Beam Splitter

Introduction

A beam splitter is used for splitting a beam of light in two. One way of making a splitter is to deposit a thin layer of metal between two glass prisms. The beam is slightly attenuated within the layer and then split into two paths. This example models the thin metal layer using a transition boundary condition, which reduces the memory requirements. Losses in the metal layer are also computed.
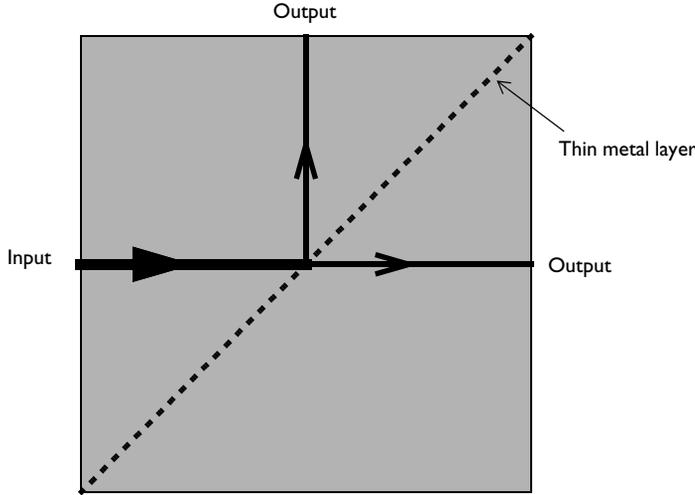

Figure 1: A beam splitter composed of two prisms with a thin layer of metal between them.

Model Definition

Model the beam splitter in the 2D plane, as shown in Figure 1, under the assumption that the electric field is polarized perpendicular to the plane. A Gaussian beam of wavelength 700 nm propagates in the x direction through the glass prism of refractive index $n = 1.5$. A 13 nm thin layer of silver sandwiched between the two prisms splits the beams.

The model geometry is a square region around the region where the Gaussian beam crosses the silver layer. The focus of the beam is at the left boundary, so the expression for the beam intensity at the focal plane can be used as the excitation. The expression for the relative electric field intensity at the focal plane of a Gaussian is

$$E(y) = \exp\left(-\left(\frac{y}{w}\right)^2\right)$$

where w = 3500 nm is the beam waist, and the y = 0 line is the centerline of the beam. Use this expression in a Scattering Boundary Condition on the left side to model the incident beam. Model all the other domain boundaries using Scattering Boundary Conditions. These conditions are appropriate when they are placed several wavelengths away from any scattering objects and the wave is known to be traveling at normal or almost normal incidence.

The thin silver layer is modeled using a Transition Boundary Condition. At a free-space wavelength of 700 nm, the dielectric of silver is about ε_r = −16.5 −1.06i, where the imaginary part accounts for the losses. Thus, you can set the conductivity of the metal to zero. This boundary condition allows for a discontinuity in the fields across the interface by splitting the mesh at the boundary. It can introduce both losses and a phase shift across the interface. It does not require a mesh of the thickness of the domain, and thus saves significant memory. Mesh the two domains with triangular elements, with the maximum size set such that there are six elements per wavelength in the glass.

The second part of this tutorial uses the Electromagnetic Waves, Beam Envelopes interface to model the beam splitter. In this case a coarser mesh is used, making the simulation time much shorter.

Results and Discussion

Figure 2 shows the electric field intensity in the modeling domain. The beam is split into two beams, one propagating in the x direction and the other one in the y direction. The splitting can be evaluated by computing the flux crossing the incoming boundary and the two outgoing boundaries. Figure 3 plots the power crossing these boundaries as well as the losses at the mirror.

Figure 4 also shows the electric field norm, as in Figure 2, but is the result of using the Electromagnetic Waves, Beam Envelopes interface.
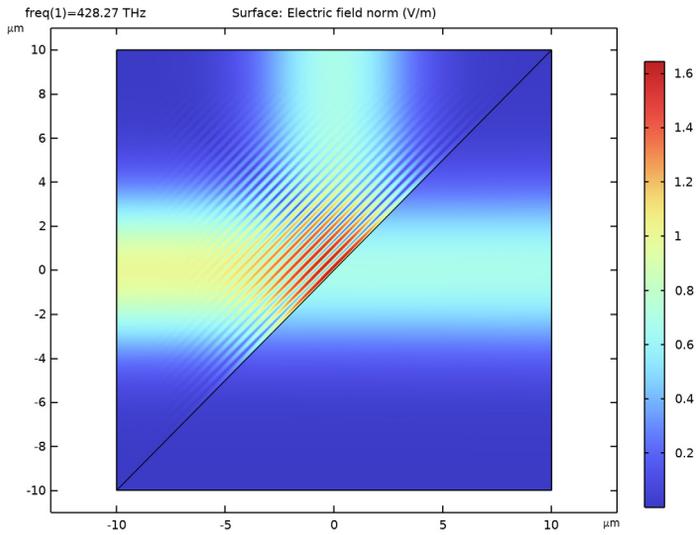
Figure 2: The electric field intensity shows that the incoming beam is split into two beams of approximately equal intensity.
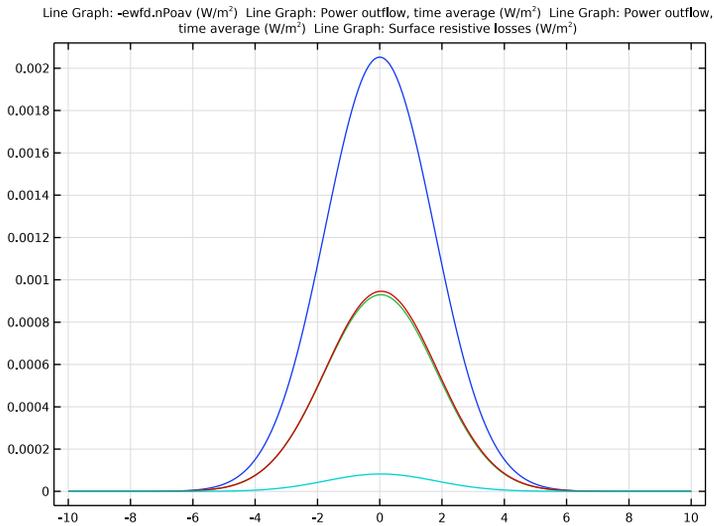

Figure 3: The power flux crossing the input boundary and the two output boundaries as well as the losses at the silver surface.
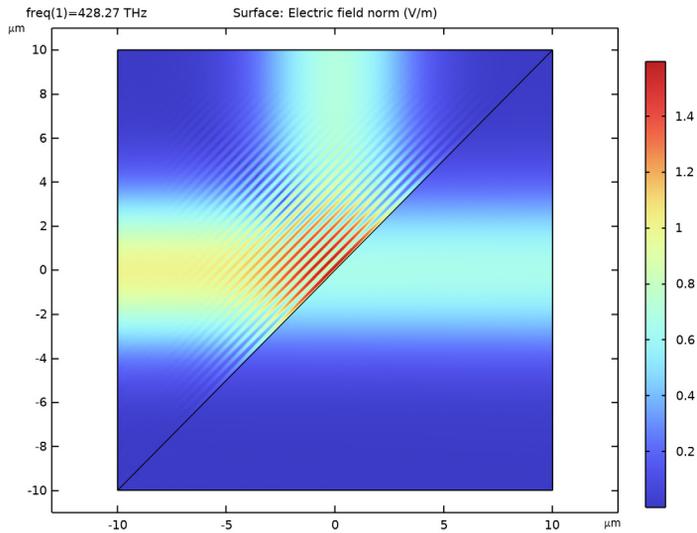
freq(1)=428.27 THz   Surface: Electric field norm (V/m)

Figure 4: The electric field intensity when simulated with the Electromagnetic Waves, Beam Envelopes interface (compare with Figure 2*).*

Application Library path: `Wave_Optics_Module/Optical_Scattering/`
`beam_splitter`

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 **General Studies** > **Frequency Domain**.

6 Click ☑ **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
lda0	700[nm]	7E-7 m	Vacuum wavelength
lda	lda0/1.5	4.6667E-7 m	Material wavelength
f0	c_const/lda0	4.2827E14 1/s	Frequency
eps_Ag	-16.5-1.06*i	-16.5-1.06i	Relative dielectric constant, Silver
w0	5*lda0	3.5E-6 m	Spot radius

Here, c_const is a predefined COMSOL constant for the speed of light in vacuum and lda is the material wavelength, defined as the vacuum wavelength divided by the refractive index. In this case, 1.5 is the refractive index of the glass material that will be used in the model.

GEOMETRY 1

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

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

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

Create a triangle using a **Polygon** for one prism.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click ⟋⏍ **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (µm)	y (µm)
-10	-10
-10	10
10	10

4 Click **Build Selected**.

Rotate the triangle to create the other prism.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.
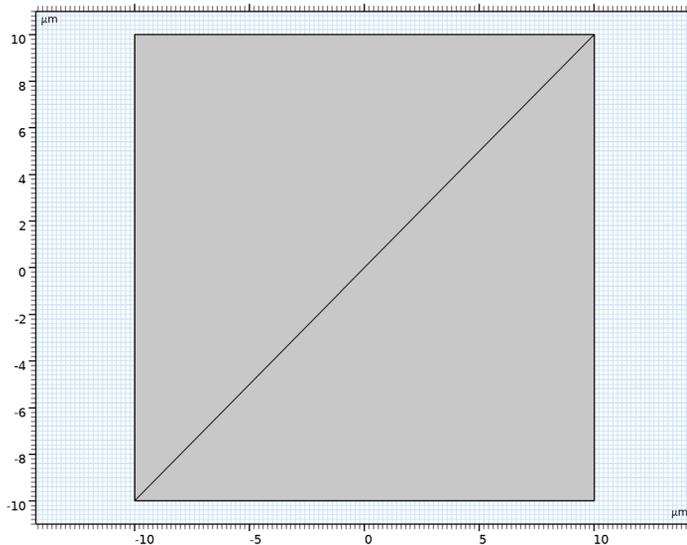
2 Select the object **po11** only.

3 In the **Settings** window for **Rotate**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

5 Locate the **Rotation** section. In the **Angle** text field, type 180.

6 Click **Build All Objects**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

Now set up the physics. Since we are only exciting the out-of-plane component of the electric field, we only need to solve for that component too.

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**.

Scattering Boundary Condition 1

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Scattering Boundary Condition**.
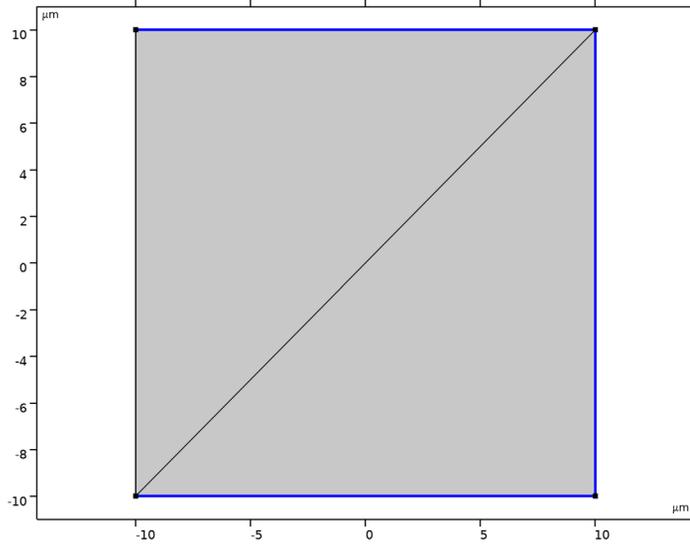
2 Select Boundary 1 only.

3 In the **Settings** window for **Scattering Boundary Condition**, locate the **Scattering Boundary Condition** section.

4 From the **Incident field** list, choose **Gaussian beam**.

5 In the w_0 text field, type w0.

6 Specify the $\mathbf{E}_{g0}$ vector as

0	x
0	y
1[V/m]	z

Scattering Boundary Condition 2

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Scattering Boundary Condition**.
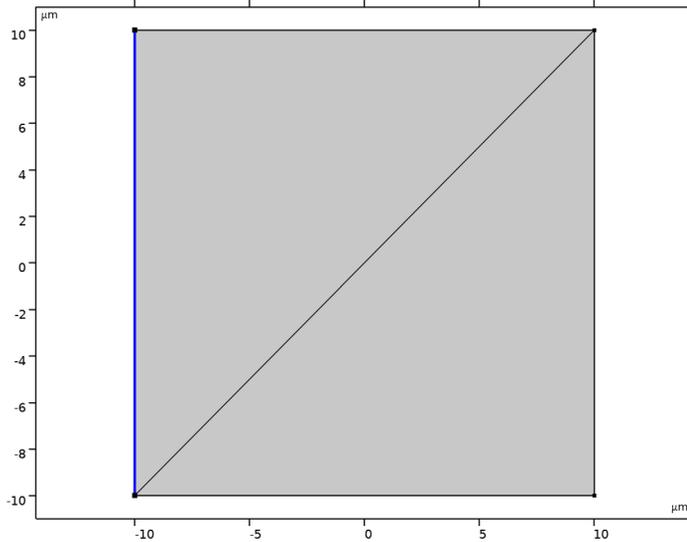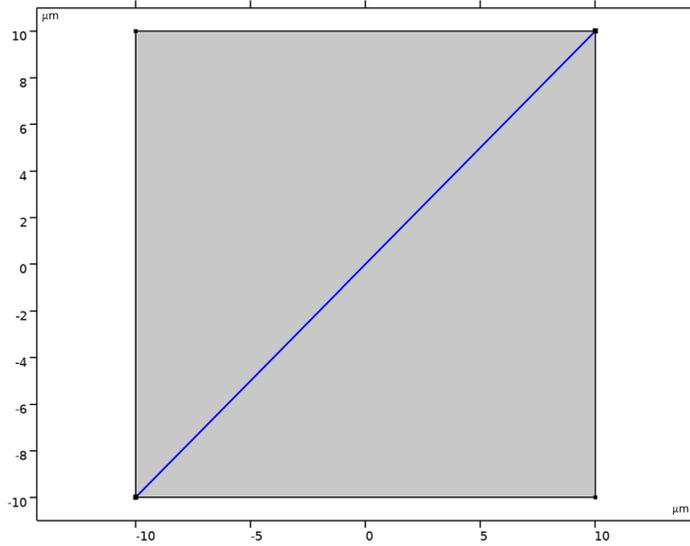
2 Select Boundaries 2, 4, and 5 only.

Transition Boundary Condition 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Transition Boundary Condition**, locate the **Transition Boundary Condition** section.

4 From the **Electric displacement field model** list, choose **Relative permittivity**.

5 From the ε_r list, choose **User defined**. In the associated text field, type eps_Ag.

6 From the μ_r list, choose **User defined**. Leave the default value of 1.

7 From the σ list, choose **User defined**. Leave the default value of 0.

8 In the d text field, type 13[nm].

MATERIALS

Next, assign material properties. Use Glass (quartz) for all domains.

ADD MATERIAL

1 In the **Materials** toolbar, click ⏬ **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Glass (quartz)**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click ⏬ **Add Material** to close the **Add Material** window.

STUDY 1

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type f0.

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

RESULTS

Electric Field (ewfd)

The default plot shows the E-field norm. Compare the plot with Figure 2.

Follow the steps below to reproduce the plot in Figure 3.

1D Plot Group 2

In the **Results** toolbar, click 〰 **1D Plot Group**.

Line Graph 1

1 Right-click **1D Plot Group 2** and choose **Line Graph**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type -ewfd.nPoav.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type y.

Line Graph 2

1 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Line Graph**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type ewfd.nPoav.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type y.

Line Graph 3

1 Right-click **1D Plot Group 2** and choose **Line Graph**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type ewfd.nPoav.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type x.

Line Graph 4

1 Right-click **1D Plot Group 2** and choose **Line Graph**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type ewfd.Qsrh.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type x.

7 In the **1D Plot Group 2** toolbar, click ▣ **Plot**.

The plot describes the power flux crossing the input boundary and the two output boundaries together with the losses at the silver surface. Compare with Figure 3.

ADD PHYSICS

The following instructions illustrate how this problem is solved with the **Electromagnetic Waves, Beam Envelopes (ewbe)** interface.

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, Beam Envelopes (ewbe)**.

4 Click the **Add to Component 1** button in the window toolbar.

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

ELECTROMAGNETIC WAVES, BEAM ENVELOPES (EWBE)

1 In the **Settings** window for **Electromagnetic Waves, Beam Envelopes**, locate the **Components** section.

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

**3** Locate the **Wave Vectors** section. Specify the $\mathbf{k}_1$ vector as

ewbe.k	x
0	y

4 Specify the $\mathbf{k}_2$ vector as

0	x
ewbe.k1x	y

Scattering Boundary Condition 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Scattering Boundary Condition**, locate the **Scattering Boundary Condition** section.

4 From the **Incident field** list, choose **Gaussian beam**.
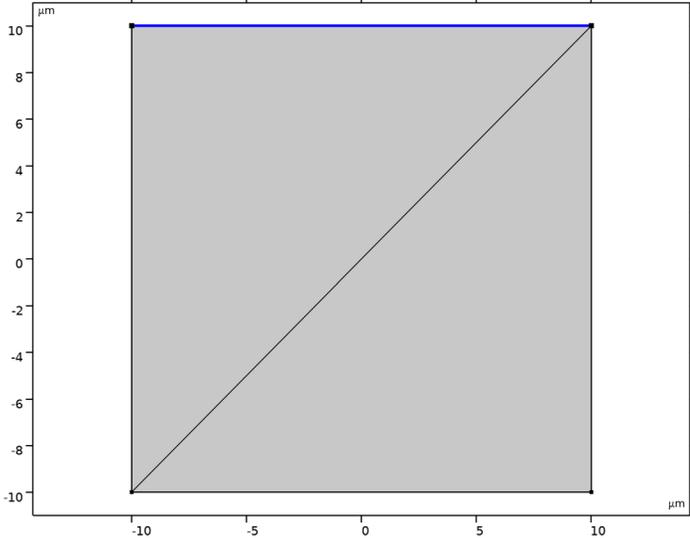
5 In the w_0 text field, type w0.

6 Specify the $\mathbf{E}_{g0}$ vector as

0	x
0	y
1[V/m]	z

Scattering Boundary Condition 2

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Scattering Boundary Condition**.
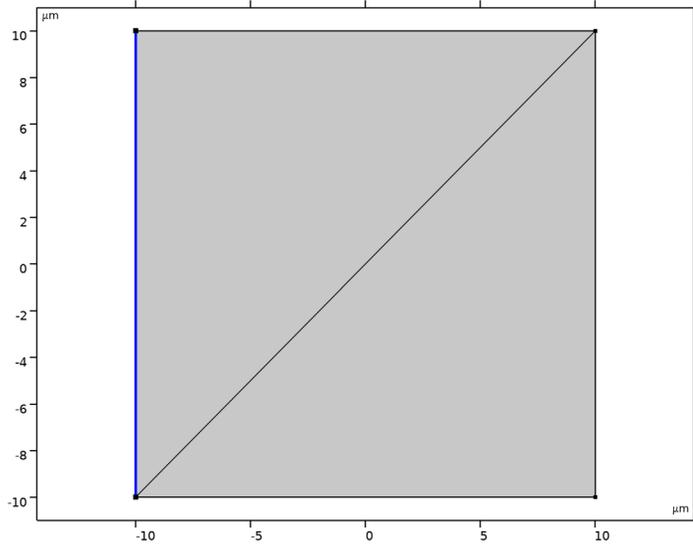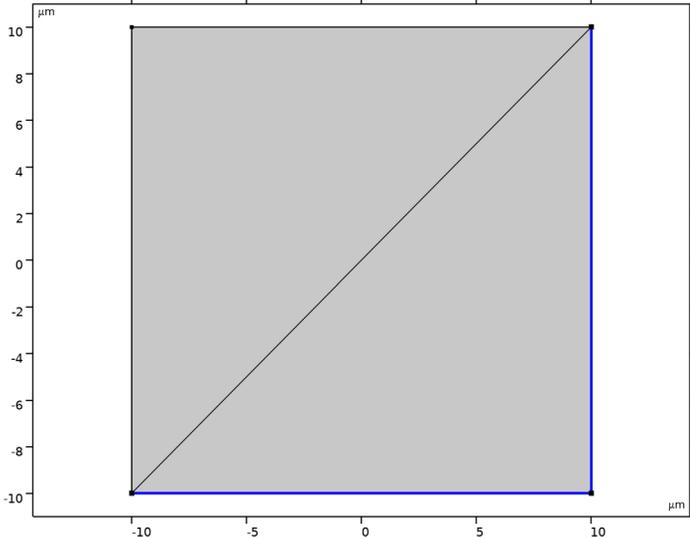
2 Select Boundary 4 only.

Scattering Boundary Condition 3

1 In the **Physics** toolbar, click ⬭ **Boundaries** and choose **Scattering Boundary Condition**.
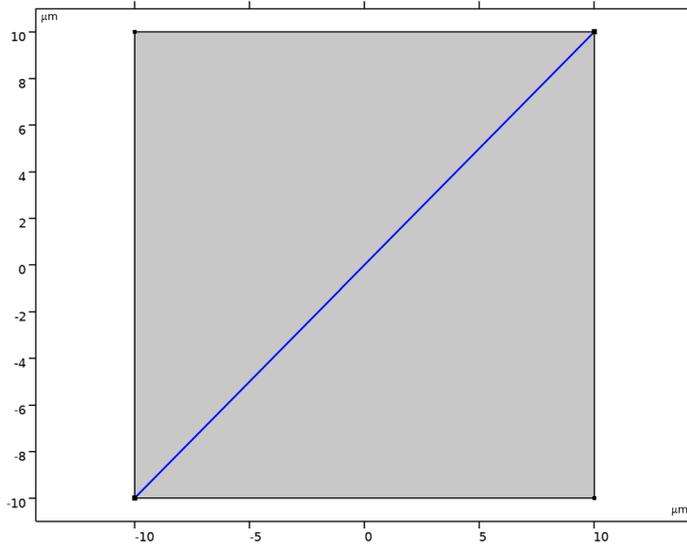
2 Select Boundaries 2 and 5 only.

3 In the **Settings** window for **Scattering Boundary Condition**, locate the **Scattering Boundary Condition** section.

4 From the **Input wave** list, choose **Second wave**.

Transition Boundary Condition 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Transition Boundary Condition**, locate the **Transition Boundary Condition** section.

4 From the **Electric displacement field model** list, choose **Relative permittivity**.

5 From the ε_r list, choose **User defined**. In the associated text field, type eps_Ag.

6 From the μ_r list, choose **User defined**. From the σ list, choose **User defined**. In the d text field, type 13[nm].

M E S H 1

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

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

**3** In the table, clear the **Use** checkbox for **Electromagnetic Waves, Beam Envelopes (ewbe)**.

M E S H 2

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

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

3 In the table, enter the following settings:

Contributor	Use
Electromagnetic Waves, Frequency Domain (ewfd)	
Electromagnetic Waves, Beam Envelopes (ewbe)	√

4 Locate the **Electromagnetic Waves, Beam Envelopes (ewbe)** section. From the **Mesh type** list, choose **Triangular mesh**.

5 In the h_{max} text field, type 2*lda0.

6 Click ▦ **Build All**.
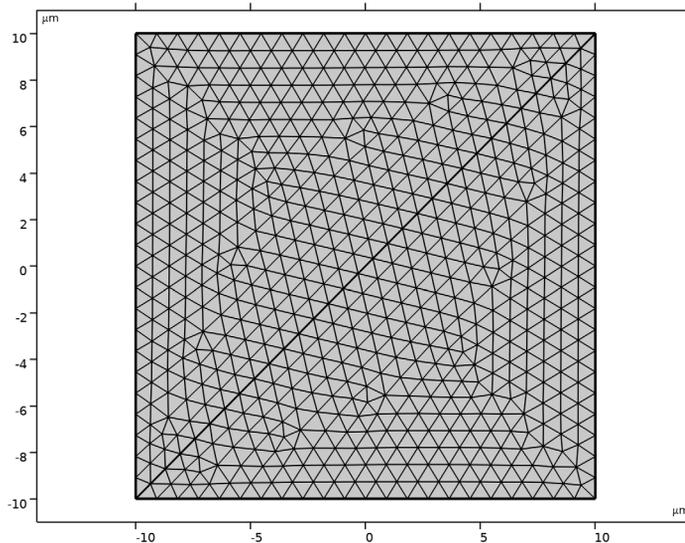
ADD STUDY

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 **General Studies** > **Frequency Domain**.

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

Physics	Solve
Electromagnetic Waves, Frequency Domain (ewfd)	
Electromagnetic Waves, Beam Envelopes (ewbe)	√

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

6 In the **Home** toolbar, click ⬡ **Add Study** to close the **Add Study** window.
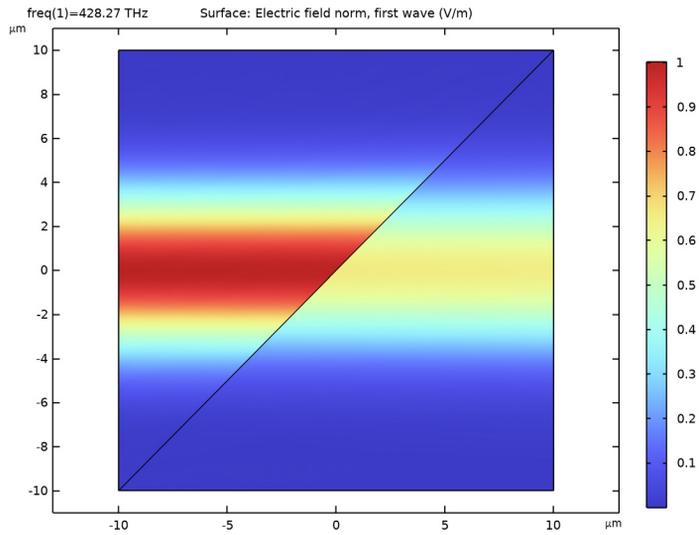
STUDY 2

Step 1: Frequency Domain

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

2 In the **Frequencies** text field, type f0.
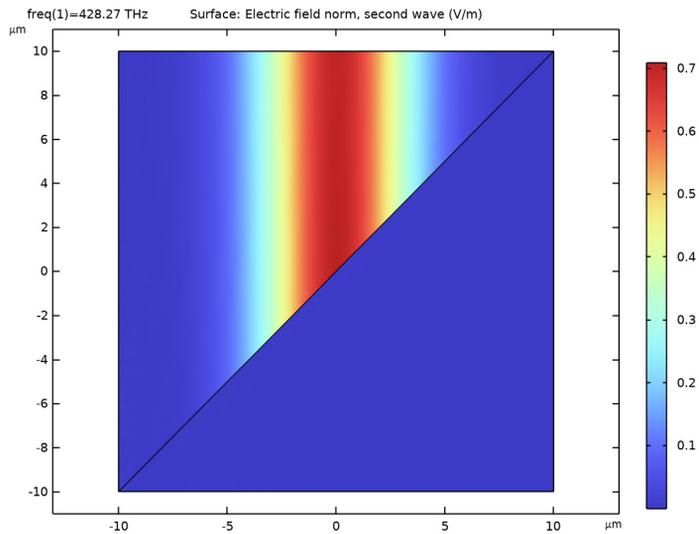
3 In the **Study** toolbar, click ▬ **Compute**.

Electric Field, First Wave (ewbe)

freq(1)=428.27 THz   Surface: Electric field norm, first wave (V/m)

Electric Field, Second Wave (ewbe)

In the **Model Builder** window, click **Electric Field, Second Wave (ewbe)**.

freq(1)=428.27 THz   Surface: Electric field norm, second wave (V/m)

Electric Field (ewbe)

1 Right-click **Electric Field, Second Wave (ewbe)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Field (ewbe)` in the **Label** text field.

Electric Field

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

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

3 In the **Expression** text field, type `ewbe.normE`.

4 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

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