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.

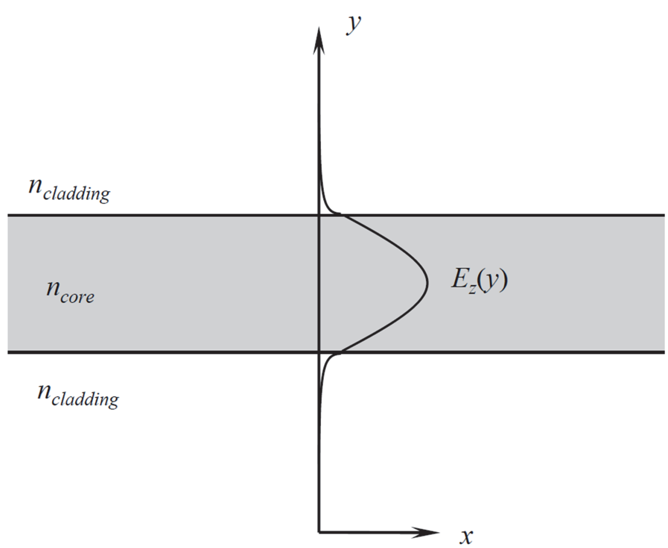


Figure 1: The guided modes in a dielectric slab waveguide have a known analytic solution.

Model Definition

A dielectric slab of thickness $h_{\text{slab}} = 1$ µm and refractive index $n_{\text{core}} = 1.5$ forms the core of the waveguide, and sits in free space with $n_{\text{cladding}} = 1$. Light polarized out of the plane of propagation, of wavelength $\lambda = 1550$ 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 $E(y) = C_0\exp(-\alpha(|y| - (h_{slab}/2))$ in the cladding. Because the electric and magnetic fields must be continuous at the interface, the guidance condition is

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

where k_y and α satisfy

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

with $k_{core} = 2\pi n_{core}/\lambda$ and $k_{cladding} = 2\pi n_{cladding}/\lambda$. 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 Numerical 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 numerical port boundary condition at the left side excites a mode that propagates in the x direction and is perfectly absorbed by the numerical port on the right side. The analytic and numerically computed propagation constants agree.

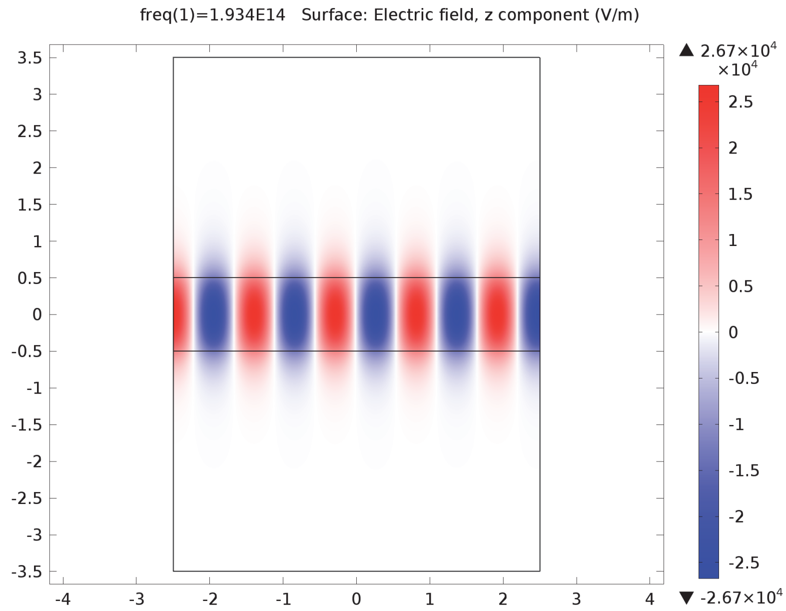


Figure 2: The electric field in a dielectric slab waveguide.

Model Library path: `Wave_Optics_Module/Verification_Models/dielectric_slab_waveguide`

Modeling Instructions

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

NEW

1 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 **Custom Studies>Empty Study**.

6 Click **Done**.

DEFINITIONS

Parameters

1 On the **Model** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
lambda0	1550[nm]	1.550E-6 m	Wavelength
n_core	1.5	1.500	Refractive index, core
n_cladding	1	1.000	Refractive index, cladding
h_core	1[um]	1.000E-6 m	Thickness, core
h_cladding	7[um]	7.000E-6 m	Thickness, cladding
w_slab	5[um]	5.000E-6 m	Slab width
k_core	2*pi[rad]*n_core/lambda0	6.081E6 rad/m	Wave number, core
k_cladding	2*pi[rad]*n_cladding/lambda0	4.054E6 rad/m	Wave number, cladding
f0	c_const/lambda0	1.934E14 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 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size** 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 Right-click **Component 1 (comp1)>Geometry 1>Rectangle 1 (r1)** and choose **Build Selected**.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size** 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 the **Build All Objects** button.

7 Click the **Zoom Extents** button on 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 On 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**.

5 From the **Wave excitation at this port** list, choose **On**.

Now, add the exit port.

Port 2

1 On 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

Material 1 (mat1)

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

2 In the **Settings** window for Material, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Refractive index	n	`n_cladding`	1	Refractive index

4 Right-click **Component 1 (comp1)>Materials>Material 1 (mat1)** and choose **Rename**.

5 In the **Rename Material** dialog box, type `Cladding` in the **New label** text field.

6 Click **OK**.

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

Material 2 (mat2)

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 Select Domain 2 only.

3 In the **Settings** window for Material, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Refractive index	n	`n_core`	1	Refractive index

5 Right-click **Component 1 (comp1)>Materials>Material 2 (mat2)** and choose **Rename**.

6 In the **Rename Material** dialog box, type `Core` in the **New label** text field.

7 Click **OK**.

MESH 1

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Settings** window for Size, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lambda0/n_cladding/8`.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for Size, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

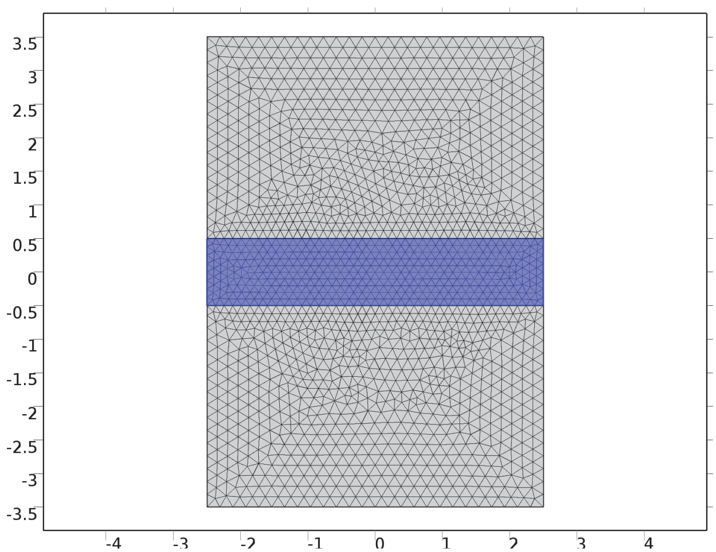
4 Select Domain 2 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `lambda0/n_core/8`.

8 Click the **Build All** button.



STUDY 1

Step 1: Boundary Mode Analysis

1 On the **Study** toolbar, click **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** 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`.

Add another boundary mode analysis, for the second port.

Step 2: Boundary Mode Analysis 2

1 On the **Study** toolbar, click **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** text field, type `n_core`.
4 In the **Port name** text field, type `2`.
5 In the **Mode analysis frequency** text field, type `f0`.

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

Step 3: Frequency Domain

1 On the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain**.
2 In the **Settings** window for Frequency Domain, locate the **Study Settings** section.
3 In the **Frequencies** text field, type `f0`.
4 On 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).

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>Electromagnetic Waves, Frequency Domain>Electric>Electric field>ewfd.Ez - Electric field, z component**.
3 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.
4 On the **2D plot group** toolbar, click **Plot**.

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 On the **Model** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the **Add physics** tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.

4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
Study 1	×

5 Click **Add to Component** in the window toolbar.

6 On the **Model** toolbar, click **Add Physics** to close the **Add Physics** window.

ADD STUDY

1 On the **Model** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select study** tree, select **Custom Studies>Preset Studies for Some Physics Interfaces>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Electromagnetic Waves, Frequency Domain (ewfd)	×

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

6 On the **Model** toolbar, click **Add Study** to close the **Add Study** window.

GLOBAL ODES AND DAES (GE)

On the **Physics** toolbar, click **Electromagnetic Waves, Frequency Domain (ewfd)** and choose **Global ODEs and DAEs (ge)**.

Global Equations 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1**.

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (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

On the **Model** toolbar, click **Compute**.

RESULTS

Derived Values

Finally, compare analytical and computed propagation constants.

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

2 In the **Settings** window for Global Evaluation, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>ewfd.beta_1 - Propagation constant**.

3 Locate the **Expression** section. Select the **Description** check box.

4 In the associated text field, type `Propagation constant,beta_1`.

5 Click the **Evaluate** button.

6 On the **Results** toolbar, click **Global Evaluation**.

7 In the **Settings** window for Global Evaluation, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Ports>ewfd.beta_2 - Propagation constant**.

8 Locate the **Expression** section. Select the **Description** check box.

9 In the associated text field, type `Propagation constant, beta_2`.

10 Right-click **Results>Derived Values>Global Evaluation 4** and choose **Evaluate>Table 1 - Global Evaluation 3 (ewfd.beta_1)**.
11 On the **Results** toolbar, click **Global Evaluation**.
12 In the **Settings** window for Global Evaluation, locate the **Data** section.
13 From the **Data set** list, choose **Study 2/Solution 2**.
14 Locate the **Expression** section. In the **Expression** text field, type `sqrt(k_core^2-k_y^2)`.
15 Select the **Description** check box.
16 In the associated text field, type `Propagation constant, computed`.
17 Click the **Evaluate** button.