Created in COMSOL Multiphysics 6.2

Modeling a Negative Refractive Index

Introduction

It is possible to engineer the structure of materials such that both the permittivity and permeability are negative. Such materials are realized by engineering a periodic structure with features comparable in scale to the wavelength. It is possible to model both the individual unit cells of such a material, as well as to model to properties of a bulk negative index material. This example demonstrates the correct way to model a metamaterial domain with bulk negative permittivity and permeability.

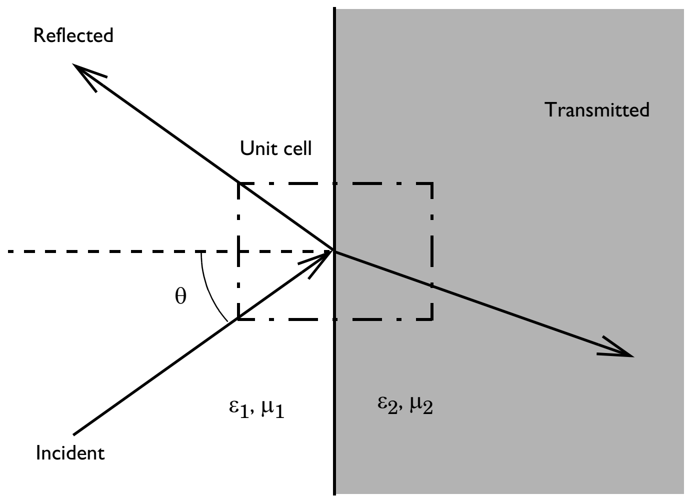


Figure 1: A plane wave of light incident upon an infinite half-space of material with negative permittivity and permeability.

Model Definition

A plane wave ($\lambda = 1$ m) traveling through vacuum ($\varepsilon_r = 1$, $\mu_r = 1$) is incident at an angle of 30° upon an infinite half-space of metamaterial with bulk negative permittivity and permeability ($\varepsilon_r = -1$, $\mu_r = -1$) as shown in Figure 1. The objective of the analysis is to observe the fields in the metamaterial.

Since the model space is infinite and invariant along the interface, it is possible to model a finite-sized unit cell around the interface, and use Floquet-periodic boundary conditions. The incident wave is modeled using a Port boundary condition, which both launches the incident wave and absorbs the reflected wave.

Two different ways are demonstrated for modeling boundary to the infinite metamaterial domain. In the first case, a Port boundary condition is used. This boundary condition requires computing the wave vector in the metamaterial, and manually adjusting the propagation constant at the boundary to account for the negative index. In the other case, a Perfectly Matched Layer (PML) is used to truncate the domain. The PML acts as an absorbing medium for all energy incident upon it, but must also be adjusted to account for the negative index. It is simpler to use, but increases the model size.

The transition between the two materials requires some special care. The natural boundary condition between domains of different material properties does not account for the change in direction of the flux. An additional degree of freedom has to be added to the model. This can be done by using the Transition Boundary Condition (TBC), which allows for a change in flux across the boundary. The TBC takes as input the material properties on one side of the domain, it does not matter which side. The thickness of the TBC should be approximately 1/1000 of the wavelength. If it is too small, it can introduce numerical difficulties. If it is too big, it alters the results significantly. The TBC only needs to be used in this way if the effective refractive index is similar in magnitude, but opposite in sign.

Results and Discussion

The electric field is plotted in Figure 2. The fields within the PML are not plotted, since these have no physical meaning. The wave is observed to switch direction at the interface between vacuum and the metamaterial.

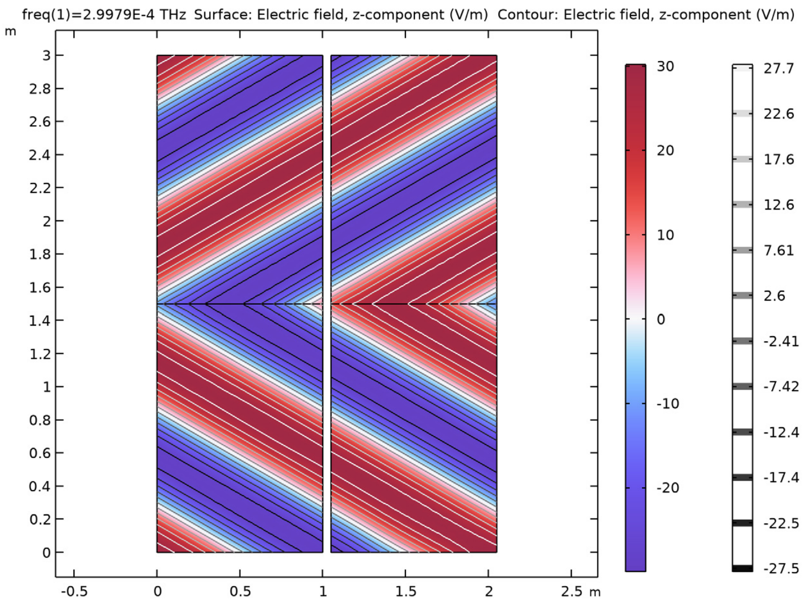


Figure 2: The electric fields in vacuum and in the metamaterial. The waves are observed to switch direction.

Application Library path: `Wave_Optics_Module/Gratings_and_Metamaterials/negative_refractive_index`

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

Load parameters from a file that are useful when setting up the physics and the materials.

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 `negative_refractive_index_parameters.txt`.

   Here, `c_const` is a predefined COMSOL constant for the speed of light in vacuum.

GEOMETRY 1

Two port model

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, type `Two port model` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Height** text field, type `3`.
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`1.5`

   The first rectangle consists of two rectangular domains representing the vacuum and metamaterial, respectively.

PML model

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, type `PML model` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Height** text field, type `4`.

4 Locate the **Position** section. In the **x** text field, type `1.05`.

5 In the **y** text field, type `-1`.

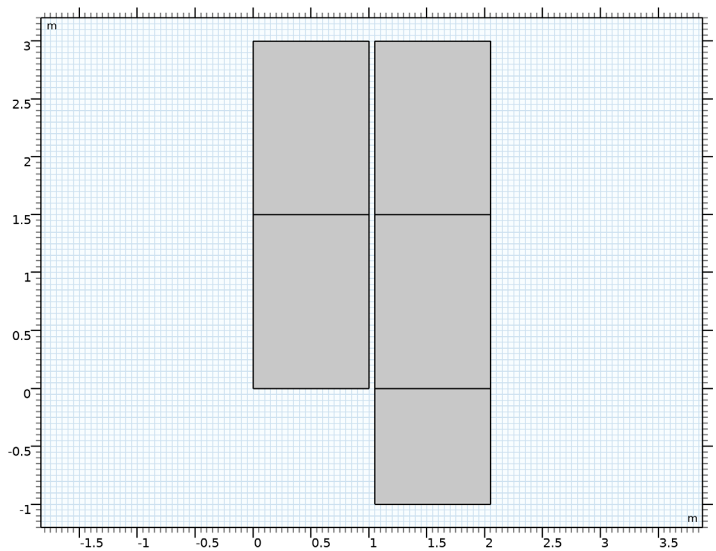
6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`1`
`Layer 2`	`1.5`

The second rectangle consists of three rectangular domains representing the vacuum, metamaterial, and PML.

7 Click **Build All Objects**.

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



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

Next, set up the physics. Use two different ways to model the infinite metamaterial domain.

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**.

Wave Equation, Electric 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (ewfd)** click **Wave Equation, Electric 1**.

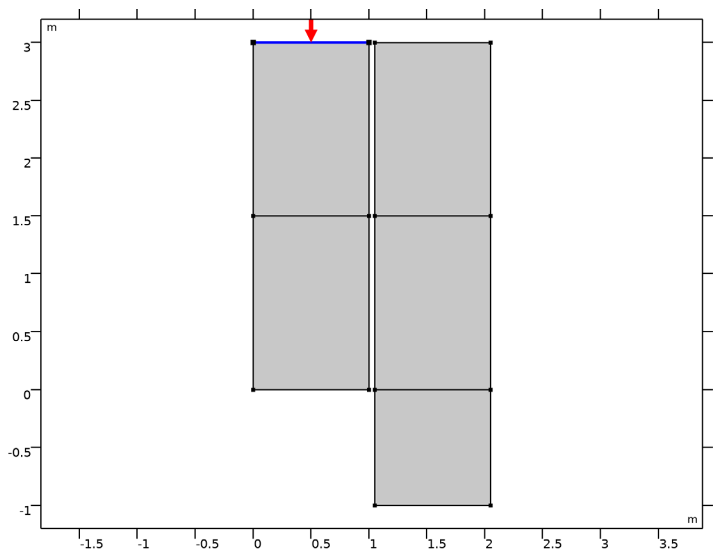
2 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

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

Port 1

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

2 Select Boundary 5 only.



For the first port, wave excitation is **on** by default.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Periodic**.

5 Locate the **Port Mode Settings** section. Specify the $\mathbf{E}_0$ vector as

0	x
0	y
1	z

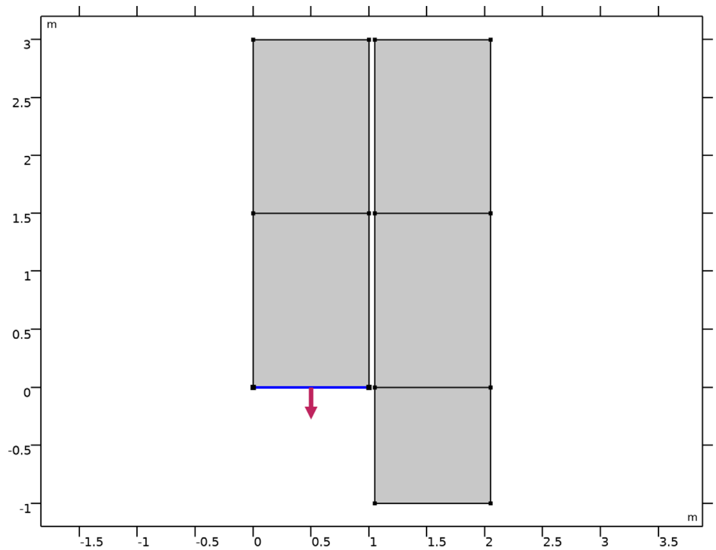
6 In the α text field, type `alpha`.

7 Locate the **Automatic Diffraction Order Calculation** section. In the n text field, type `n_a`.

Port 2

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

2 Select Boundary 2 only.



3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Periodic**.

5 Locate the **Port Mode Settings** section. Specify the $\mathbf{E}_0$ vector as

0	x
0	y
1	z

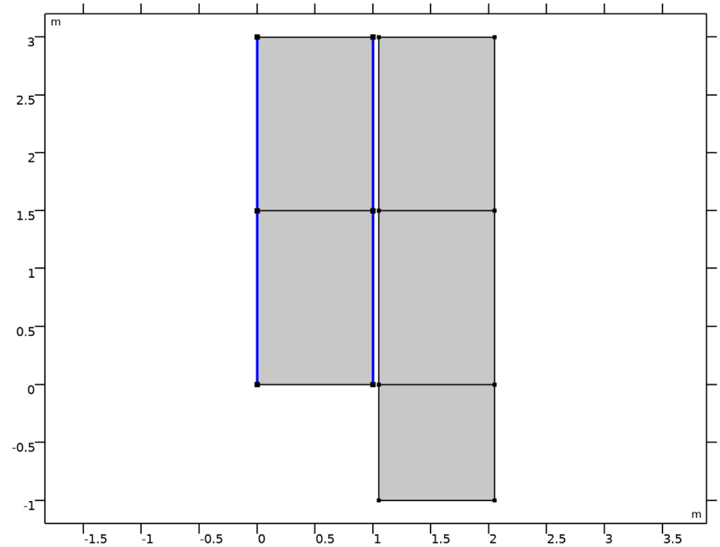
6 Locate the **Automatic Diffraction Order Calculation** section. In the n text field, type `n_b`.

The first method uses two ports.

Periodic Condition 1

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

2 Select Boundaries 1, 3, 6, and 7 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

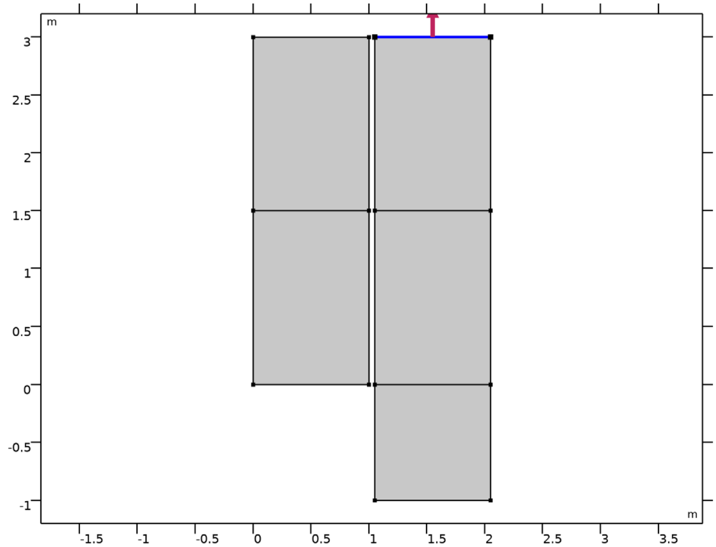
4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.

Port 3

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

2 Select Boundary 14 only.



3 In the **Settings** window for **Port**, locate the **Port Properties** section.

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

5 From the **Type of port** list, choose **Periodic**.

6 Locate the **Port Mode Settings** section. Specify the $\mathbf{E}_0$ vector as

0	x
0	y
1	z

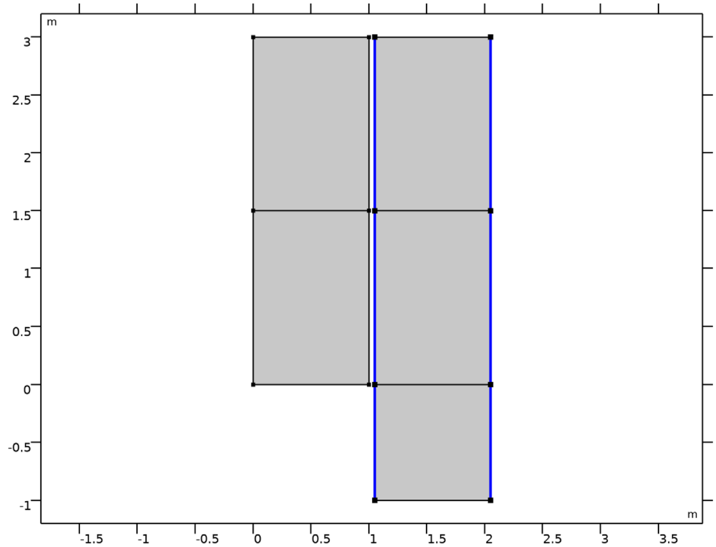
7 In the α text field, type `alpha`.

8 Locate the **Automatic Diffraction Order Calculation** section. In the *n* text field, type `n_a`.

Periodic Condition 2

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

2 Select Boundaries 8, 10, 12, and 15–17 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

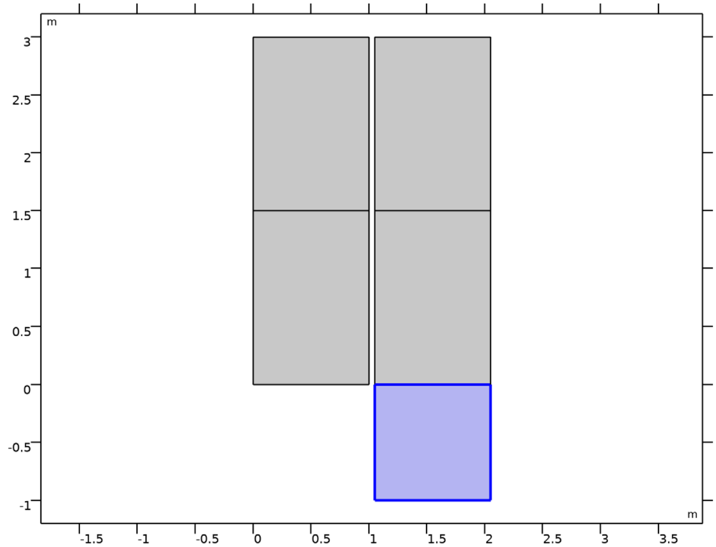
5 From the **k-vector for Floquet periodicity** list, choose **From periodic port**.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

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

2 Select Domain 3 only.



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

4 In the **PML scaling factor** text field, type -1.

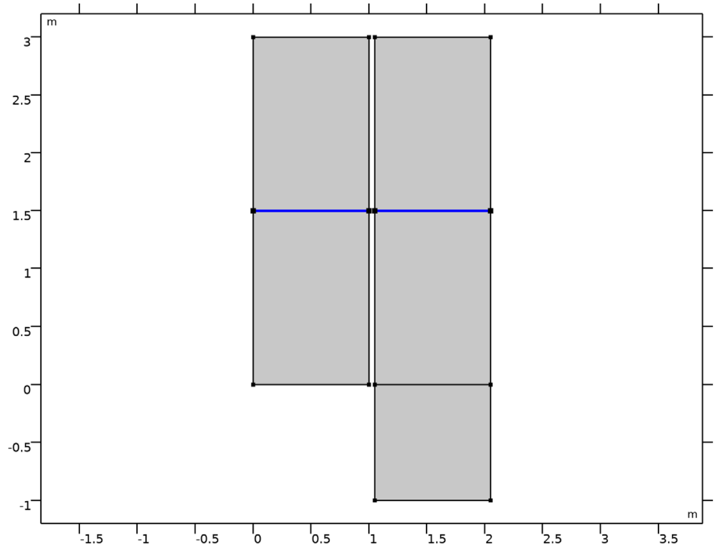
The second method uses one port and the PML.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

Transition Boundary Condition 1

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

2 Select Boundaries 4 and 13 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 μ_r list, choose **User defined**. In the associated text field, type `mu_a`.

6 From the ε_r list, choose **User defined**. In the associated text field, type `e_a`.

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

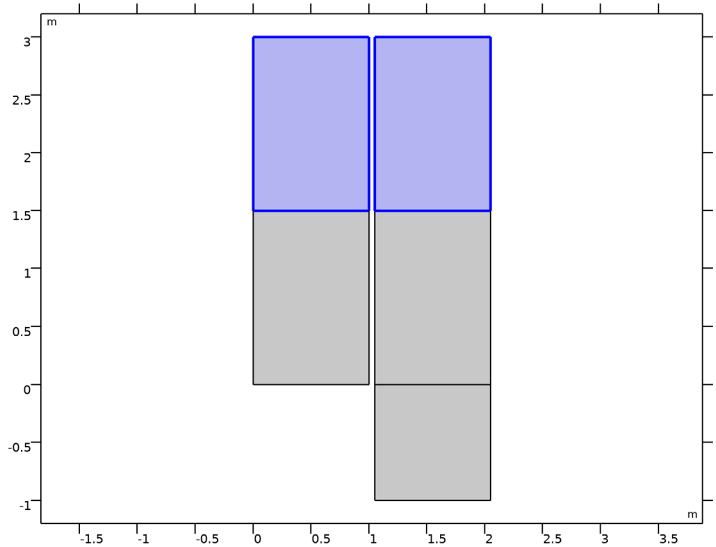
8 In the d text field, type `lda0/1000`.

MATERIALS

Material 1 (mat1)

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

2 Select Domains 2 and 5 only.



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

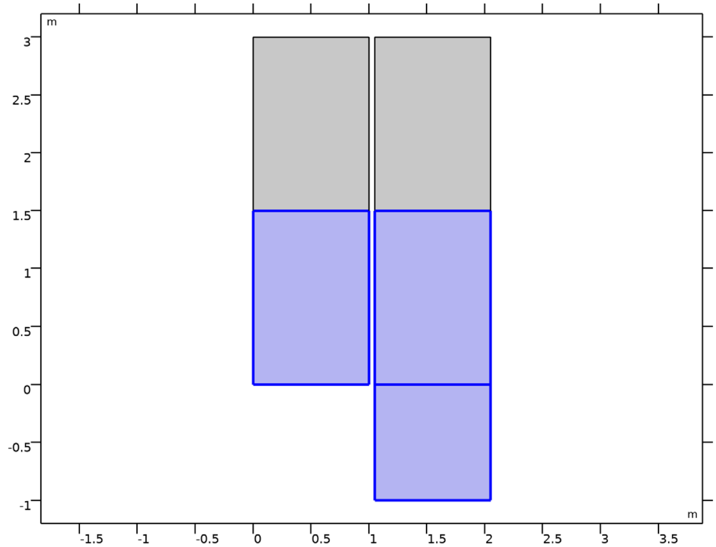
4 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	e_a	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	mu_a	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

Material 2 (mat2)

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

2 Select Domains 1, 3, and 4 only.



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

4 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	e_b	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	mu_b	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

STUDY 1

Step 1: Frequency Domain

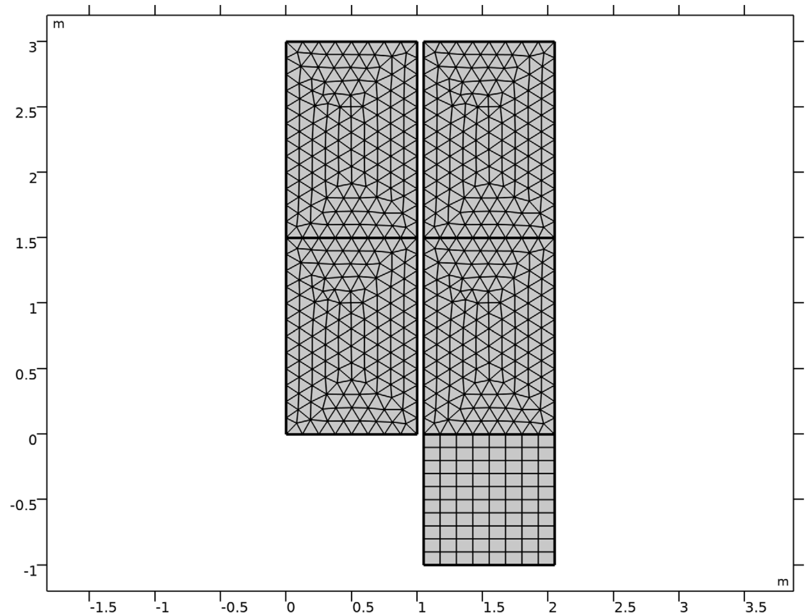
1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

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

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

MESH 1

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



STUDY 1

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

RESULTS

Study 1/Solution 1 (sol1)

The PML is not of interest for the result visualization; use only the air and metamaterial domains for this purpose.

Selection

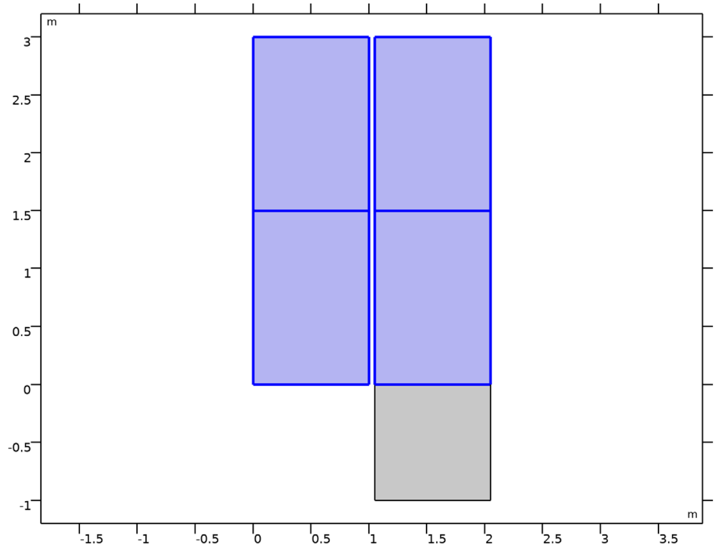
1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Study 1/Solution 1 (sol1)** and choose **Selection**.

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

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

5 Select Domains 1, 2, 4, and 5 only.



Electric Field (ewfd)

The default plot shows the norm of the electric field. Modify the plot to show the z component of the electric field, then add a contour plot.

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**, locate the **Expression** section.

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

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.

6 Click **OK**.

Contour 1

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

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

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

4 Locate the **Levels** section. In the **Total levels** text field, type `12`.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Linear>GrayPrint** in the tree.

7 Click **OK**.

8 Click the **Zoom Extents** button in the **Graphics** toolbar. Compare the resulting plot with that shown in Figure 2.