Created in COMSOL Multiphysics 6.2

Waveguide S-Bend

Introduction

In optical waveguides, light propagates along a predefined path. The light wave is also confined in the dimensions perpendicular to the propagation direction. This is contrary to plane waves that have infinite spatial extension.

The guiding mechanism is most often based on total internal reflection. That is, if a wave reaches an interface between two media, it will be completely reflected if the refractive index in the incident medium is larger than the refractive index on the transmission side and the angle of incidence is larger than a certain angle.

The material domain with the higher refractive index is normally called the core domain and the surrounding domains are called the cladding domains. The cross section of the core domains can be circular, as for optical fibers (see the Step-Index Fiber Bend model), or rectangular, which is more common for integrated optical waveguides. The Optical Ring Resonator Notch Filter 3D model is an example of a rectangular waveguide structure.

The length of optical waveguides is often much longer than the wavelength of the light that propagates through the guides. To accommodate that requirement, the *Electromagnetic Waves, Beam Envelopes* interface is available in the Wave Optics Module. With that physics interface, it is possible to simulate the propagation of waves over distances that are much longer than the wavelength, as the mesh does not have to resolve the wavelength. Instead, if you define the electric field as

$$\mathbf{E}(\mathbf{r}) = \mathbf{E}_0(\mathbf{r})\exp(-j\phi(\mathbf{r})), \qquad (1)$$

and you can provide a good approximation for the rapidly varying phase function $\phi(\mathbf{r})$, the amplitude or enveloped function $\mathbf{E}_0(\mathbf{r})$ will vary very slowly. Then, when you solve for $\mathbf{E}_0(\mathbf{r})$, a very coarse mesh can be used.

However, a problem with the approach described above, is that it can be difficult to provide the required guess for the phase function $\phi(\mathbf{r})$, if the waveguide structure is more complex. In this model, it is shown how to define auxiliary partial differential equations to solve for a good approximation of the phase function $\phi(\mathbf{r})$.

In addition, since the simulation is performed in 2D and the waveguide length is not too long, a comparison is also made with results using the *Electromagnetic Waves, Frequency Domain* interface. Notice though, that the *Electromagnetic Waves, Frequency Domain* interface cannot be used for solving more realistic optical waveguide problems in 3D and for longer propagation lengths, as that physics interface requires that the mesh resolves the waves on a wavelength scale.

Model Definition

This model simulates a waveguide S-bend. To build the geometry, parts from the Wave Optics Module Parts library are used. This simplifies the process considerably, when creating complex waveguide structures.

An additional advantage gained when using the parts, is that selections can easily be defined for materials and boundary conditions, *etc*.

To provide the required phase input for the *Electromagnetic Waves, Beam Envelopes* interface, two auxiliary *General Form* interfaces are added to the model. The first *General Form* interface computes the propagation length, `s`, along the waveguide core, whereas the second *General Form* interface computes the propagation length in the cladding domain. Using two interfaces, makes it easy to compute the propagation length with high accuracy in the core, where it is especially important. The boundary conditions in the two physics interfaces make sure the propagation length variable, `s`, is everywhere continuous. Furthermore, a constraint is added to make the propagation length constant at the output boundary, after the S-bend. This makes it easier to define a continuous phase expression for all domains, including the Perfectly Matched Layer (PML) domains.

As the waveguide is bent, the excited wave will be converted to higher order modes. To make sure that these modes are absorbed, the waveguide structure is terminated by PML domains.

In Study 1, the propagation length approximation is first solved for using **Stationary** study steps for the *General Form* interfaces. The approximate propagation length is then used for defining the phase expression in **Variable** nodes, found under the **Definitions** node in the **Model Builder** tree. As Numeric ports are used, **Boundary Mode Analysis** study steps are used for solving for the port mode fields. Finally, a **Frequency Domain** study step solves for the electric field envelope, using the previously computed phase approximation and port mode fields.

In Study 2, the electric field is solved for using the *Electromagnetic Waves, Frequency Domain* interface. Also here, Numeric ports, backed by PMLs, are used.

Results and Discussion

Figure 1 shows the electric field norm for the wave propagating along the waveguide S-bend, when using the Electromagnetic Waves, Beam Envelopes interface.

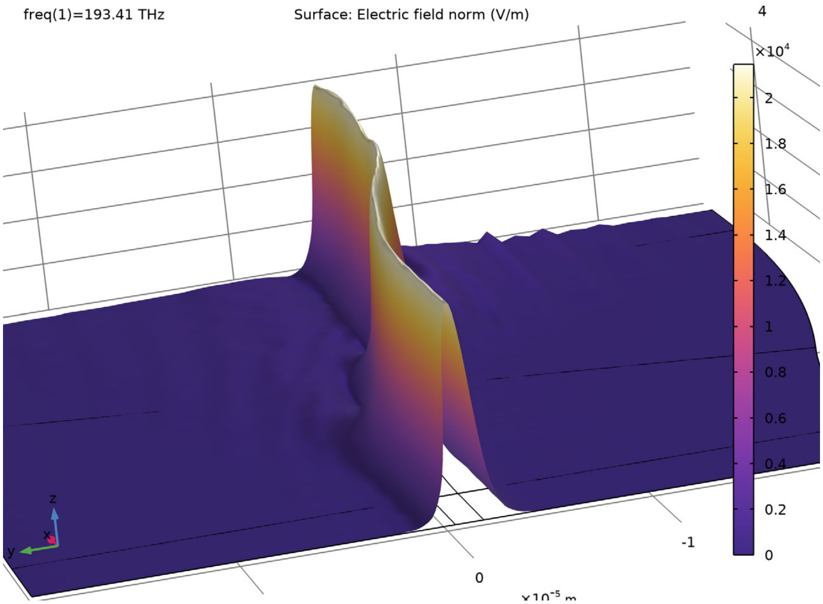


Figure 1: The electric field norm along the waveguide S-bend, when using the Electromagnetic Waves, Beam Envelopes interface.

Figure 2 show the phase approximation, computed by the General Form interfaces and used by the Electromagnetic Waves, Beam Envelopes interface. Notice that it is everywhere continuous and that the phase fronts in the core are orthogonal to the core-cladding boundaries.

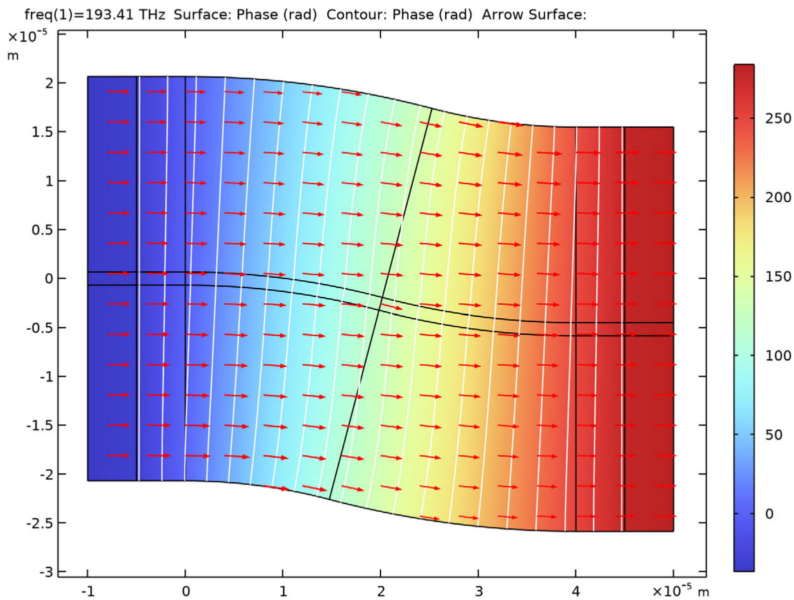


Figure 2: Surface and contour plots of the phase along the waveguide structure. The arrows represent the wave vector, defined as the gradient of the phase, that is used by the Electromagnetic Waves, Beam Envelopes interface.

Figure 3 shows the z component of the electric field envelope. Thanks to the good provided phase approximation, the envelope function has a very slow variation along the waveguide. However, it is clear from the plot that higher order radiation is generated due to the waveguide bends.

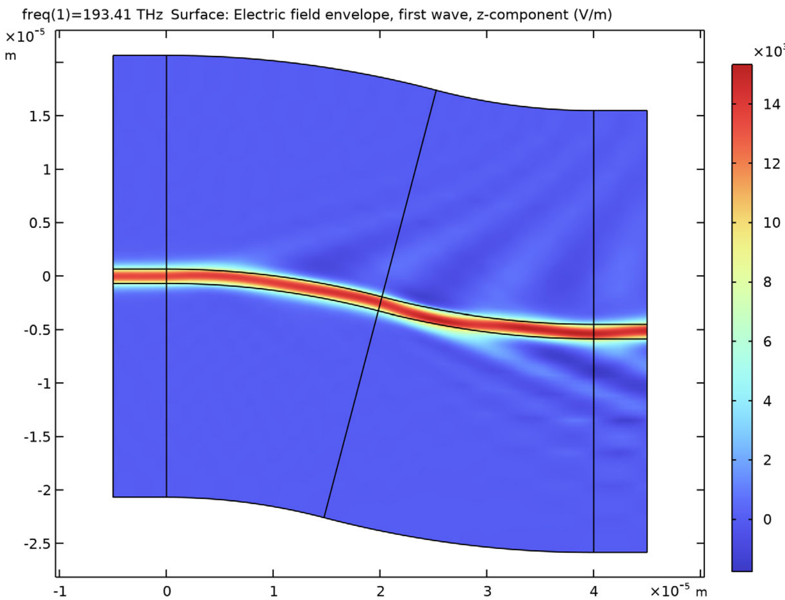


Figure 3: The z component of the electric field envelope.

Figure 4 shows a comparison between the electric field norms, when computed by the Electromagnetic Waves, Beam Envelopes and Electromagnetic Waves, Frequency Domain interfaces, respectively. Notice that the field plots are almost the same for the two interfaces.

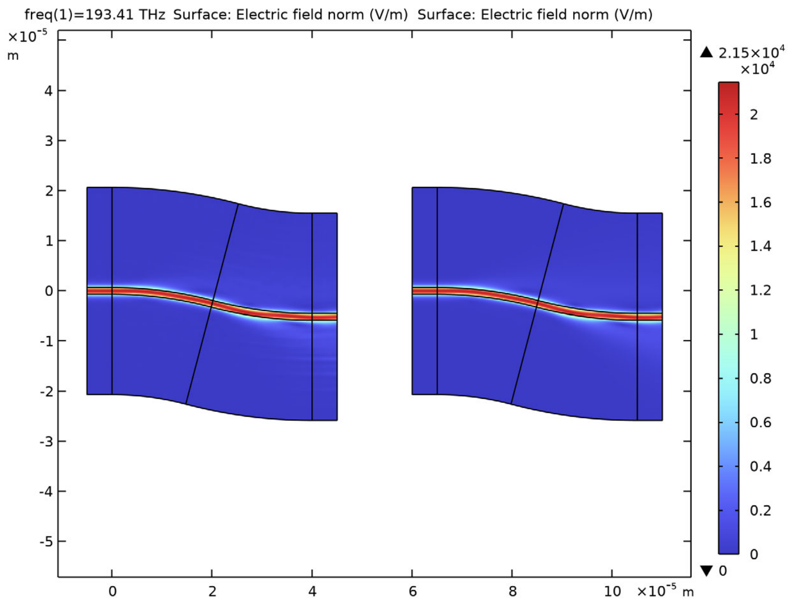


Figure 4: The electric field norms computed by the Electromagnetic Waves, Beam Envelopes (left) and the Electromagnetic Waves, Frequency Domain (right) interfaces, respectively.

Finally, Figure 5 shows a comparison similar to that in Figure 4, but here the plots show the expressions 20*log10(<tag>.normE) (<tag> is the physics interface tag – ewbe and ewfd, respectively). This comparison shows that the field values are very similar for the range from 30 dB to more than 85 dB.

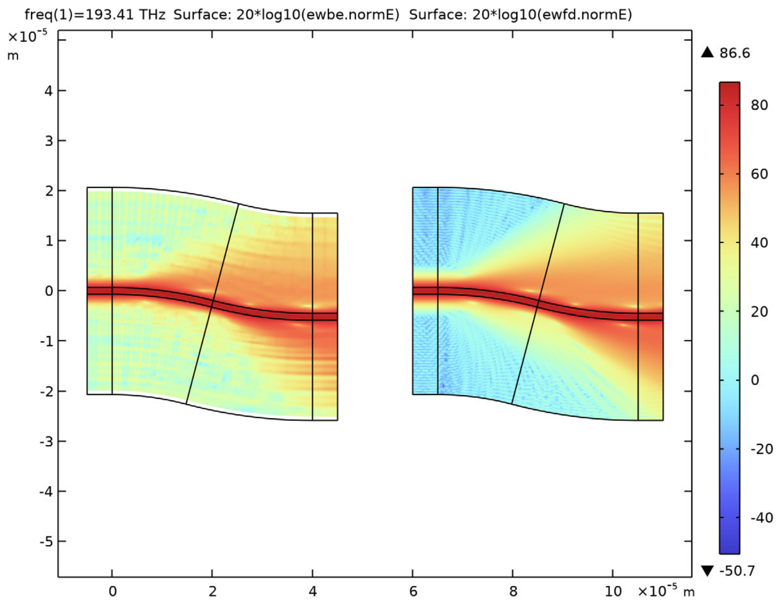


Figure 5: Same kind of comparison as in Figure 4, but using dB scale.

Application Library path: `Wave_Optics_Module/Waveguides/waveguide_s_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, Beam Envelopes (ewbe)**, **Optics>Wave Optics>Electromagnetic Waves, Frequency Domain (ewfd)**, and **Mathematics>PDE Interfaces>General Form PDE (g)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Mathematics>PDE Interfaces>General Form PDE (g)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces> Boundary Mode Analysis**.

8 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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `waveguide_s_bend_parameters.txt`.

The parameters above define the operation wavelength, the geometry dimensions, and the material properties.

PART LIBRARIES

Build the geometry using parts from the Wave Optics Module Parts library.

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **Wave Optics Module>Slab Waveguides> slab_waveguide_s_bend** in the tree.

3 Click **Add to Geometry**.

GEOMETRY 1

Slab Waveguide S-Bend 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Slab Waveguide S-Bend 1 (pi1)**.

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

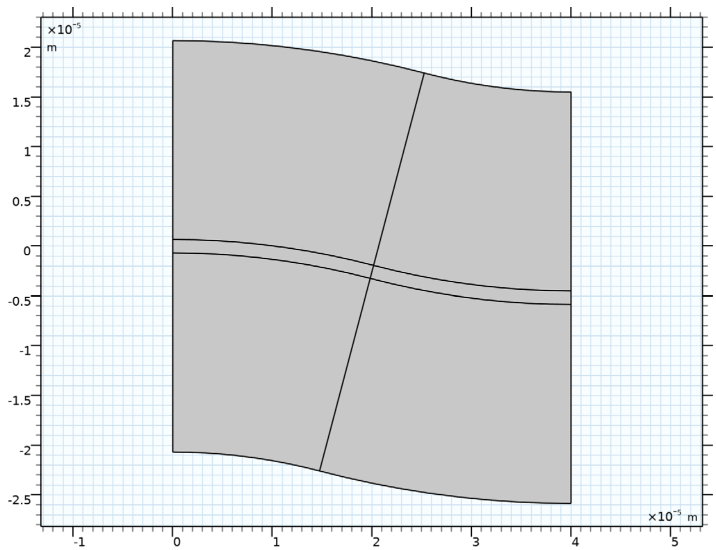
3 In the table, enter the following settings:

Name	Expression	Value	Description
core_width	`w_core`	1.35E-6 m	Core width
cladding_width	`w_clad`	4.135E-5 m	Cladding width
horizontal_displacement	`offset`	5.1687E-6 m	Horizontal displacement
element_length	`d_bent_wg`	4E-5 m	Element length

Notice that the `core_offset` parameter should not be changed.

4 Locate the **Position and Orientation of Output** section. In the **Rotation angle** text field, type `-90`.

5 Click **Build Selected**.



6 Click to expand the **Domain Selections** section. Click **New Cumulative Selection**.

7 In the **New Cumulative Selection** dialog box, type `Core` in the **Name** text field.

8 Click **OK**.

9 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

10 Click **New Cumulative Selection**.

11 In the **New Cumulative Selection** dialog box, type `Cladding` in the **Name** text field.

12 Click **OK**.

13 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

14 Click **New Cumulative Selection**.

15 In the **New Cumulative Selection** dialog box, type `Non-PML` in the **Name** text field.

16 Click **OK**.

17 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

18 In the table, enter the following settings:

Name	Keep	Physics	Contribute to
All		√	Non-PML
Core		√	Core
Cladding		√	Cladding

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **Wave Optics Module>Slab Waveguides> slab_waveguide_straight** in the tree.

3 Click **Add to Geometry**.

GEOMETRY 1

Slab Waveguide Straight 1 (pi2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Slab Waveguide Straight 1 (pi2)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
core_width	`w_core`	1.35E-6 m	Core width
cladding_width	`w_clad`	4.135E-5 m	Cladding width
element_length	`d_straight_wg/2`	5E-6 m	Element length

Again, do not change the value for the `core_offset` parameter.

4 Locate the **Position and Orientation of Output** section. In the **Rotation angle** text field, type 90.

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

Name	Keep	Physics	Contribute to
Core		√	Core
Cladding		√	Cladding
All		√	Non-PML

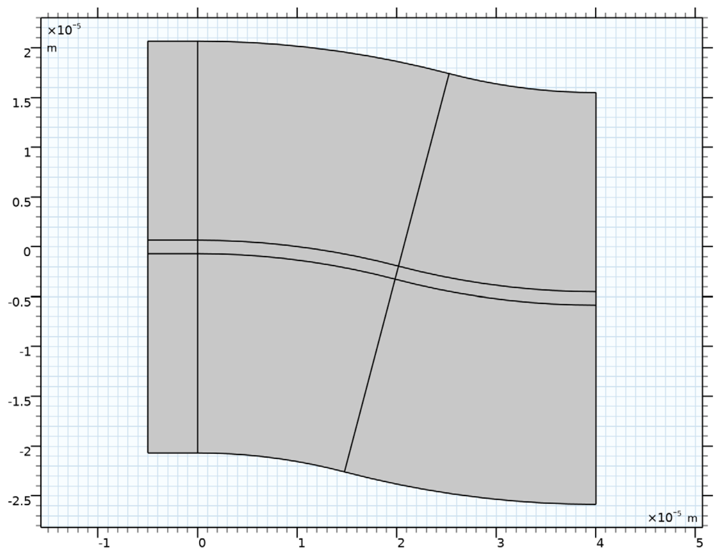
6 Click to expand the **Boundary Selections** section. Click to select row number 7 in the table.

7 Click **New Cumulative Selection**.

8 In the **New Cumulative Selection** dialog box, type `Port 1` in the **Name** text field.

9 Click **OK**.

10 In the **Settings** window for **Part Instance**, click **Build Selected**.



11 Right-click **Component 1 (comp1)>Geometry 1>Slab Waveguide Straight 1 (pi2)** and choose **Duplicate**.

Slab Waveguide Straight 2 (pi3)

1 In the **Model Builder** window, click **Slab Waveguide Straight 2 (pi3)**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `-d_straight_wg/2`.

4 Locate the **Domain Selections** section. Click **New Cumulative Selection**.

5 In the **New Cumulative Selection** dialog box, type `Left PML` in the **Name** text field.

6 Click **OK**.

7 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

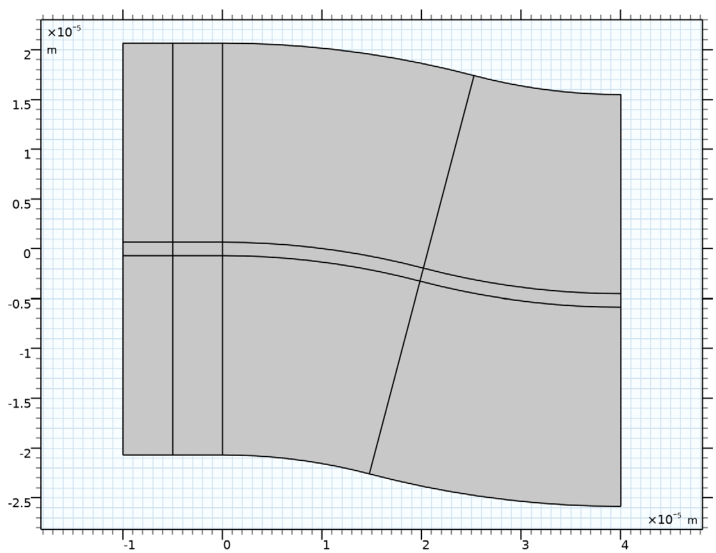
8 In the table, enter the following settings:

Name	Keep	Physics	Contribute to
All		√	Left PML

9 Locate the **Boundary Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Port 2		√	None

10 Click **Build Selected**.



Slab Waveguide Straight 1 (pi2)

In the **Model Builder** window, right-click **Slab Waveguide Straight 1 (pi2)** and choose **Duplicate**.

Slab Waveguide Straight 3 (pi4)

1 In the **Model Builder** window, click **Slab Waveguide Straight 3 (pi4)**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `d_bent_wg`.

4 In the **y-displacement** text field, type `-offset`.

5 In the **Rotation angle** text field, type `-90`.

6 Locate the **Boundary Selections** section. Click **New Cumulative Selection**.

7 In the **New Cumulative Selection** dialog box, type `Port 2` in the **Name** text field.

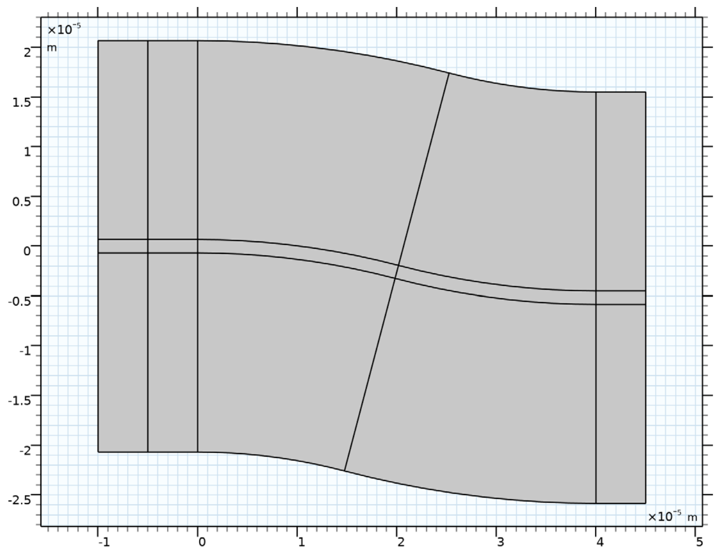
8 Click **OK**.

9 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

10 In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Port 2		√	Port 2

11 Click **Build Selected**.



Slab Waveguide Straight 2 (pi3)

In the **Model Builder** window, right-click **Slab Waveguide Straight 2 (pi3)** and choose **Duplicate**.

Slab Waveguide Straight 4 (pi5)

1 In the **Model Builder** window, click **Slab Waveguide Straight 4 (pi5)**.

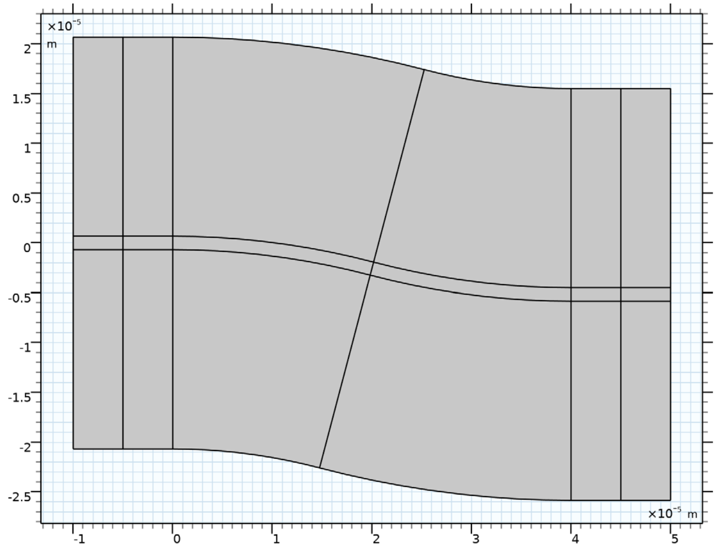
2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 In the **x-displacement** text field, type `d_bent_wg+d_straight_wg/2`.

4 In the **y-displacement** text field, type `-offset`.

5 In the **Rotation angle** text field, type `-90`.

6 Click **Build All Objects**.



7 Locate the **Domain Selections** section. Click **New Cumulative Selection**.

8 In the **New Cumulative Selection** dialog box, type `Right PML` in the **Name** text field.

9 Click **OK**.

10 In the **Settings** window for **Part Instance**, locate the **Domain Selections** section.

11 In the table, enter the following settings:

Name	Keep	Physics	Contribute to
All		√	Right PML

Now, add some selections to make it easier to add selections for materials and physics features.

PML

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `PML` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Left PML** and **Right PML**.

5 Click **OK**.

DEFINITIONS

Non-PML Core

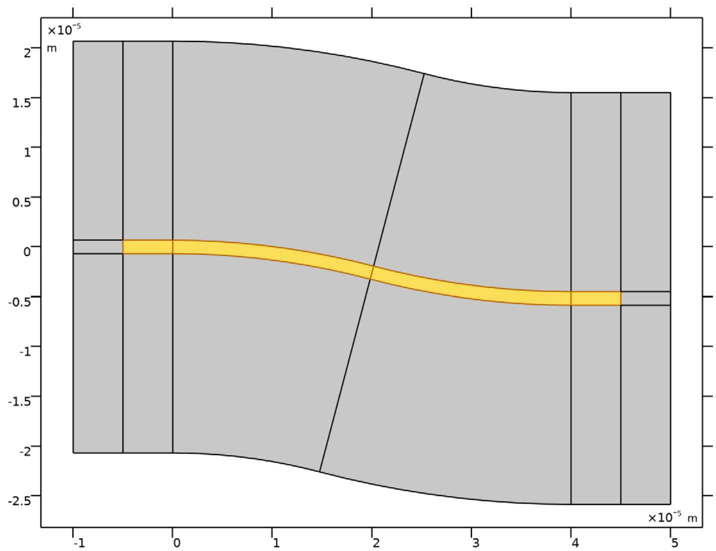
1 In the **Definitions** toolbar, click **Intersection**.

2 In the **Settings** window for **Intersection**, type `Non-PML Core` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

4 In the **Add** dialog box, in the **Selections to intersect** list, choose **Core** and **Non-PML**.

5 Click **OK**.



6 Right-click **Non-PML Core** and choose **Duplicate**.

Non-PML Cladding

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Non-PML Core 1**.

2 In the **Settings** window for **Intersection**, type `Non-PML Cladding` in the **Label** text field.

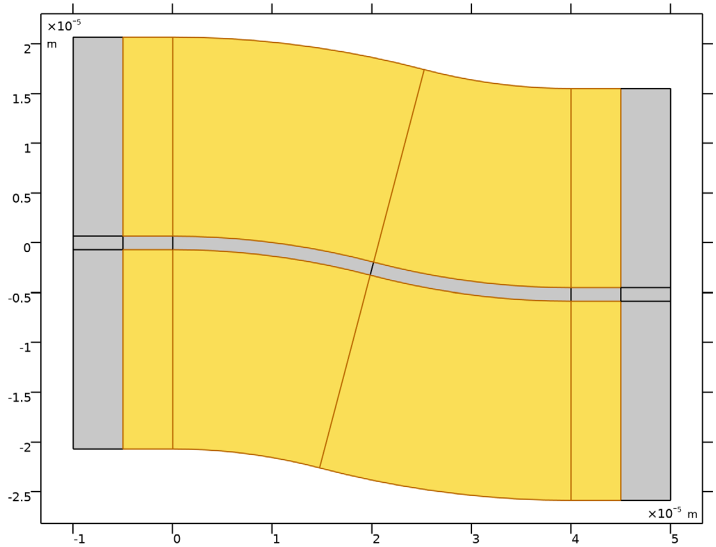
3 Locate the **Input Entities** section. In the **Selections to intersect** list, select **Core**.

4 Under **Selections to intersect**, click **Delete**.

5 Under **Selections to intersect**, click **Add**.

6 In the **Add** dialog box, select **Cladding** in the **Selections to intersect** list.

7 Click **OK**.



Non-PML Core Boundaries

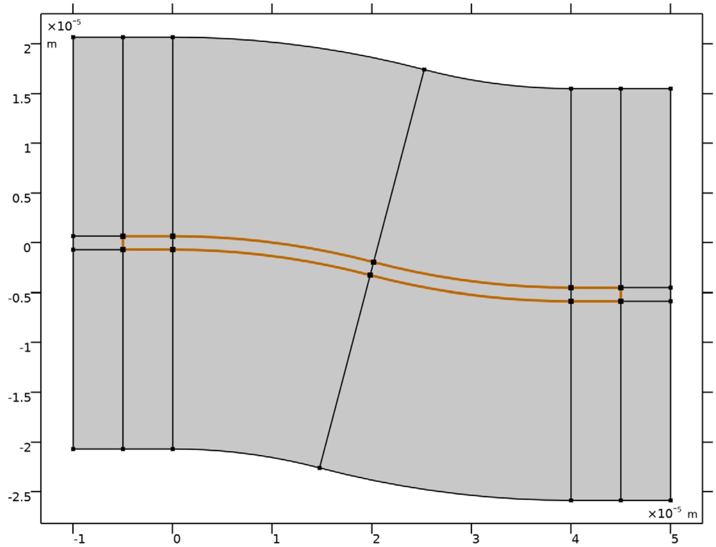
1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Non-PML Core Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Non-PML Core** in the **Input selections** list.

5 Click **OK**.



6 Right-click **Non-PML Core Boundaries** and choose **Duplicate**.

Non-PML Cladding Boundaries

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Non-PML Core Boundaries 1**.

2 In the **Settings** window for **Adjacent**, type `Non-PML Cladding Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. In the **Input selections** list, select **Non-PML Core**.

4 Under **Input selections**, click **Delete**.

5 Under **Input selections**, click **Add**.

6 In the **Add** dialog box, select **Non-PML Cladding** in the **Input selections** list.

7 Click **OK**.

Non-PML Core-Cladding Boundaries

1 In the **Definitions** toolbar, click **Intersection**.

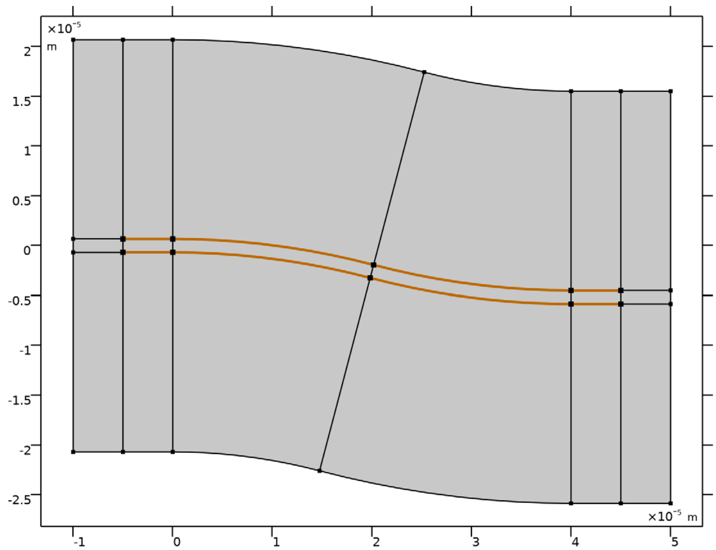
2 In the **Settings** window for **Intersection**, type `Non-PML Core-Cladding Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Non-PML Core Boundaries** and **Non-PML Cladding Boundaries**.

6 Click **OK**.



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_clad`	1	Refractive index

Core

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Core**.

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

ELECTROMAGNETIC WAVES, BEAM ENVELOPES (EWBE)

Port 1

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

2 In the **Settings** window for **Port**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Port 1**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Numeric**.

5 Select the **Activate slit condition on interior port** check box.

6 From the **Slit type** list, choose **Domain-backed**.

7 Click **Toggle Power Flow Direction**, to make the arrows in the **Graphics** window point toward the waveguide S-bend.

Port 2

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

2 In the **Settings** window for **Port**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Port 2**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Numeric**.

5 Select the **Activate slit condition on interior port** check box.

6 From the **Slit type** list, choose **Domain-backed**.

7 Click **Toggle Power Flow Direction**, to make the arrows in the **Graphics** window point away from the S-bend.

Port 1, Port 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Beam Envelopes (ewbe)**, Ctrl-click to select **Port 1** and **Port 2**.

2 Right-click and choose **Copy**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (ewfd)** and choose **Paste Multiple Items**.

GENERAL FORM PDE (G)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **General Form PDE (g)**.

2 In the **Settings** window for **General Form PDE**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Non-PML Core**.

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `Length` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Length (m)**.

8 Click **OK**.

9 In the **Settings** window for **General Form PDE**, locate the **Units** section.

10 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	`m^-1`

General Form PDE 1

1 In the **Model Builder** window, under **Component 1 (comp1)>General Form PDE (g)** click **General Form PDE 1**.

2 In the **Settings** window for **General Form PDE**, locate the **Source Term** section.

3 In the f text field, type `0`.

4 Locate the **Damping or Mass Coefficient** section. In the d_a text field, type `0`.

Dirichlet Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Dirichlet Boundary Condition**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Port 1**.

4 Locate the **Dirichlet Boundary Condition** section. In the r text field, type `-d_straight_wg/2`.

Flux/Source 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Flux/Source**.

2 In the **Settings** window for **Flux/Source**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Port 2**.

4 Locate the **Boundary Flux/Source** section. In the g text field, type `1`, to make the x-derivative of u be 1[m]/1[m] = 1.

Now, add a global variable that will make the path length constant on the **Port 2** boundary.

Global Equations 1 (ODE1)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

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
u0		0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, click **OK**, as `Length` is already entered in the search field.

Constraint 1

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

2 In the **Settings** window for **Constraint**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Port 2**.

4 Locate the **Constraint** section. In the R text field, type `u0-u`. This is the constraint that will make the path length constant on the **Port 2** boundary.

GENERAL FORM PDE 2 (G2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **General Form PDE 2 (g2)**.

2 In the **Settings** window for **General Form PDE**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Non-PML Cladding**.

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, click **Filter**.

6 In the tree, select **General>Length (m)**.

7 Click **OK**. Again, as `Length` is already entered in the search field.

8 In the **Settings** window for **General Form PDE**, locate the **Units** section.

9 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	`m^-1`

GENERAL FORM PDE (G)

Constraint 1, Dirichlet Boundary Condition 1, Flux/Source 1

1 In the **Model Builder** window, under **Component 1 (comp1)>General Form PDE (g)**, Ctrl-click to select **Dirichlet Boundary Condition 1**, **Flux/Source 1**, and **Constraint 1**.

2 Right-click and choose **Copy**.

GENERAL FORM PDE 2 (G2)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **General Form PDE 2 (g2)** and choose **Paste Multiple Items**.

Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)>General Form PDE 2 (g2)** click **Constraint 1**.

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

3 In the R text field, type `u0-u2`.

Dirichlet Boundary Condition 1

In the **Model Builder** window, right-click **Dirichlet Boundary Condition 1** and choose **Duplicate**.

Dirichlet Boundary Condition 2

1 In the **Model Builder** window, click **Dirichlet Boundary Condition 2**.

2 In the **Settings** window for **Dirichlet Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Non-PML Core-Cladding Boundaries**.

4 Locate the **Dirichlet Boundary Condition** section. In the r text field, type `u`.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

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

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **PML**.

4 Locate the **Scaling** section. From the **Typical wavelength from** list, choose **User defined**.

5 In the **Typical wavelength** text field, type `2*pi/ewbe.beta_1`.

6 Right-click **Perfectly Matched Layer 1 (pml1)** and choose **Duplicate**.

Perfectly Matched Layer 2 (pml2)

1 In the **Model Builder** window, click **Perfectly Matched Layer 2 (pml2)**.

2 In the **Settings** window for **Perfectly Matched Layer**, locate the **Scaling** section.

3 In the **Typical wavelength** text field, type `2*pi/ewfd.beta_1`.

Core Path Length

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

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

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Non-PML Core**.

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

Name	Expression	Unit	Description
s	u	m	Path length

6 Right-click **Core Path Length** and choose **Duplicate**.

Cladding Path Length

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Core Path Length 1**.

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Non-PML Cladding**.

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

Name	Expression	Unit	Description
s	u2	m	Path length

Left PML

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Left PML` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Left PML**.

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

Name	Expression	Unit	Description
phi	ewbe.beta_1*pml1.x*1[m]	rad	Phase
k1x	ewbe.beta_1	rad/m	Wave vector, first wave, x-component
k1y	0[rad/m]	rad/m	Wave vector, first wave, y-component

The variables above will be used later when finishing the definitions for the **Electromagnetic Waves, Beam Envelopes** interface.

6 Right-click **Left PML** and choose **Duplicate**.

Non-PML

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Left PML 1**.

2 In the **Settings** window for **Variables**, type `Non-PML` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Non-PML**.

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

Name	Expression	Unit	Description
phi	ewbe.beta_1*s	rad	Phase
k1x	d(phi,x)	rad/m	Wave vector, first wave, x-component
k1y	d(phi,y)	rad/m	Wave vector, first wave, y-component

5 Right-click **Non-PML** and choose **Duplicate**.

Right PML

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Non-PML 1**.

2 In the **Settings** window for **Variables**, type `Right PML` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Right PML**.

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

Name	Expression	Unit	Description
phi	ewbe.beta_1*u0+ ewbe.beta_1* (pml1.x-d_bent_wg-d_straight_wg/2)	rad	Phase
k1x	ewbe.beta_1	rad/m	Wave vector, first wave, x-component
k1y	0[rad/m]	rad/m	Wave vector, first wave, y-component

ELECTROMAGNETIC WAVES, BEAM ENVELOPES (EWBE)

Now, finalize the settings for this physics interface, using the variables defined in the previous steps.

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 **Components** section.

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

4 Locate the **Wave Vectors** section. From the **Number of directions** list, choose **Unidirectional**.

5 From the **Type of phase specification** list, choose **User defined**.

6 In the ϕ_1 text field, type `phi`.

7 Click to expand the **User-Defined Wave Vector Specification** section. Specify the $\mathbf{k}_1$ vector as

k1x	x
k1y	y

We must use the **User definedType of phase specification** with the special variables previously defined, as the default expression (`d(phi,x), d(phi,y)`) for the wave vector would give wrong attenuation in the PML domains.

MAPPED MESH

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

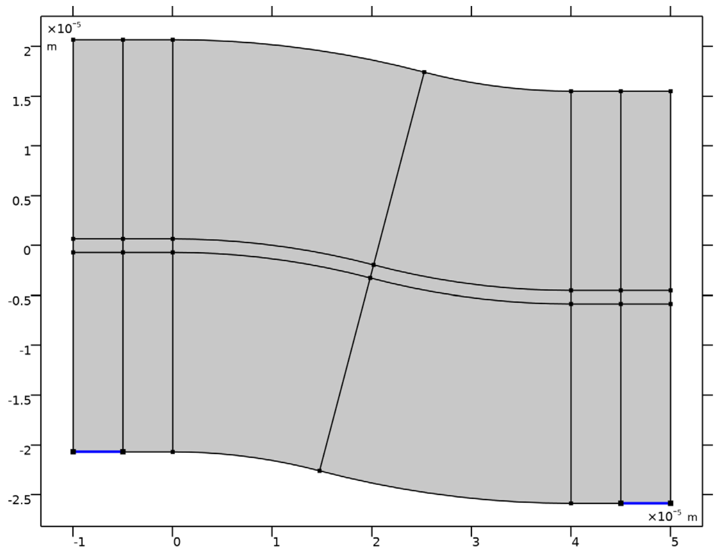
2 In the **Settings** window for **Mesh**, type `Mapped Mesh` in the **Label** text field.

PML Distribution

1 Right-click **Component 1 (comp1)>Mapped Mesh** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, type `PML Distribution` in the **Label** text field.

3 Select Boundaries 2 and 29 only.



4 Locate the **Distribution** section. In the **Number of elements** text field, type `10`.

Size Along Waveguide

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, type `Size Along Waveguide` in the **Label** text field.

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

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `2*lda0`.

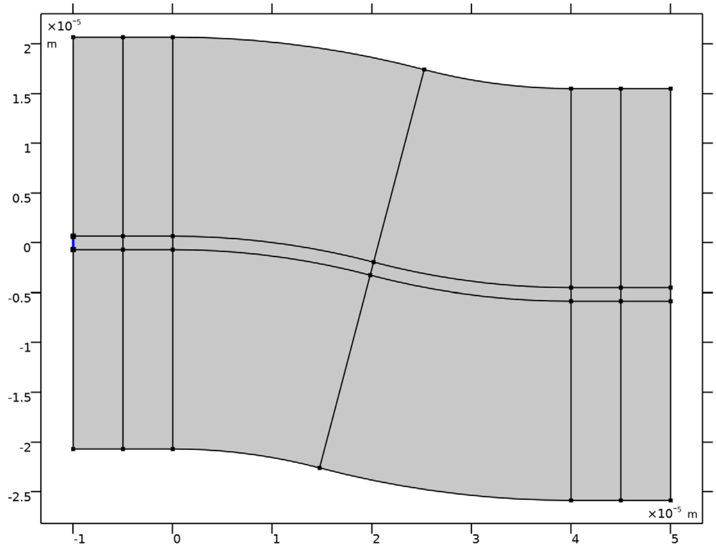
Core Size

1 In the **Model Builder** window, right-click **Mapped Mesh** and choose **Size**.

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

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.



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

6 Locate the **Element Size Parameters** section.

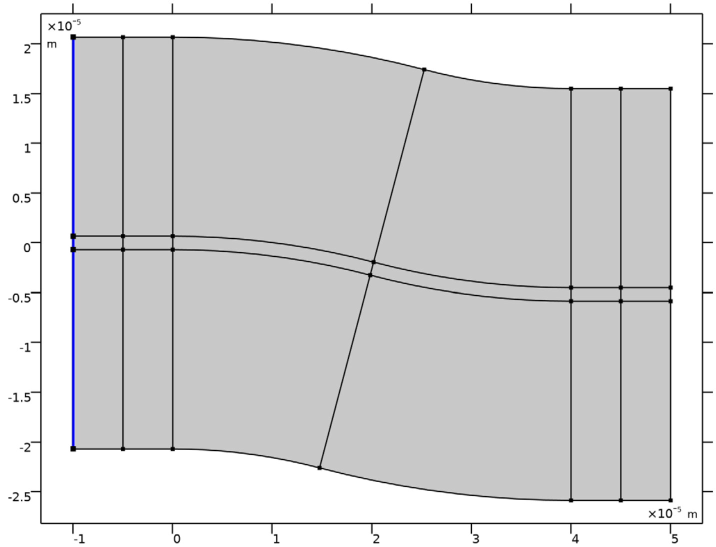
7 Select the **Maximum element size** check box. In the associated text field, type `w_core/3`.

Left Edge

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.

2 In the **Settings** window for **Edge**, type `Left Edge` in the **Label** text field.

3 Select Boundaries 1, 3, and 5 only.



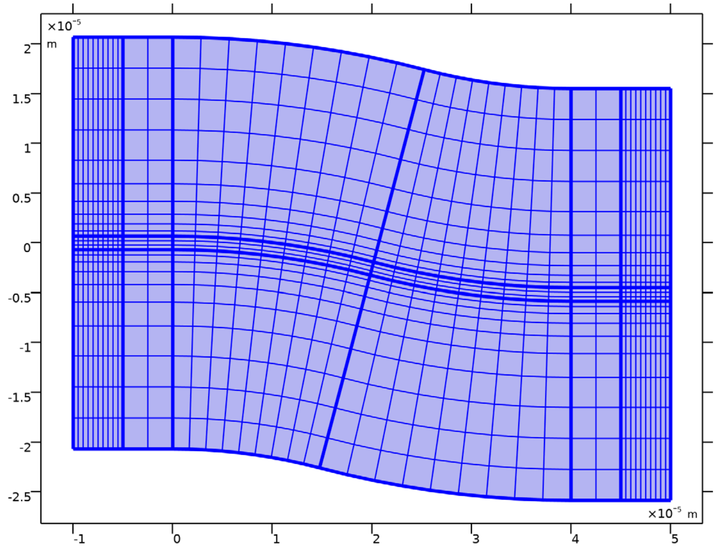
Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, click to expand the **Reduce Element Skewness** section.

3 Select the **Adjust edge mesh** check box.

4 Click **Build All**.



STUDY 1

Step 1: Boundary Mode Analysis

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

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

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

4 Select the **Search for modes around shift** check box. In the associated text field, type `n_core`.

5 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **Electromagnetic Waves, Frequency Domain (ewfd)**, **General Form PDE (g)**, and **General Form PDE 2 (g2)**.

6 Right-click **Study 1>Step 1: Boundary Mode Analysis** and choose **Duplicate**.

Step 3: Boundary Mode Analysis 1

1 In the **Model Builder** window, right-click **Step 3: Boundary Mode Analysis 1** and choose **Move Up**.

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

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

Step 3: Frequency Domain

1 In the **Model Builder** window, click **Step 3: Frequency Domain**.

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

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

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Definitions>Artificial Domains>Perfectly Matched Layer 2 (pml2)**.

6 Click **Disable**.

7 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (ewfd)**.

8 Click **Disable in Solvers**.

9 In the tree, select **Component 1 (comp1)>General Form PDE (g)**.

10 Click **Disable in Solvers**.

11 In the tree, select **Component 1 (comp1)>General Form PDE 2 (g2)**.

12 Click **Disable in Solvers**.

Step 4: Stationary

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.

2 Drag and drop above **Step 2: Boundary Mode Analysis**.

3 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

4 In the table, clear the **Solve for** check box for **General Form PDE 2 (g2)**.

5 Right-click **Step 1: Stationary** and choose **Duplicate**.

Step 5: Stationary 1

1 In the **Model Builder** window, click **Step 5: Stationary 1**.

2 Drag and drop below **Step 1: Stationary**.

3 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

4 In the table, enter the following settings:

Physics interface	**Solve for**	**Equation form**
General Form PDE (g)		Automatic (Stationary)
General Form PDE 2 (g2)	√	Automatic (Stationary)

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

RESULTS

Electric Field (ewbe)

1 In the **Settings** window for **2D Plot Group**, click to expand the **Selection** section.

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

3 From the **Selection** list, choose **Non-PML**.

4 Select the **Apply to dataset edges** check box.

5 In the **Electric Field (ewbe)** toolbar, click **Plot**.

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

Height Expression 1

1 In the **Model Builder** window, expand the **Electric Field (ewbe)** node.

2 Right-click **Electric Field** and choose **Height Expression**.

3 In the **Settings** window for **Height Expression**, locate the **Axis** section.

4 Select the **Scale factor** check box. In the associated text field, type `5e-10`.

Electric Field

1 In the **Model Builder** window, click **Electric Field**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

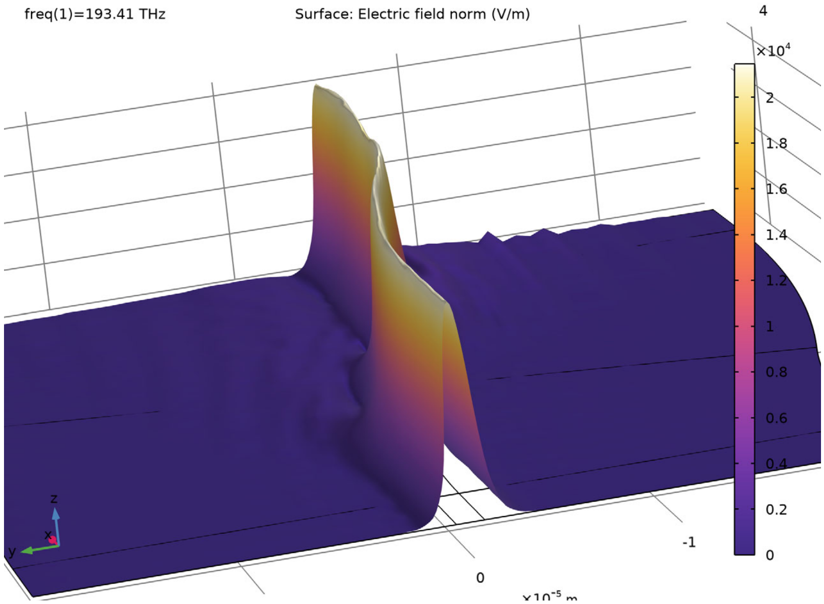
3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

5 Click **OK**.

6 In the **Electric Field (ewbe)** toolbar, click **Plot**.

7 Click the **Zoom Box** button on the **Graphics** toolbar and then use the mouse to zoom in.



This plot shows how the wave propagates along the S-bend.

Phase

1 In the **Model Builder** window, under **Results** click **General Form PDE**.

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

Surface 1

1 In the **Model Builder** window, expand the **Phase** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `phi`.

Contour 1

1 In the **Model Builder** window, right-click **Phase** and choose **Contour**.

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

3 In the **Expression** text field, type `phi`.

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

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

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

Arrow Surface 1

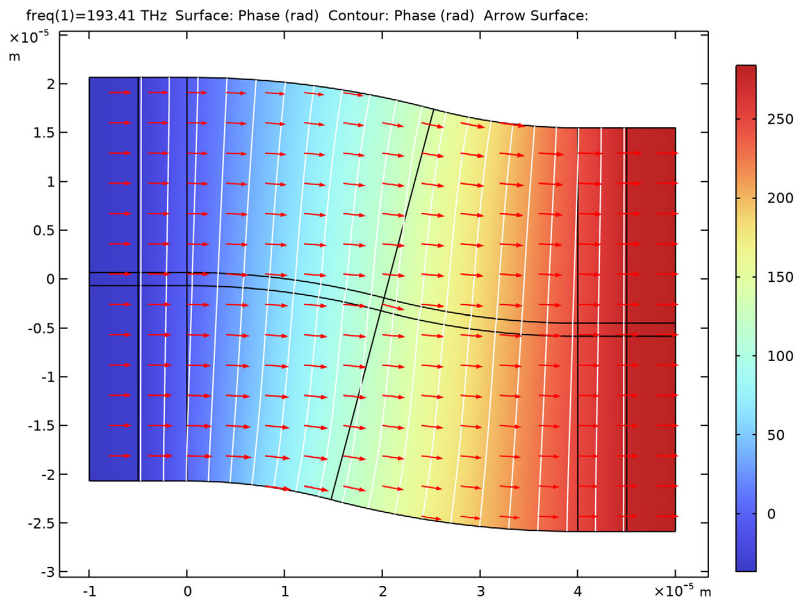
1 Right-click **Phase** and choose **Arrow Surface**.

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

3 In the **X-component** text field, type `k1x`.

4 In the **Y-component** text field, type `k1y`.

5 In the **Phase** toolbar, click **Plot**.



Amplitude, First Wave

1 In the **Model Builder** window, under **Results** click **General Form PDE 2**.

2 In the **Settings** window for **2D Plot Group**, type `Amplitude, First Wave` in the **Label** text field.

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

4 From the **Selection** list, choose **Non-PML**.

5 Select the **Apply to dataset edges** check box.

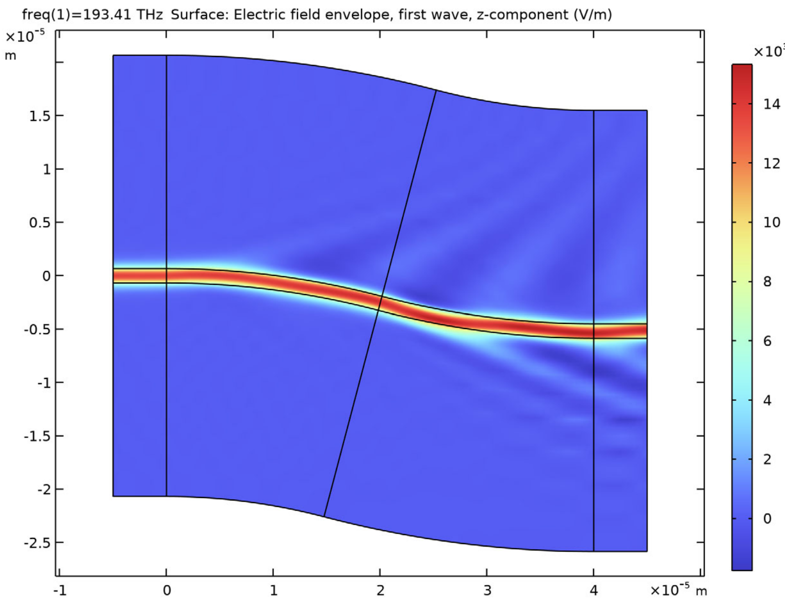
Surface 1

1 In the **Model Builder** window, expand the **Amplitude, First Wave** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `E1z`. This is the dependent variable solved for, the *z* component of the electric field envelope.

4 In the **Amplitude, First Wave** toolbar, click **Plot**.



Notice that the amplitude is almost constant along the propagation path. However, it is also clear that part of the wave power is converted to higher order modes.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

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**.

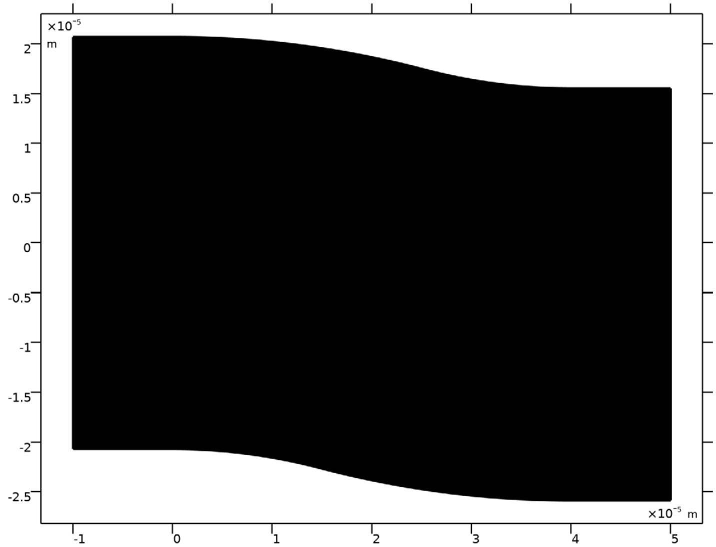
TRIANGULAR MESH

1 In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

2 In the **Settings** window for **Mesh**, type `Triangular Mesh` in the **Label** text field.

3 Locate the **Physics-Controlled Mesh** section. In the table, clear the **Use** check boxes for **Electromagnetic Waves, Beam Envelopes (ewbe)**, **General Form PDE (g)**, and **General Form PDE 2 (g2)**.

4 Click **Build All**.



Notice that this mesh is much denser than the mesh used by the **Electromagnetic Waves, Beam Envelopes** interface.

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 **Empty Study**.

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

STUDY 1

Step 3: Boundary Mode Analysis, Step 4: Boundary Mode Analysis 1, Step 5: Frequency Domain

1 In the **Model Builder** window, under **Study 1**, Ctrl-click to select **Step 3: Boundary Mode Analysis**, **Step 4: Boundary Mode Analysis 1**, and **Step 5: Frequency Domain**.

2 Right-click and choose **Copy**.

STUDY 2

In the **Model Builder** window, right-click **Study 2** and choose **Paste Multiple Items**.

Step 1: Boundary Mode Analysis

1. In the **Model Builder** window, under **Study 2** click **Step 1: Boundary Mode Analysis**.
2. In the **Settings** window for **Boundary Mode Analysis**, locate the **Physics and Variables Selection** section.
3. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Electromagnetic Waves, Beam Envelopes (ewbe)		Automatic (Frequency domain)
Electromagnetic Waves, Frequency Domain (ewfd)	√	Automatic (Boundary mode analysis)

4. Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Triangular Mesh

Step 2: Boundary Mode Analysis 1

1. In the **Model Builder** window, click **Step 2: Boundary Mode Analysis 1**.
2. In the **Settings** window for **Boundary Mode Analysis**, locate the **Physics and Variables Selection** section.
3. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Electromagnetic Waves, Beam Envelopes (ewbe)		Automatic (Frequency domain)
Electromagnetic Waves, Frequency Domain (ewfd)	√	Automatic (Boundary mode analysis)

4. Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Triangular Mesh

Step 3: Frequency Domain

1. In the **Model Builder** window, click **Step 3: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)>Definitions>Artificial Domains>Perfectly Matched Layer 1 (pml1)**.
4 Click **Disable**.
5 In the tree, select **Component 1 (comp1)>Definitions>Artificial Domains>Perfectly Matched Layer 2 (pml2)**.
6 Click **Enable**.
7 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Beam Envelopes (ewbe)**.
8 Click **Disable in Solvers**.
9 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (ewfd)**.
10 Click **Solve For**.
11 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Triangular Mesh

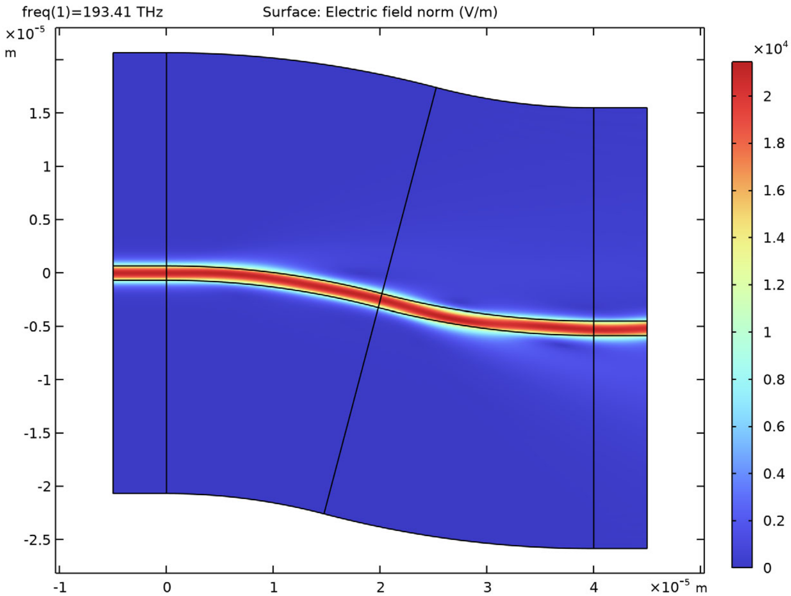
12 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.
13 In the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (ewfd)

1 In the **Settings** window for **2D Plot Group**, locate the **Selection** section.
2 From the **Geometric entity level** list, choose **Domain**.
3 From the **Selection** list, choose **Non-PML**.
4 Select the **Apply to dataset edges** check box.

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



This plot is very similar to the corresponding plot for the **Electromagnetic Waves, Beam Envelopes** interface. Next, we will add two plots comparing the electric fields computed by the two different electromagnetic waves physics interfaces.

Field Comparison, Linear Scale

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Field Comparison, Linear Scale` in the **Label** text field.

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

4 From the **Selection** list, choose **Non-PML**.

5 Select the **Apply to dataset edges** check box.

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

7 Click to expand the **Plot Array** section. Select the **Enable** check box.

Surface 1

1 Right-click **Field Comparison, Linear Scale** and choose **Surface**.

2 Right-click **Surface 1** and choose **Duplicate**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

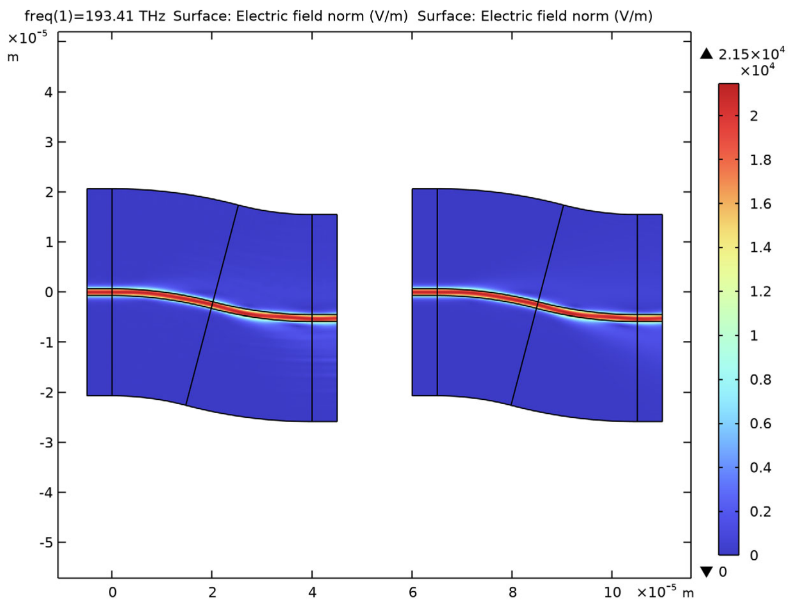
3 From the **Dataset** list, choose **Study 2/Solution 6 (sol6)**.

4 Locate the **Expression** section. In the **Expression** text field, type `ewfd.normE`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

6 In the **Field Comparison, Linear Scale** toolbar, click **Plot**.

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



This plot confirms that the result from the two different physics interfaces produce almost the same results.

Field Comparison, Linear Scale

Now, create a final plot comparing the electric fields, using a logarithmic scale.

In the **Model Builder** window, right-click **Field Comparison, Linear Scale** and choose **Duplicate**.

Field Comparison, Log Scale

1 In the **Model Builder** window, expand the **Results>Field Comparison, Linear Scale 1** node, then click **Field Comparison, Linear Scale 1**.

2 In the **Settings** window for **2D Plot Group**, type `Field Comparison, Log Scale` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, click **Surface 1**.

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

3 In the **Expression** text field, type `20*log10(ewbe.normE)`.

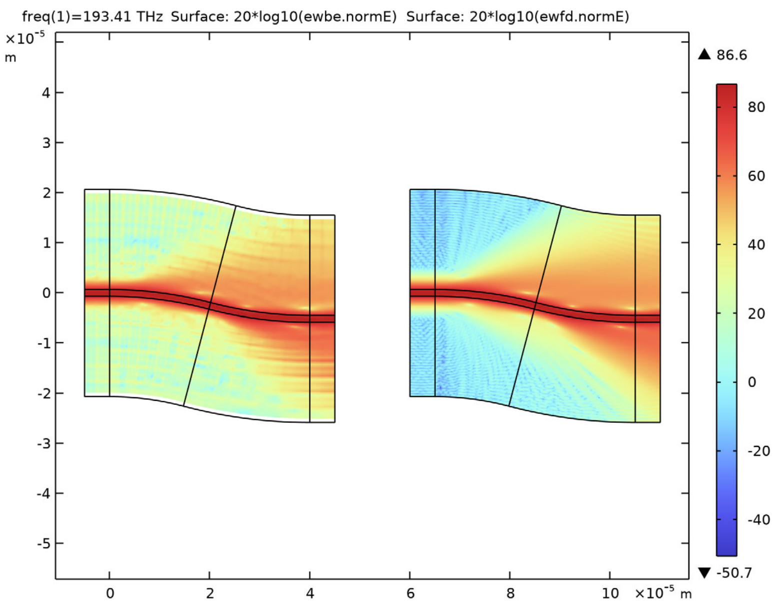
Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 In the **Expression** text field, type `20*log10(ewfd.normE)`.

4 In the **Field Comparison, Log Scale** toolbar, click **Plot**.



Also on a logarithmic scale, the results are very similar, for value above 40 dB.