Step-Index Fiber

Introduction

The transmission speed of optical waveguides is superior to microwave waveguides because optical devices have a much higher operating frequency than microwaves, enabling a far higher bandwidth.

Today the silica glass (SiO_2) fiber is forming the backbone of modern communication systems. Before 1970, optical fibers suffered from large transmission losses, making optical communication technology merely an academic issue. In 1970, researchers showed, for the first time, that low-loss optical fibers really could be manufactured. Earlier losses of 2000 dB/km now went down to 20 dB/km. Today's fibers have losses near the theoretical limit of 0.16 dB/km at 1.55 μm (infrared light).

One of the winning devices has been the single-mode fiber, having a step-index profile with a higher refractive index in the centre core and a lower index in the outer cladding. Numerical software plays an important role in the design of single-mode waveguides and fibers. For a fiber cross section, even the most simple shape is difficult and cumbersome to deal with analytically. A circular step-index waveguide is a basic shape where benchmark results are available (see Ref. 1).

This example is a model of a single step-index waveguide made of silica glass. The inner core is made of pure silica glass with refractive index $n_1 = 1.4457$ and the cladding is doped, with a refractive index of $n_2 = 1.4378$. These values are valid for free-space wavelengths of 1.55 μm. The radius of the cladding is chosen to be large enough so that the field of confined modes is zero at the exterior boundaries.

For a confined mode there is no energy flow in the radial direction, thus the wave must be evanescent in the radial direction in the cladding. This is true only if

$$n_{\text{eff}} > n_2$$

On the other hand, the wave cannot be radially evanescent in the core region. Thus

$$n_2 < n_{\text{eff}} < n_1$$

The waves are more confined when n_{eff} is close to the upper limit in this interval.

©2012 COMSOL

Model Definition

The mode analysis is made on a cross-section in the *xy*-plane of the fiber. The wave propagates in the *z* direction and has the form

$$\mathbf{H}(x, y, z, t) = \mathbf{H}(x, y)e^{j(\omega t - \beta z)}$$

where ω is the angular frequency and β the propagation constant. An eigenvalue equation for the magnetic field **H** is derived from Helmholtz equation

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

which is solved for the eigenvalue $\lambda = -j\beta$.

As boundary condition along the outside of the cladding the magnetic field is set to zero. Because the amplitude of the field decays rapidly as a function of the radius of the cladding this is a valid boundary condition.

Results and Discussion

When studying the characteristics of optical waveguides, the effective mode index of a confined mode,

$$n_{\text{eff}} = \frac{\beta}{k_0}$$

as a function of the frequency is an important characteristic. A common notion is the normalized frequency for a fiber. This is defined as

$$V = \frac{2\pi a}{\lambda_0}\sqrt{n_1^2 - n_2^2} = k_0 a\sqrt{n_1^2 - n_2^2}$$

 ©2012 COMSOL

where a is the radius of the core of the fiber. For this simulation, the effective mode index for the fundamental mode, 1.4444 corresponds to a normalized frequency of 4.895. The electric and magnetic fields for this mode is shown in Figure 1 below.

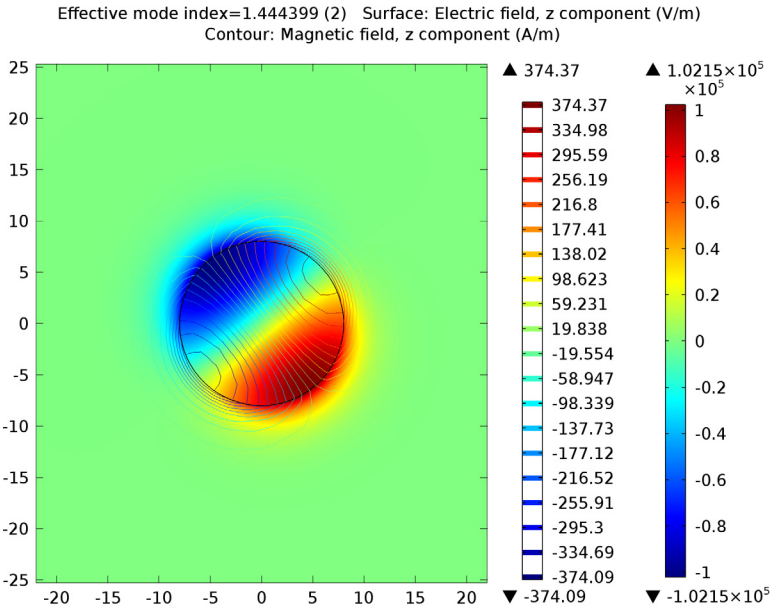


Figure 1: The surface plot visualizes the z component of the electric field. This plot is for the effective mode index 1.4444.

1. A. Yariv, *Optical Electronics in Modern Communications*, 5th edition, Oxford University Press, 1997.

Model Library path: `RF_Module/Optics_and_Photonics/step_index_fiber`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Mode Analysis**.

7 Click **Finish**.

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

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

Circle 1

1 Right-click **Model 1>Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `40`.

4 Click the **Build Selected** button.

Circle 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `8`.

4 Click the **Build Selected** button.

MATERIALS

Material 1

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

2 Right-click **Material 1** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Silica Glass` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 2 only.

6 In the **Material** settings window, locate the **Material Properties** section.

7 In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index (n)**.

8 Click **Add to Material**.

 ©2012 COMSOL

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

Property	Name	Value
Refractive index	n	1.4457

Material 2

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

2 Right-click **Material 2** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Doped Silica Glass` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 1 only.

6 In the **Material** settings window, locate the **Material Properties** section.

7 In the **Material properties** tree, select **Electromagnetic Models>Refractive Index>Refractive index (n)**.

8 Click **Add to Material**.

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

Property	Name	Value
Refractive index	n	1.4378

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Wave Equation, Electric 1

1 In the **Model Builder** window, expand the **Model 1>Electromagnetic Waves, Frequency Domain** node, then click **Wave Equation, Electric 1**.

2 In the **Wave Equation, Electric** settings window, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Refractive index**.

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Finer**.

4 Click the **Build All** button.

STUDY 1

Step 1: Mode Analysis

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

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

3 In the **Search for modes around** edit field, type `1.446`.

The modes of interest have an effective mode index somewhere between the refactive indices of the two materials. The fundamental mode has the highest index. Therefore, setting the mode index to search around to something just above the core index guarantees that the solver will find the fundamental mode.

4 In the **Mode analysis frequency** edit field, type `c_const/1.55[um]`.

This frequency corresponds to a free space wavelength of 1.55 μm.

5 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

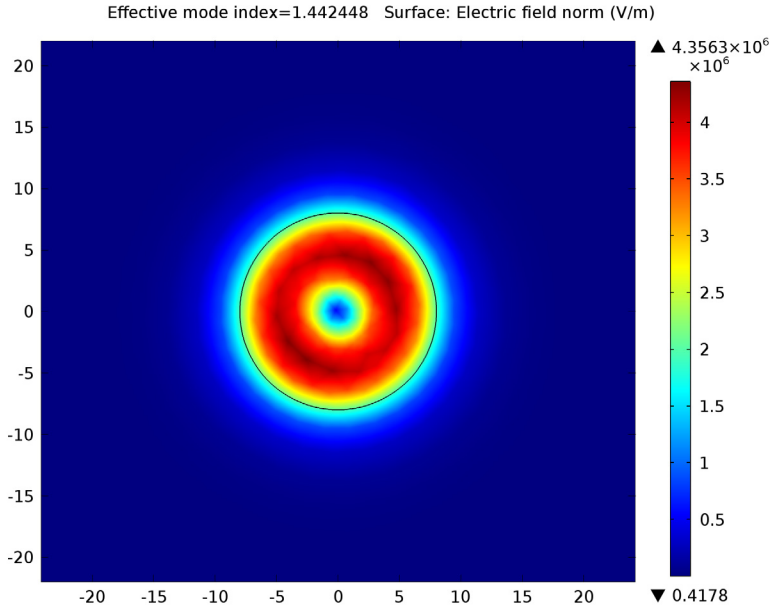
Electric field

1 Click the **Zoom Extents** button on the Graphics toolbar.

 ©2012 COMSOL

2 Click the **Zoom In** button on the Graphics toolbar.

The default plot shows the distribution of the norm of the electric field for the highest of the 6 computed modes (the one with the lowest effective mode index).



To study the fundamental mode, choose the lowest mode index. Because the magnetic field is exactly 90 degrees out of phase with the electric field you can see both the magnetic and the electric field distributions by plotting the solution at a phase angle of 45 degrees.

Data Sets

1 In the **Model Builder** window, expand the **Results>Data Sets** node, then click **Solution 1**.

2 In the **Solution** settings window, locate the **Solution** section.

3 In the **Solution at angle (phase)** edit field, type 45.

Electric field

1 In the **Model Builder** window, under **Results** click **Electric field**.

2 In the **2D Plot Group** settings window, locate the **Data** section.

3 From the **Effective mode index** list, choose **1.444399 (2)**.

4 In the **Model Builder** window, expand the **Electric field** node, then click **Surface**.

5 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field>Electric field, z component (emw.Ez)**.

6 Click the **Plot** button.

Add a contour plot of the H-field.

7 In the **Model Builder** window, right-click **Electric field** and choose **Contour**.

8 In the **Contour** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field>Magnetic field, z component (emw.Hz)**.

9 Click the **Plot** button.

The distribution of the transversal E and H field components confirms that this is the HE11 mode. Compare the resulting plot with that in Figure 1.

 ©2012 COMSOL