Model created in COMSOL Multiphysics 6.4

Dielectric Slab Waveguide

Introduction

A planar dielectric slab waveguide demonstrates the principles behind any kind of dielectric waveguide, such as a ridge waveguide or a step index fiber, and has a known analytic solution. This model solves for the effective index of a dielectric slab waveguide as well as for the fields, and compares to analytic results.

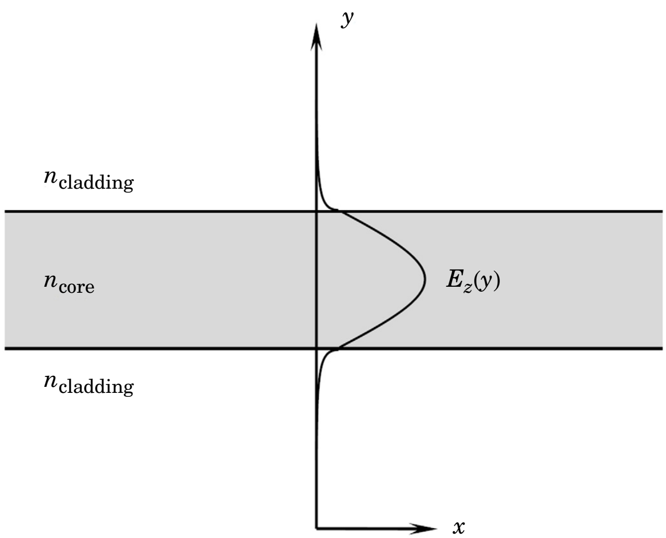


Figure 1: The guided modes in a dielectric slab waveguide have a known analytic solution.

Model Definition

A dielectric slab of thickness h_{slab} = 1 µm and refractive index n_{core} = 1.5 forms the core of the waveguide, and sits in free space with n_{cladding} = 1. Light polarized out of the plane of propagation, of wavelength λ = 1550 nm, is perfectly guided along the axis of the waveguide structure, as shown in Figure 1. Here, only the TE_0 mode can propagate. The structure varies only in the y direction, and it is infinite and invariant in the other two directions.

The analytic solution is found by assuming that the electric field along the direction of propagation varies as $E_z = E(y)\exp(-ik_x x)$, where $E(y) = C_1\cos(k_y y)$ inside the dielectric slab, and $E(y) = C_0\exp(-\alpha(|y| - (h_{\mathrm{slab}}/2))$ in the cladding. Because the electric and magnetic fields must be continuous at the interface, the guidance condition is

$$\alpha = k_y \tan\left(k_y \frac{t_{\text{slab}}}{2}\right)$$

where k_y and α satisfy

$$k_y^2 = k_{\text{core}}^2 - k_{\text{cladding}}^2 - \alpha^2$$

with $k_{\text{core}} = 2\pi n_{\text{core}}/\lambda$ and $k_{\text{cladding}} = 2\pi n_{\text{cladding}}/\lambda$. It is possible to find the solution to the above two equations via the Newton–Raphson method, which is used whenever COMSOL Multiphysics detects a system of nonlinear equations, the only requirement being that of an adequate initial guess.

This model considers a section of a dielectric slab waveguide that is finite in the x and y directions. Because the fields drop off exponentially outside the waveguide, the fields can be assumed to be zero at some distance away. This is convenient as it makes the boundary conditions in the y direction irrelevant, assuming that they are imposed sufficiently far away.

Use the Numeric Port boundary conditions in the x direction to model the guided wave propagating in the positive x direction. These boundary conditions require first solving an eigenvalue problem that solves for the fields and propagation constants at the boundaries.

Results and Discussion

Figure 2 shows the results. The numeric port boundary condition at the left side excites a mode that propagates in the x direction and is perfectly absorbed by the numeric port on the right side. The analytic and numerically computed propagation constants agree.

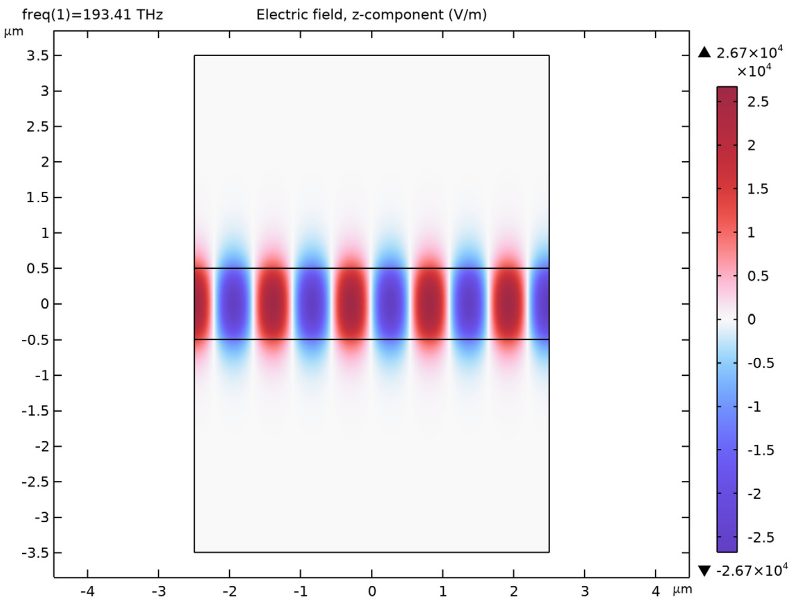


Figure 2: The electric field in a dielectric slab waveguide.

Application Library path: `Wave_Optics_Module/Verification_Examples/dielectric_slab_waveguide`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Optics** > **Wave Optics** > **Electromagnetic Waves, Frequency Domain (ewfd)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Empty Study**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`lda0`	`1550[nm]`	1.55E-6 m	`Wavelength`
`n_core`	`1.5`	1.5	`Refractive index, core`
`n_cladding`	`1`	1	`Refractive index, cladding`
`h_core`	`1[um]`	1E-6 m	`Thickness, core`
`h_cladding`	`7[um]`	7E-6 m	`Thickness, cladding`
`w_slab`	`5[um]`	5E-6 m	`Slab width`
`k_core`	`2*pi[rad]*n_core/lda0`	6.0805E6 rad/m	`Wave number, core`
`k_cladding`	`2*pi[rad]*n_cladding/lda0`	4.0537E6 rad/m	`Wave number, cladding`
`f0`	`c_const/lda0`	1.9341E14 1/s	`Frequency`

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **µm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `w_slab`.

4 In the **Height** text field, type `h_core`.

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

6 Click **Build Selected**.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `w_slab`.

4 In the **Height** text field, type `h_cladding`.

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

6 Click **Build All Objects**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

Only solve for the out-of-plane electric field component, since we are only interested in solving for a transverse electric (TE) mode.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (ewfd)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Components** section.

3 From the **Electric field components solved for** list, choose **Out-of-plane vector**.

The wave is excited at the port on the left side.

Port 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Port**.

2 Select Boundaries 1, 3, and 5 only.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Numeric**.

For the first port, wave excitation is **on** by default.

Now, add the exit port.

Port 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Port**.

2 Select Boundaries 8–10 only (the boundaries on the right side).

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Numeric**.

MATERIALS

Cladding

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Cladding` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	`n_cladding`	1	Refractive index

By default, the first material you add applies on all domains. Add a core material.

Core

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Core` in the **Label** text field.

3 Select Domain 2 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	`n_core`	1	Refractive index

STUDY 1

Step 1: Boundary Mode Analysis

1 In the **Study** toolbar, click **More Study Steps** and choose **Other** > **Boundary Mode Analysis**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Search for modes around shift** text field, type `n_core`. This value should be in the vicinity of the value that you expect the fundamental mode to have.

4 In the **Mode analysis frequency** text field, type `f0`.

Step 2: Boundary Mode Analysis 2

Add another boundary mode analysis, for the second port.

1 In the **Study** toolbar, click **More Study Steps** and choose **Other** > **Boundary Mode Analysis**.

2 In the **Settings** window for **Boundary Mode Analysis**, locate the **Study Settings** section.

3 In the **Search for modes around shift** text field, type `n_core`.

4 In the **Port name** text field, type `2`.

5 In the **Mode analysis frequency** text field, type `f0`.

Step 3: Frequency Domain

Finally, add the study step for the propagating wave in the waveguide.

1 In the **Study** toolbar, click **Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `f0`.

4 In the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (ewfd)

The default plot shows the norm of the electric field. Modify the plot to shows the *z* component (compare with Figure 2).

Surface 1

1 In the **Model Builder** window, expand the **Electric Field (ewfd)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Electric** > **Electric field - V/m** > **ewfd.Ez - Electric field, z-component**.

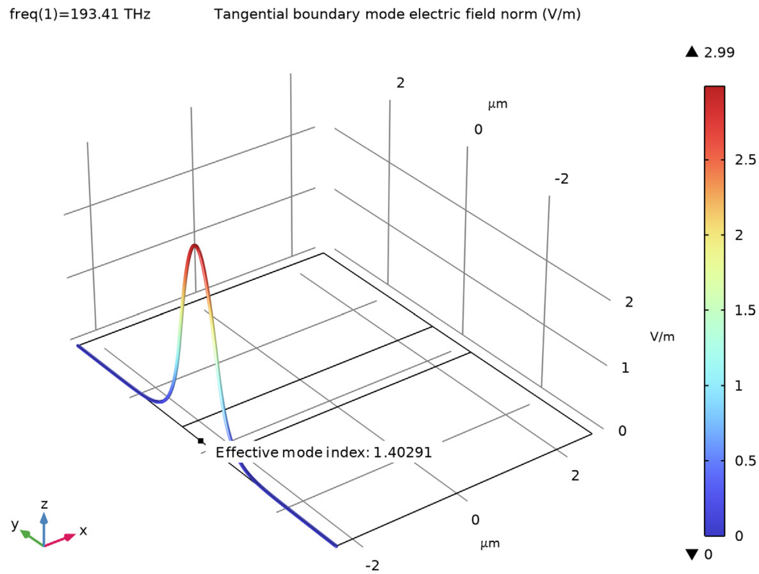
3 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

4 In the **Electric Field (ewfd)** toolbar, click **Plot**.

Now inspect the mode field plot and the effective mode index resulting from the boundary mode analysis performed for each port.

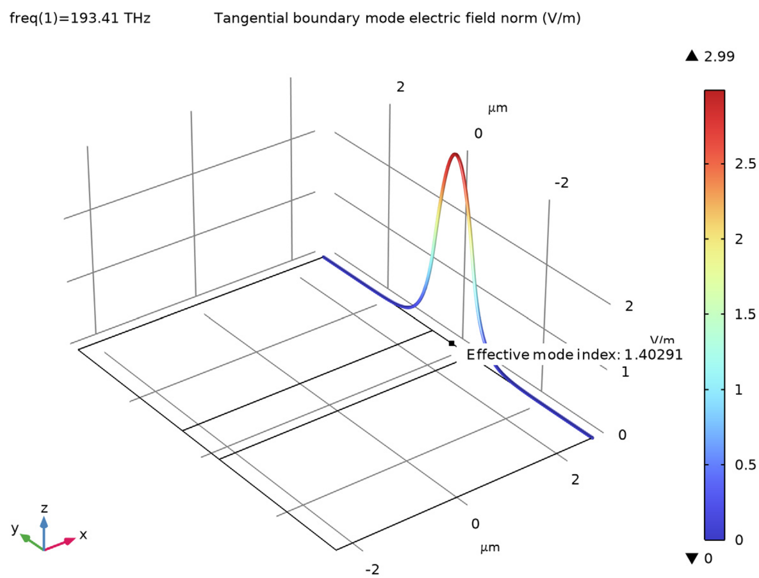
Electric Mode Field, Port 1 (ewfd)

In the **Model Builder** window, under **Results** click **Electric Mode Field, Port 1 (ewfd)**.



Electric Mode Field, Port 2 (ewfd)

In the **Model Builder** window, click **Electric Mode Field, Port 2 (ewfd)**.



Finish by comparing the simulation results to the analytic solution. To compute the latter, add a Global ODEs and DAEs interface and then set up and solve the relevant equations.

ADD PHYSICS

1. In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2. Go to the **Add Physics** window.
3. In the tree, select **Mathematics** > **ODE and DAE Interfaces** > **Global ODEs and DAEs (ge)**.
4. Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Study 1**.
5. Click the **Add to Component 1** button in the window toolbar.
6. In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Electromagnetic Waves, Frequency Domain (ewfd)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

5 Click the **Add Study** button in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

GLOBAL ODES AND DAES (GE)

Global Equations 1 (ODE1)

1 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

2 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
alpha	alpha-k_y*tan(k_y*h_core/2)	k_core/2	0	
k_y	k_y^2-(k_core^2-k_cladding^2-alpha^2)	k_core/2	0	

STUDY 2

In the **Study** toolbar, click **Compute**.

RESULTS

Global Evaluation 1

Finally, compare analytical and computed propagation constants.

Global Evaluation 2

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Ports** > **Propagation constants** > **All expressions in this group**.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
ewfd.beta_1	rad/m	Propagation constant, port 1
ewfd.beta_2	rad/m	Propagation constant, port 2

4 Click **Evaluate**.

Global Evaluation 3

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 4 (sol4)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
sqrt(k_core^2-k_y^2)		Propagation constant, computed

5 Click **Evaluate**.