zoom Extents button  to produce the figure below.

freq(1)=260.69 THz

Slice: Electric field norm, first wave (V/m)

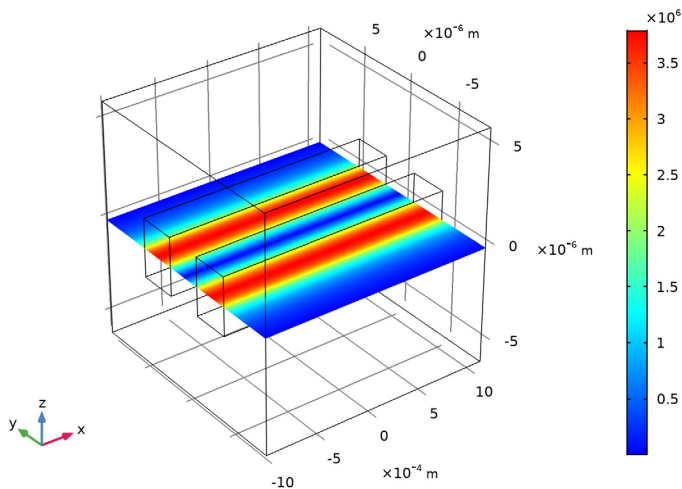


10 Locate the Expression section. In the Expression text field, type `ewbe.normE2`.

11 In the 3D Plot Group 2 toolbar, click the Plot button .

freq(1)=260.69 THz

Slice: Electric field norm, second wave (V/m)



This concludes this introduction to the Wave Optics Module. For further reading including theory sections, see the Wave Optics Module User's Guide.