Created in COMSOL Multiphysics 6.0

Step-Index Fiber Bend

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 center 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 simplest 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, so the wave must be evanescent in the radial direction in the cladding. This is true only if

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

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

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

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

For a bent fiber, the mode is no longer completely guided by the refractive index structure. This can be qualitatively explained by considering that for a straight waveguide, the wavefronts (planes with a constant phase) are orthogonal to the fiber axis. For a circularly bent fiber, see Figure 1, the wavefronts rotate around the center point of the circle with a constant angular velocity. As a result, the wavelength and the propagation constant varies with the distance from the circle center point. At some distance from the center point, the wavelength is longer than the local material wavelength. Consequently the propagation constant is smaller than the local wave number defined by the vacuum wavelength and the refractive index of the cladding material. Beyond this radius, the wave cannot have a constant angular velocity and the wavefronts must bend, implying that the wave starts to radiate energy out from the fiber. For a more complete discussion about waves in bent waveguides, see Ref. 2.

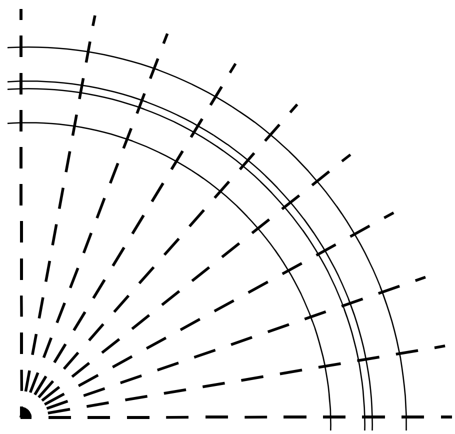


Figure 1: Schematic of a bent waveguide with dashed phase fronts indicated.

Model Definition

The first 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

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

where ω is the angular frequency and β the propagation constant. An eigenvalue equation for the electric field $\mathbf{E}$ is derived from Helmholtz equation

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

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

As boundary condition along the outside of the cladding, the electric 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.

The second mode analysis is performed for a 2D axisymmetric geometry. In this case, the wave propagates in the azimuthal direction, φ, and the electric field is expressed as

$$\mathbf{E}(r, \varphi, z, t) = \mathbf{E}(r, z)e^{j(\omega t - \beta r_0 \varphi)}$$

where r_0 is an average radius for the mode in the bent fiber. The radius r_0 is often slightly larger than the radius of curvature for the bent fiber. The eigenvalue solved for in this case is $\lambda = -j\beta r_0$. As a consequence of this eigenvalue definition, the effective indices you provide as input to the eigenvalue solver and the effective indices that the solver returns are all scaled with the radius r_0.

The geometry is defined as a rectangle surrounding the circular core domain. To absorb the radiating mode, there is a perfectly matched layer (PML) surrounding the rectangular cladding domain. The wavelength in the PML should correspond to the wave vector component normal to the PML–cladding boundary. Here we approximate this wave vector component with the radial wave vector component for the radiating wave in the cladding. The radial wave vector component can be obtained by first defining the azimuthal wave vector component as

$$\beta_\varphi = \beta\frac{r_0}{r_0 + \rho},$$

where the radial coordinate ρ is measured from the center of the waveguide. As seen from the equation above, the azimuthal wave vector component equals the mode's propagation constant within the waveguide core, where $\rho \approx 0$, and it decreases when ρ increases.

We approximate the radial wave vector component, assuming that the squared sum of the radial and the azimuthal wave vector components should be equal to the wave number squared for the cladding material. So, for the radial wave vector component we get

$$\beta_r = \sqrt{k_{cl}^2 - \beta_\varphi^2} = \frac{2\pi}{\lambda_0}\sqrt{n_{cl}^2 - \left(n_{\text{eff}}\frac{r_0}{r_0 + \rho}\right)^2}$$

and then the corresponding wavelength can be defined as

$$\lambda_r = \frac{2\pi}{\beta_r} = \frac{\lambda_0}{\sqrt{n_{cl}^2 - \left(n_{\text{eff}}\frac{r_0}{r_0 + \rho}\right)^2}}.$$

As an approximation, the effective index could be replaced by the refractive index of the core material. The coordinate ρ should here be the distance from the waveguide core to the PML boundary.

Results and Discussion

When studying the characteristics of straight 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

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

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 norm of the transverse electric field components and the electric field polarization are shown in Figure 2 below.

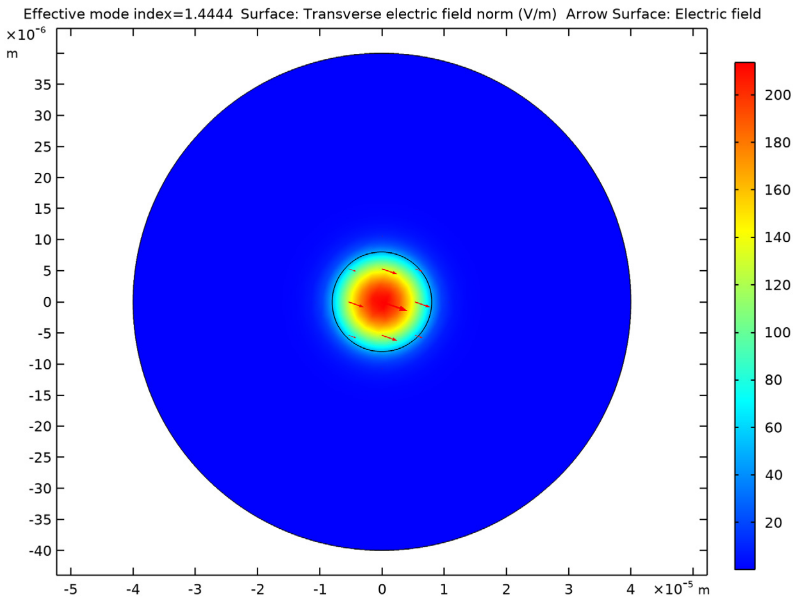


Figure 2: The norm of the transverse electric field (surface plot) and the electric field polarization (arrows) for the fundamental mode.

As a comparison, the longitudinal components of the electric and magnetic fields for this mode is shown in Figure 3 below. Comparing the color bars for Figure 2 and Figure 3, it is clear that the mode has a predominantly transverse polarization.

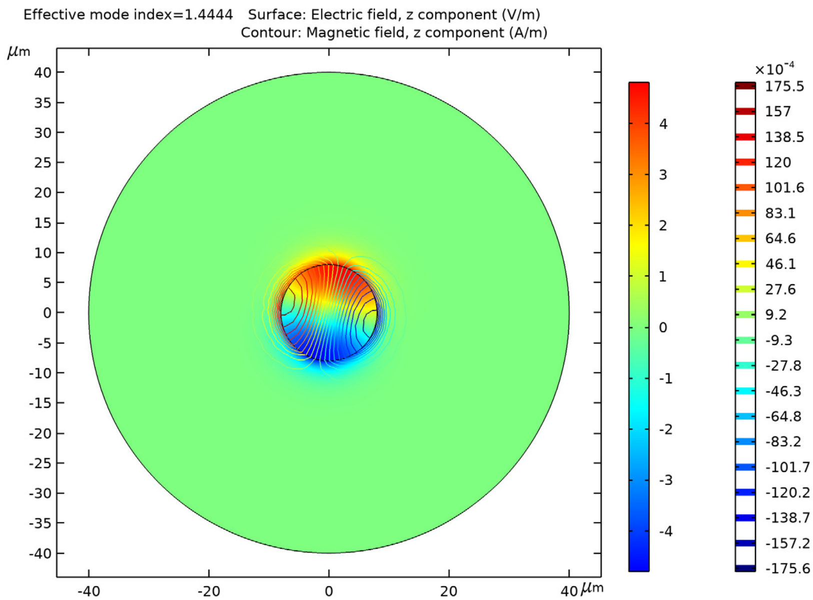


Figure 3: The surface plot visualizes the z component of the electric field. This plot is for the effective mode index 1.4444.

Figure 4 shows the result for the bend fiber, indicating that the mode is leaky and radiates some power in the radial direction.

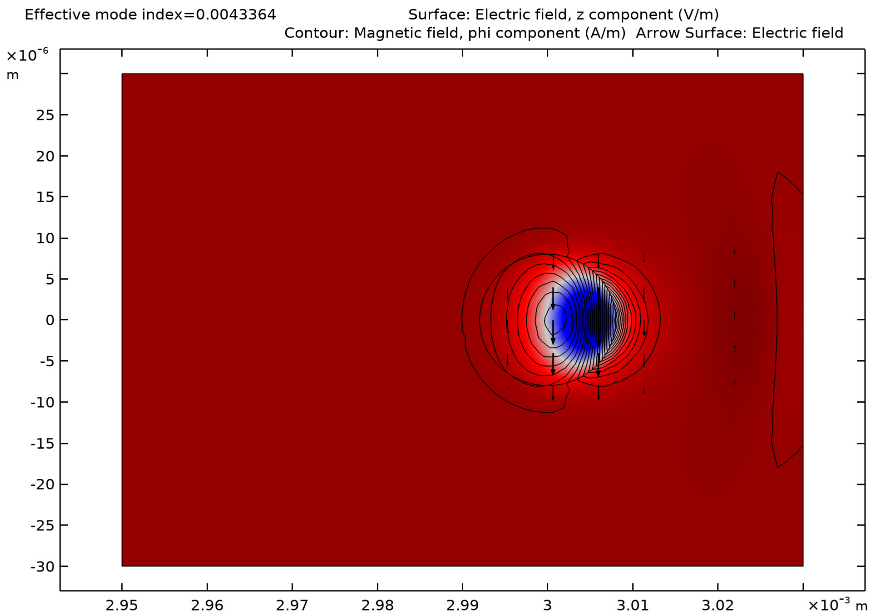


Figure 4: Surface plot of the z component of the electric field for the mode in the bent fiber. The contour plot shows the ϕ *component (in the direction of propagation) of the magnetic field and the arrow plot shows the electric field polarization.*

References

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

2. A.W. Snyder and J.D. Love, *Optical Waveguide Theory*, Chapman and Hall, 1983.

Application Library path: `Wave_Optics_Module/Waveguides_and_Couplers/step_index_fiber_bend`

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 **Preset Studies for Selected Physics Interfaces> Mode Analysis**.

6 Click **Done**.

GEOMETRY 1

First add some parameters that define the properties for the wave, the geometry, and the material.

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	1.55[um]	1.55E-6 m	Wavelength
f0	c_const/lda0	1.9341E14 1/s	Frequency
nClad	1.4378	1.4378	Refractive index, cladding
nCore	1.4457	1.4457	Refractive index, core
aCore	8[um]	8E-6 m	Core radius
aClad	40[um]	4E-5 m	Cladding radius
Rb	3[mm]	0.003 m	Bend radius
aSquare	100[um]	1E-4 m	Side length of square
tPML	20[um]	2E-5 m	PML thickness

Name	Expression	Value	Description
dr	aSquare/2-tPML	3E-5 m	Distance from core center to PML boundary
ldaPML	lda0/sqrt(nClad^2-(nCore*Rb/(Rb+dr))^2)	1.1426E-5 m	Radial wavelength in PML

Silica Glass

Next, add two global material definitions. Those two materials will be used in both of the two model components that will be defined. The material properties will be added once these materials have been linked to the first model component.

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

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

Doped Silica Glass

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

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

STRAIGHT FIBER

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

2 In the **Settings** window for **Component**, type `Straight Fiber` in the **Label** text field.

GEOMETRY 1

Circle 1 (c1)

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

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

3 In the **Radius** text field, type `aClad`.

4 Click **Build Selected**.

Circle 2 (c2)

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

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

3 In the **Radius** text field, type `aCore`.

4 Click **Build Selected**.

MATERIALS

Cladding

1 In the **Model Builder** window, under **Straight Fiber (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

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

Core

1 Right-click **Materials** and choose **More Materials>Material Link**.

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

3 Locate the **Link Settings** section. From the **Material** list, choose **Doped Silica Glass (mat2)**.

4 Select Domain 2 only.

GLOBAL DEFINITIONS

Silica Glass (mat1)

Now, add the material properties for the globally defined materials.

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Silica Glass (mat1)**.

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

3 In the table, enter the following settings:

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

Doped Silica Glass (mat2)

1 In the **Model Builder** window, click **Doped Silica Glass (mat2)**.

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

3 In the table, enter the following settings:

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

DEFINITIONS

Add a variable for the norm of the transverse electric field. This will be used later in a plot.

Variables 1

1 In the **Model Builder** window, under **Straight Fiber (comp1)** right-click **Definitions** and choose **Variables**.

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

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

4 From the **Selection** list, choose **All domains**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
`normEt`	`(ewfd.Ex*ewfd.Ex+ ewfd.Ey*ewfd.Ey)/ sqrt(ewfd.Ex*ewfd.Ex+ ewfd.Ey*ewfd.Ey)`	V/m	`Transverse electric field norm`

MESH 1

1 In the **Model Builder** window, under **Straight Fiber (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

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

STUDY 1

Step 1: Mode Analysis

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

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

3 Select the **Search for modes around** check box.

4 In the associated text field, type `nCore`.

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

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

6 In the **Model Builder** window, click **Study 1**.

7 In the **Settings** window for **Study**, type `Study 1 (Straight Fiber)` in the **Label** text field.

8 In the **Home** toolbar, click **=** **Compute**.

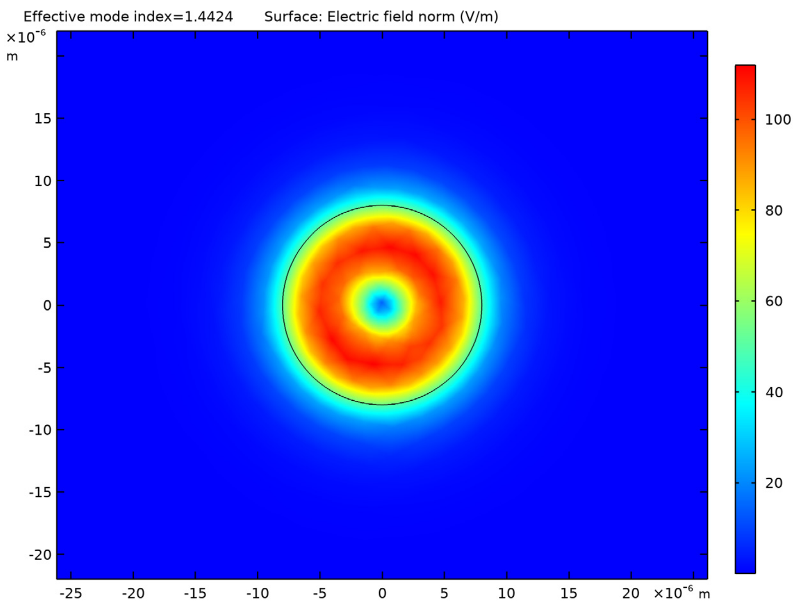
RESULTS

Electric Field (ewfd)

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

2 Click the **Zoom In** button in 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).



Study 1 (Straight Fiber)/Solution 1 (sol1)

To study the fundamental mode, choose the highest 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.

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1 (Straight Fiber)/Solution 1 (sol1)**.

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

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

Electric Field (ewfd)

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

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

3 From the **Effective mode index** list, choose **1.4444**.

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 **Straight Fiber (comp1)>Definitions> Variables>normEt - Transverse electric field norm - V/m**. This is the variable we added in a previous step. It is clear that the plot looks almost identical to the plot of the norm of the electric field, verifying that the electric field is predominantly polarized in the transverse direction (the *xy*-plane).

Electric Field (ewfd)

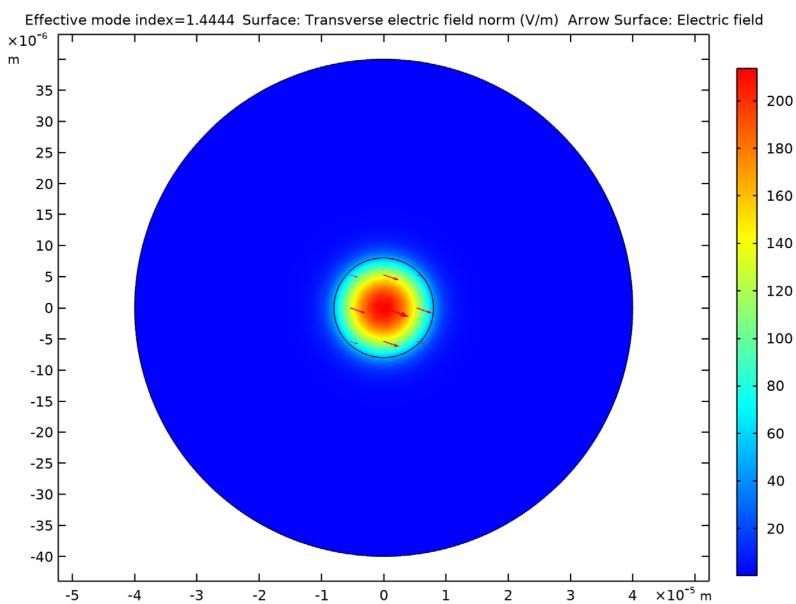
Indicate the polarization direction, using an arrow plot.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Electric Field (ewfd)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Straight Fiber (comp1)> Electromagnetic Waves, Frequency Domain>Electric>ewfd.Ex,ewfd.Ey - Electric field**.

3 In the **Electric Field (ewfd)** toolbar, click **Plot**. As the geometry has rotation symmetry, the polarization direction is arbitrary.



Surface 1

Now visualize the longitudinal components of the electric and magnetic field.

1 In the **Model Builder** window, 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 **Straight Fiber (comp1)> Electromagnetic Waves, Frequency Domain>Electric>Electric field - V/m>ewfd.Ez - Electric field, z component**.

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

Electric Field (ewfd)

Add a contour plot of the H-field.

Contour 1

1 In the **Model Builder** window, right-click **Electric Field (ewfd)** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Straight Fiber (comp1)> Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field - A/m>ewfd.Hz - Magnetic field, z component**.

3 In the **Electric Field (ewfd)** toolbar, click **Plot**. 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 3.

Solution 1 (Straight Fiber)

Rename the dataset and the plot group to refer to the Straight Fiber model component.

1 In the **Model Builder** window, under **Results>Datasets** click **Study 1 (Straight Fiber)/ Solution 1 (sol1)**.

2 In the **Settings** window for **Solution**, type `Solution 1 (Straight Fiber)` in the **Label** text field.

Straight Fiber

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

2 In the **Settings** window for **2D Plot Group**, type `Straight Fiber` in the **Label** text field.

ROOT

Now add a 2D axisymmetric model component to model the bent fiber.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component> 2D Axisymmetric**.

BENT FIBER

In the **Settings** window for **Component**, type `Bent Fiber` in the **Label** text field.

GEOMETRY 2

Add a circle representing the fiber core.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `aCore`.
4 Locate the **Position** section. In the **r** text field, type `Rb`.

Square 1 (sq1)

Add a square cladding region, representing the domain the mode essentially is propagating in.

1 In the **Geometry** toolbar, click **Square**.
2 In the **Settings** window for **Square**, locate the **Size** section.
3 In the **Side length** text field, type `aSquare`.
4 Locate the **Position** section. From the **Base** list, choose **Center**.
5 In the **r** text field, type `Rb`.

Rectangle 1 (r1)

Finally, add three rectangle domains, where you will define perfectly matched layers (PMLs).

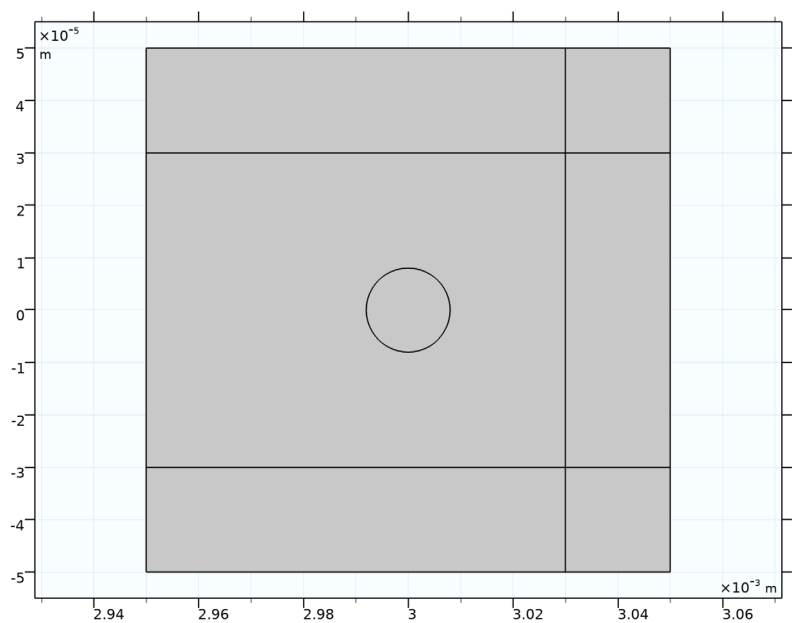
1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `aSquare`.
4 In the **Height** text field, type `tPML`.
5 Locate the **Position** section. In the **r** text field, type `Rb-aSquare/2`.
6 In the **z** text field, type `aSquare/2-tPML`.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

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 `tPML`.
4 In the **Height** text field, type `aSquare`.
5 Locate the **Position** section. In the **r** text field, type `Rb+dr`.
6 In the **z** text field, type `-aSquare/2`.

Rectangle 3 (r3)

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 `aSquare`.
4 In the **Height** text field, type `tPML`.
5 Locate the **Position** section. In the **r** text field, type `Rb-aSquare/2`.
6 In the **z** text field, type `-aSquare/2`.
7 Click **Build All Objects**.



BENT FIBER (COMP2)

Now, add the Electromagnetic Waves, Frequency Domain interface.

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 **Optics>Wave Optics>Electromagnetic Waves, Frequency Domain (ewfd)**.

4 Click **Add to Bent Fiber** in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EWFD2)

Add a Perfect Magnetic Conductor (PMC) exterior boundary condition.

Perfect Magnetic Conductor 1

1 Right-click **Bent Fiber (comp2)>Electromagnetic Waves, Frequency Domain 2 (ewfd2)** and choose **Perfect Magnetic Conductor**.

2 Select Boundaries 1–3, 5, 7, 9, and 14–17 only.

MATERIALS

Now link to the previously defined materials.

Cladding

1 In the **Model Builder** window, under **Bent Fiber (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

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

Core

1 Right-click **Materials** and choose **More Materials>Material Link**.

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

3 Select Domain 7 only.

4 Locate the **Link Settings** section. From the **Material** list, choose **Doped Silica Glass (mat2)**.

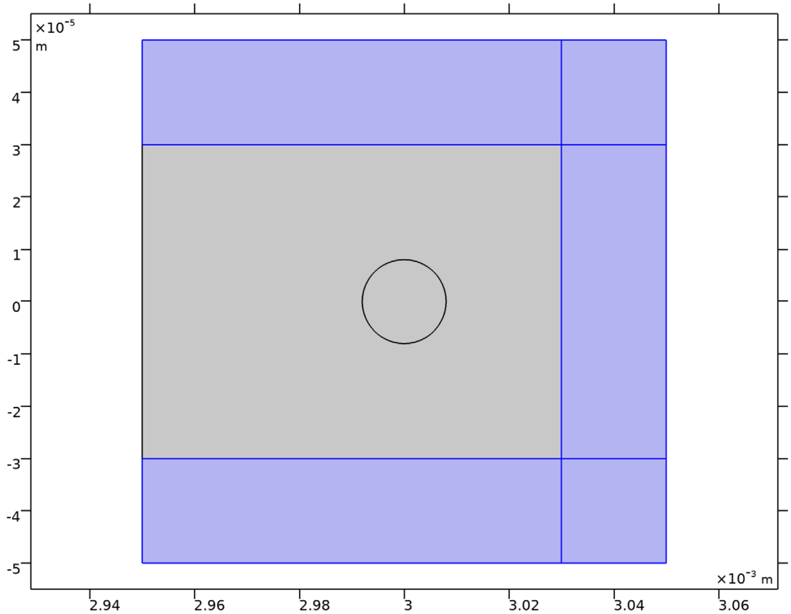
DEFINITIONS (COMP2)

Next add the PML domains.

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 3–6 only.



3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.

5 Locate the **Scaling** section. From the **Coordinate stretching type** list, choose **Rational**.

6 From the **Typical wavelength from** list, choose **User defined**.

7 In the **Typical wavelength** text field, type `ldaPML`. This wavelength setting approximates the transverse (in the radial direction) wavelength for the mode.

Integration 1 (intop1)

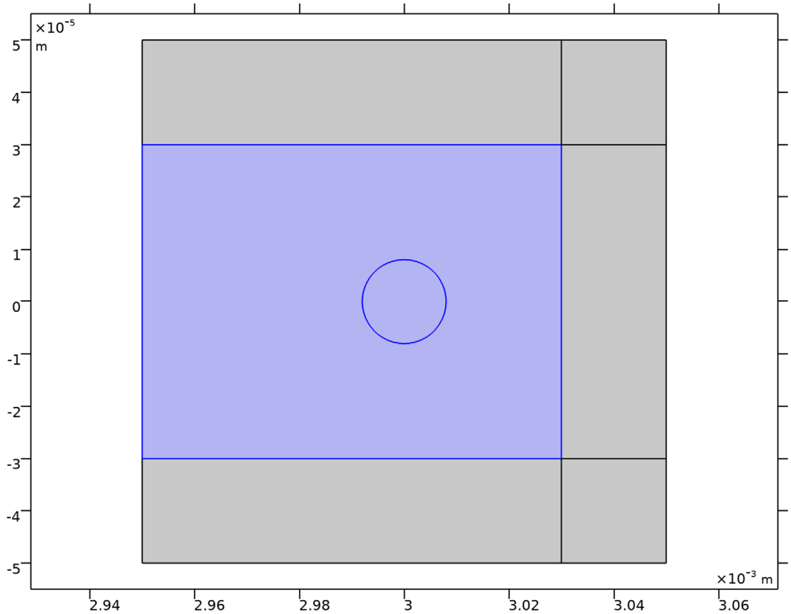
Define a variable for the mode's averaged radial position, by first defining the integration operator and then the actual variable.

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 Select Domains 2 and 7 only.

3 In the **Settings** window for **Integration**, locate the **Advanced** section.

4 Clear the **Compute integral in revolved geometry** check box.



Variables 2

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
r0	intop1(r*ewfd2.Poavphi)/ intop1(ewfd2.Poavphi)	m	

MESH 2

1 In the **Model Builder** window, under **Bent Fiber (comp2)** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 In the table, clear the **Use** check box for **Electromagnetic Waves, Frequency Domain 2 (ewfd2)**.

4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Size

Add the same mesh element size for the core and the cladding domains, as for the straight fiber model component.

1. In the **Model Builder** window, under **Bent Fiber (comp2)>Mesh 2** click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Finer**.

Free Triangular 1

1. In the **Model Builder** window, click **Free Triangular 1**.
2. In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 2 and 7 only.

Mapped 1

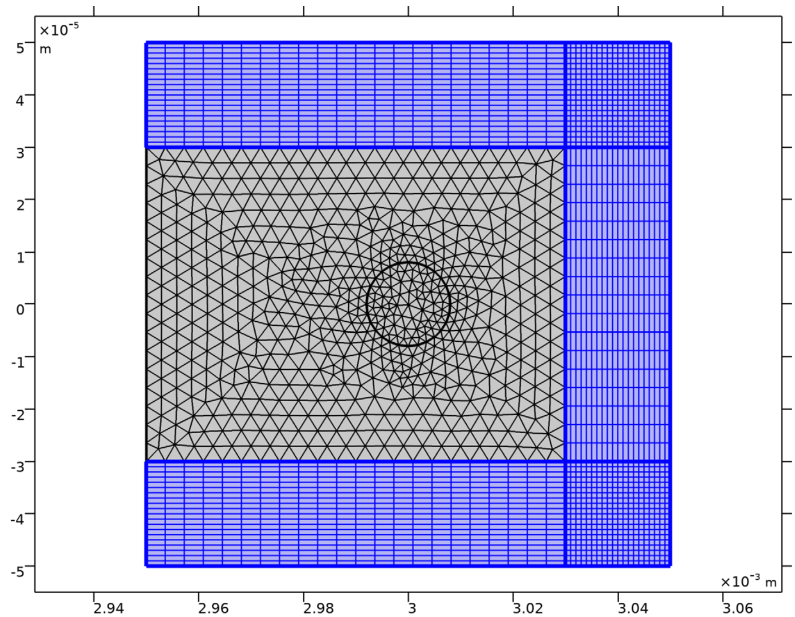
Add an extremely fine mapped mesh for the PML that resolves also the shortest transverse (radial) wavelengths.

1. In the **Mesh** toolbar, click **Mapped**.
2. In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 1 and 3–6 only.

Size 1

1. Right-click **Mapped 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Extremely fine**.

4 Click **Build All**.



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 **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Mode Analysis**.

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

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

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

STUDY 2

Step 1: Mode Analysis

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

2 Select the **Desired number of modes** check box.

3 In the associated text field, type 2.

4 Select the **Search for modes around** check box.

5 In the associated text field, type `nCore*Rb`.

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

7 In the **Model Builder** window, click **Study 2**.

8 In the **Settings** window for **Study**, type `Study 2 (Bent Fiber)` in the **Label** text field.

9 In the **Home** toolbar, click **Compute**.

RESULTS

Solution 3 (Bent Fiber)

Rename the dataset to refer to the Bent Fiber model component.

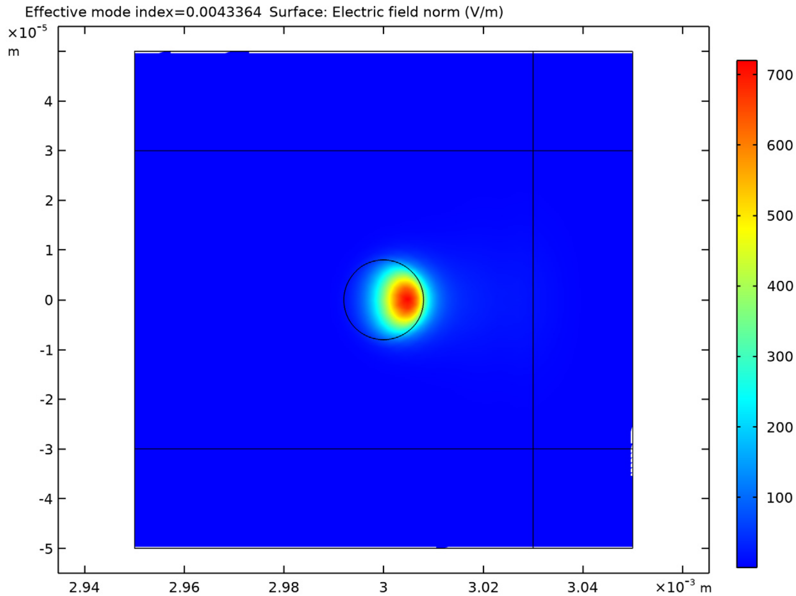
1 In the **Model Builder** window, under **Results>Datasets** click **Study 2 (Bent Fiber)/ Solution 2 (3) (sol2)**.

2 In the **Settings** window for **Solution**, type `Solution 3 (Bent Fiber)` in the **Label** text field.

Electric Field (ewfd2)

1 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

2 From the **Effective mode index** list, choose **0.0043364**.



The default plot shows the norm of the electric field. The selected solution has the field mainly polarized in the *z* direction.

Surface 1

1 In the **Model Builder** window, expand the **Electric Field (ewfd2)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `ewfd2.Ez`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

5 Clear the **Color legend** check box.

Contour 1

1 In the **Model Builder** window, right-click **Electric Field (ewfd2)** and choose **Contour**.

2 In the **Settings** window for **Contour**, locate the **Expression** section.

3 In the **Expression** text field, type `ewfd2.Hphi`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

6 Clear the **Color legend** check box.

Arrow Surface 1

1 Right-click **Electric Field (ewfd2)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Bent Fiber (comp2)> Electromagnetic Waves, Frequency Domain 2>Electric>ewfd2.Er,ewfd2.Ez - Electric field**.

3 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

4 From the **Color** list, choose **Black**.

Solution 3 (Bent Fiber) (sol2)

In the **Model Builder** window, under **Results>Datasets** click **Solution 3 (Bent Fiber) (sol2)**.

Selection

1 Right-click **Solution 3 (Bent Fiber) (sol2)** and choose **Selection**.

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

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

4 Select Domains 2 and 7 only.

Solution 3 (Bent Fiber) (sol2)

1 In the **Model Builder** window, click **Solution 3 (Bent Fiber) (sol2)**.

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

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

Bent Fiber

1 In the **Model Builder** window, under **Results** click **Electric Field (ewfd2)**.

2 In the **Settings** window for **2D Plot Group**, type `Bent Fiber` in the **Label** text field.

3 Click the **Zoom Extents** button in the **Graphics** toolbar. Compare your result with Figure 4.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **File**.

2 In the **Settings** window for **Animation**, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Scene** section. From the **Subject** list, choose **Bent Fiber**.

5 Locate the **Animation Editing** section. From the **Sequence type** list, choose **Dynamic data extension**.

6 Check the **Repeat** check box in the **Playing** section of the **Player** settings.

7 Start the **Player** by clicking the **Play** button in the **Graphics** window.

8 Stop the **Player** by clicking the **Stop** button in the **Graphics** window.

r0

Evaluate the power-averaged mode radius. You should find that the average mode radius is 4 µm larger than the prescribed fiber radius of curvature (3 mm).

1 In the **Results** toolbar, click **8.5 Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `r0` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Solution 3 (Bent Fiber) (sol2)**.

4 From the **Effective mode index selection** list, choose **From list**.

5 In the **Effective mode index** list, select **0.0043364**.

6 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Bent Fiber (comp2)>Definitions>Variables>r0 - m**.

7 Click **Evaluate**.

n_eff_geometry

1 In the **Results** toolbar, click **8.5 Global Evaluation**.

Evaluate the effective index, based on the prescribed radius of curvature of the fiber.

2 In the **Settings** window for **Global Evaluation**, type `n_eff_geometry` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Solution 3 (Bent Fiber) (sol2)**.

4 From the **Effective mode index selection** list, choose **From list**.

5 In the **Effective mode index** list, select **0.0043364**.

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

Expression	**Unit**	**Description**
`real(ewfd2.neff)/Rb`	`1/m`	

7 Click **Evaluate**.

n_eff_power

1 In the **Results** toolbar, click **8.5 Global Evaluation**.

Evaluate the effective index, based on the power-averaged mode radius.

2 In the **Settings** window for **Global Evaluation**, type `n_eff_power` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Solution 3 (Bent Fiber) (sol2)**.

4 From the **Effective mode index selection** list, choose **From list**.

5 In the **Effective mode index** list, select **0.0043364**.

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

Expression	Unit	Description
`real(ewfd2.neff)/r0`	`1/m`	

7 Click **Evaluate**.