Created in COMSOL Multiphysics 5.6

Frequency Selective Surface, Periodic Complementary Split Ring Resonator

Introduction

Frequency selective surfaces (FSS) are periodic structures with a bandpass or a bandstop frequency response. This example shows that only signals around the center frequency can pass through the periodic complementary split ring resonator layer.

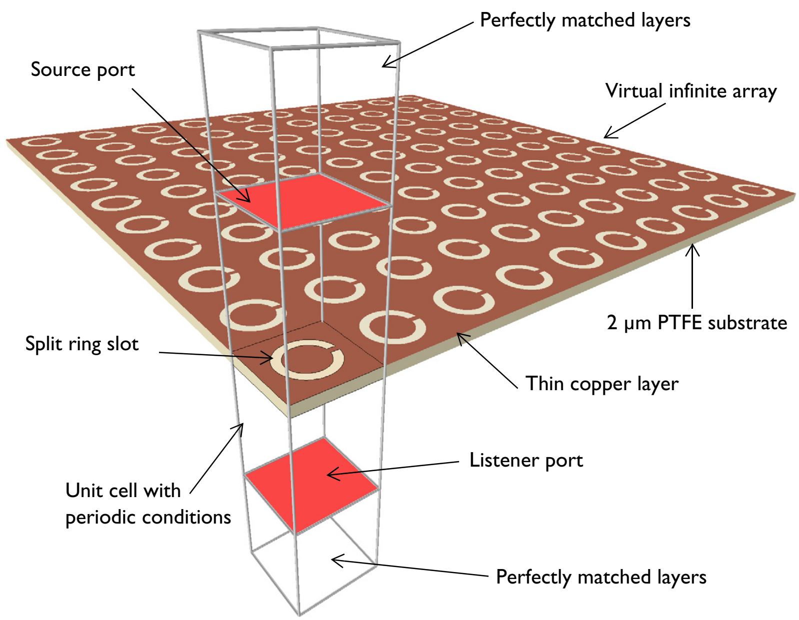


Figure 1: One unit cell of the complementary split ring resonator is modeled with periodic boundary conditions to simulate an infinite 2D array. Perfectly matched layers at the top and bottom of the unit cell absorb the excited and higher order modes.

Model Definition

A split ring slot is patterned on a geometrically thin copper layer that sits on a 2 µm PTFE substrate (Figure 1). The copper layer is much thicker than the skin depth in the simulated frequency range, so it is modeled as a perfect electric conductor (PEC). The rest of the simulation domain is filled with air.

Floquet-periodic boundary conditions are used on four sides of the unit cell to simulate the infinite 2D array. Perfectly matched layers (PMLs) on the top and bottom of the unit cell absorb the excited mode from the source port and any higher order modes generated by the periodic structure. The PMLs attenuate the wave as it propagates in the direction perpendicular to the PML boundary. Since the model is solved for a range of incident angles, the wavelength in the PMLs is set to $2\pi/|k_0\cos\theta|$. This accounts for the angular dependence of the normal component of the wave vector inside the PMLs.

Port boundary conditions are placed on the interior boundaries of the PMLs, adjacent to the air domains. The Port boundary conditions automatically determine the reflection and transmission characteristics in terms of S-parameters. The interior port boundaries with PML backing require the slit condition. The port orientation is specified to define the inward direction for the S-parameter calculation. Since higher order diffraction modes are not of particular interest in this example, the combination of Domain-backed type slit port and PMLs is used instead of adding a Diffraction order port for each diffraction order and polarization.

The periodic boundary condition requires identical surface meshes on paired boundaries. This is accomplished in two steps: first by creating a mesh on only one of the boundaries and then using the Copy Face operation for the mesh on the other boundary. This mesh configuration is automatically set when using the physics-controlled mesh as shown in the step-by-step instructions. If you are interested in seeing more details about the mesh, build the physics-controlled mesh once and then change the mesh sequence type to the user-controlled mesh in the mesh settings. Then you can inspect the generated mesh sequence.

The periodic conditions are split by the ports, having the slit conditions. A slit condition means that the dependent variable can have different values on the two sides of the boundary. To make sure that the periodic conditions couples the correct dependent variables on the two sides of the slit condition, multiple periodic conditions are defined—both in front of the port and behind the port.

Results and Discussion

The modified multislice default plot (Figure 2) shows the electric field norm on the complimentary split ring resonator. Strong fields are observed inside the slot. The S-parameter plot in Figure 3 shows that this periodic structure functions as a bandpass filter near 4.6 THz. In Figure 4, the S-parameters appear as a function of incident angle and show that the periodic structure is penetrable at 4.6 THz over the simulated range, except for grazing angles.

The resonance frequency of this periodic structure can be quickly evaluated as 4.59 THz using an Eigenfrequency study, which is not included in this example.

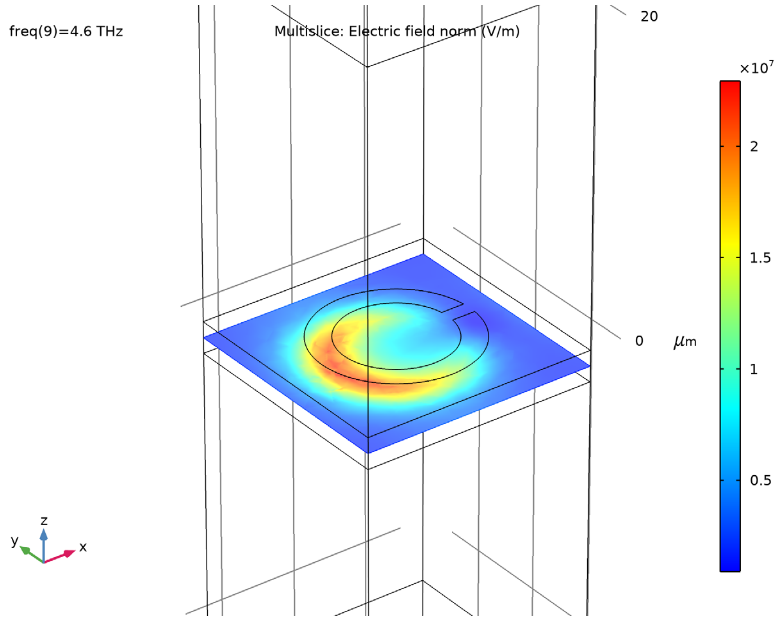


Figure 2: The fields are confined in the split ring slot.

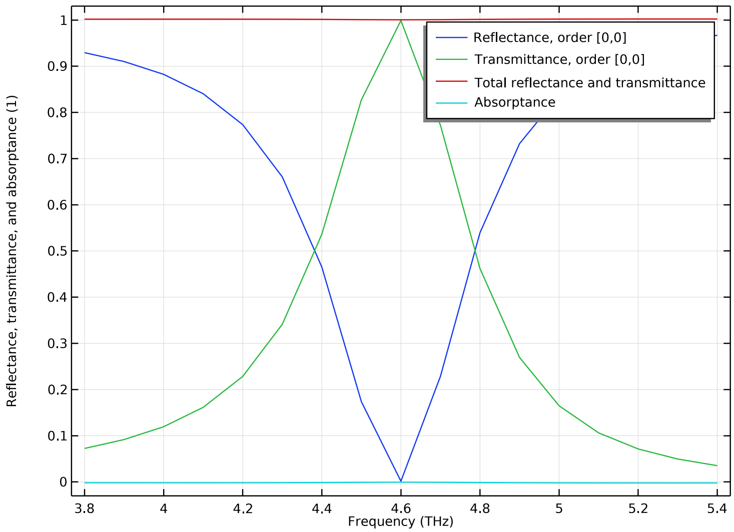


Figure 3: The S-parameter plot shows a bandpass resonance near 4.6 THz.

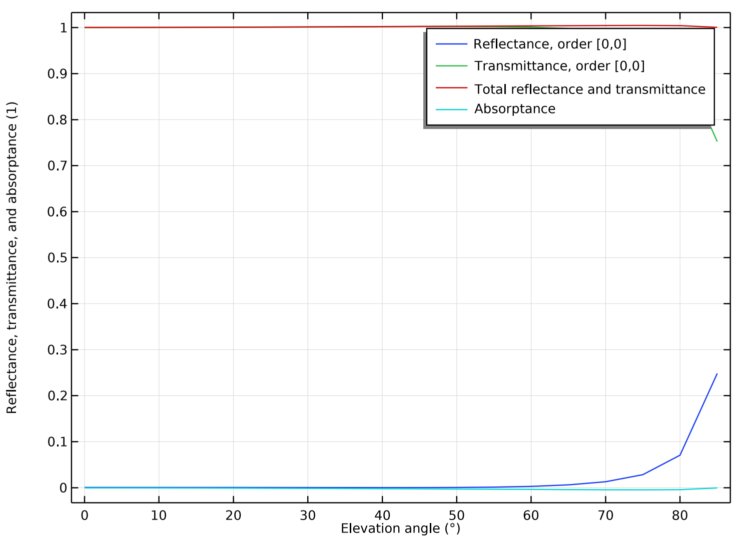


Figure 4: The S-parameter plot is shown as a function of incident angle.

Application Library path: `Wave_Optics_Module/Gratings_and_Metamaterials/frequency_selective_surface_csrr`

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 **3D**.

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**.

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 `range(3.8[THz],0.1[THz],5.4[THz])`.

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
theta	0[deg]	0 rad	Elevation angle

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**.

Block 1 (blk1)

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

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

3 In the **Width** text field, type 15.

4 In the **Depth** text field, type 15.

5 In the **Height** text field, type 45.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 Click **Build Selected**.

8 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

3 In the **Radius** text field, type 5.

4 Click **Build Selected**.

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

Work Plane 1 (wp1)>Circle 2 (c2)

1 In the **Work Plane** toolbar, click **Circle**.

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

3 In the **Radius** text field, type 3.5.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type 4.

4 Locate the **Position** section. From the **Base** list, choose **Center**.

5 In the **xw** text field, type 4.

Work Plane 1 (wp1)>Difference 1 (dif1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the objects **c2** and **r1** only.

6 Click **Build Selected**.

Block 2 (blk2)

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

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

3 In the **Width** text field, type 15.

4 In the **Depth** text field, type 15.

5 In the **Height** text field, type 2.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **z** text field, type -1.

Block 3 (blk3)

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

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

3 In the **Width** text field, type 15.

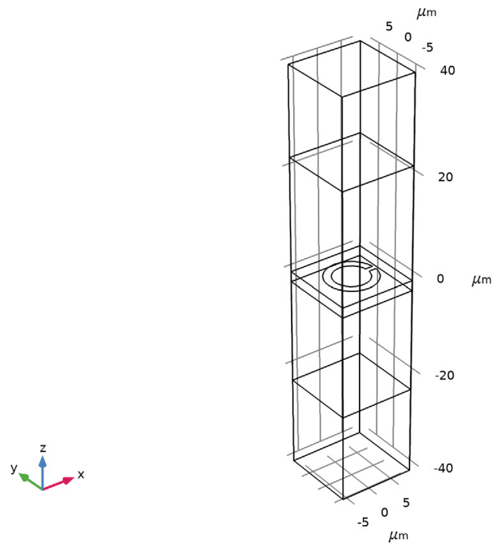
4 In the **Depth** text field, type 15.

5 In the **Height** text field, type 80.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 Click **Build All Objects**.

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



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

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**.

Perfect Electric Conductor 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.

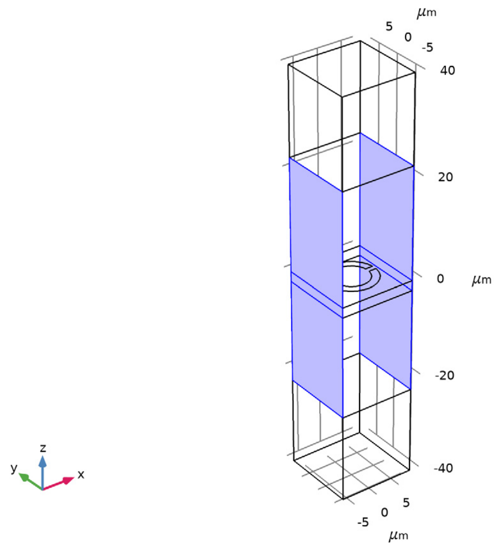
2 Select Boundary 12 only.

Use Floquet-periodic conditions on all side boundaries.

Periodic Condition 1

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

2 Select Boundaries 4, 7, 10, and 24–26 only.



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

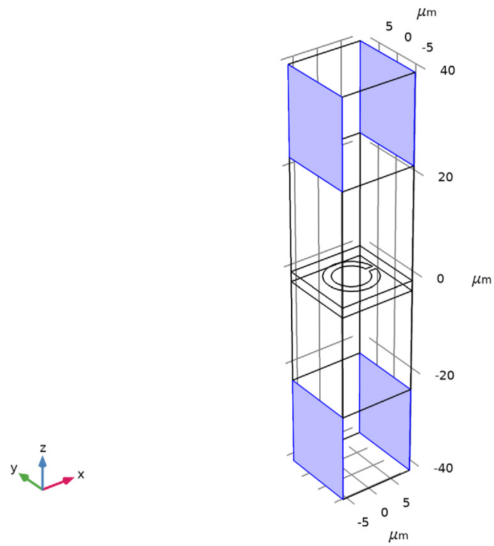
4 From the **Type of periodicity** list, choose **Floquet periodicity**.

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

Periodic Condition 2

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

2 Select Boundaries 1, 13, 23, and 27 only.



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

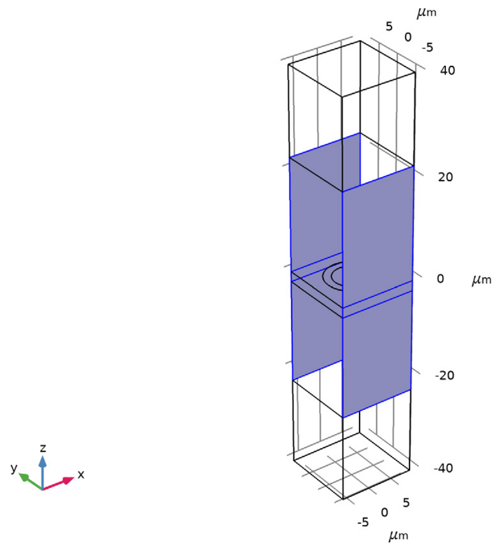
4 From the **Type of periodicity** list, choose **Floquet periodicity**.

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

Periodic Condition 3

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

2 Select Boundaries 5, 8, 11, and 18–20 only.



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

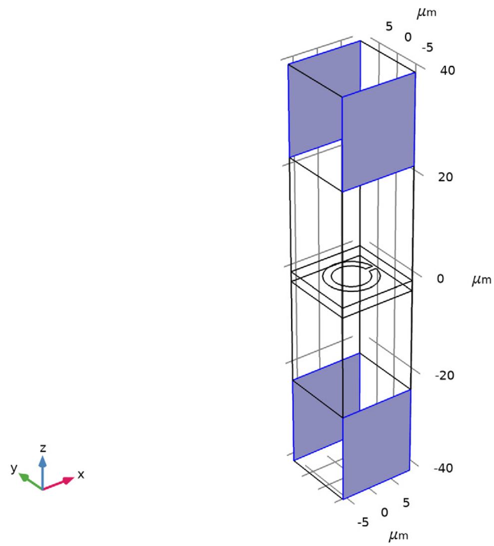
4 From the **Type of periodicity** list, choose **Floquet periodicity**.

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

Periodic Condition 4

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

2 Select Boundaries 2, 14, 17, and 21 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**.

The wave is excited from the port on the top.

Port 1

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

2 Select Boundary 15 only.

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

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

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

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**.

8 Locate the **Port Mode Settings** section. From the **Input quantity** list, choose **Magnetic field**.

9 Specify the $\mathbf{H}_0$ vector as

0	x
1	y
0	z

10 In the α_1 text field, type `theta`.

The maximum frequency in the setting window will be used only when **Compute Diffraction Order** button is clicked to generate Diffraction Order features handling higher order mode individually. In this model, PML absorbs all higher order modes, so this setting is ineffective.

Port 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Port**.
2. Select Boundary 6 only.
3. In the **Settings** window for **Port**, locate the **Port Properties** section.
4. From the **Type of port** list, choose **Periodic**.
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**.
8. Locate the **Port Mode Settings** section. From the **Input quantity** list, choose **Magnetic field**.
9. Specify the $\mathbf{H}_0$ vector as

0	x
1	y
0	z

Scattering Boundary Condition 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.
2. Select Boundary 3 only.

DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1. In the **Definitions** toolbar, click **Perfectly Matched Layer**.
2. Select Domain 5 only.

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

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

5 In the **Typical wavelength** text field, type `2*pi/abs(ewfd.k0*cos(theta))`.

Perfectly Matched Layer 2 (pml2)

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

2 Select Domain 1 only.

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

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

5 In the **Typical wavelength** text field, type `2*pi/abs(ewfd.k0*cos(theta))`.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Air**.

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

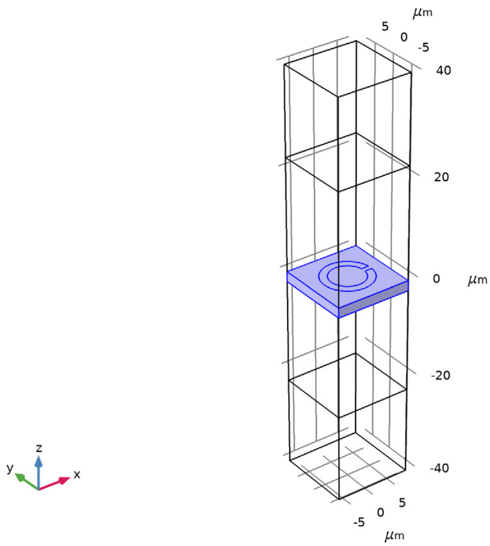
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material 2 (mat2)

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

2 Select Domain 3 only.



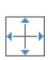
3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	2.1	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

MESH 1

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

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

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

4 From the **Element size** list, choose **Extremely fine**.

5 Click **Build All**.

DEFINITIONS

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.

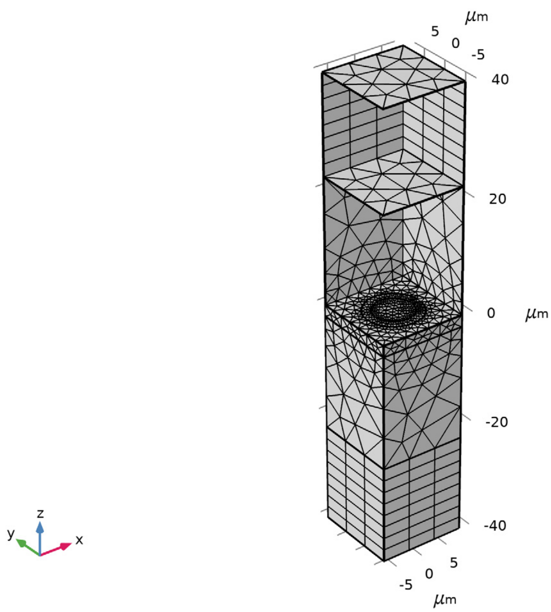
2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 10, 11, 13, and 14 only.

MESH 1

In the Model Builder window, click Mesh 1.



STUDY 1

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

RESULTS

Electric Field (ewfd)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Parameter value (freq (THz))** list, choose **4.6**.

Multislice 1

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

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

3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.

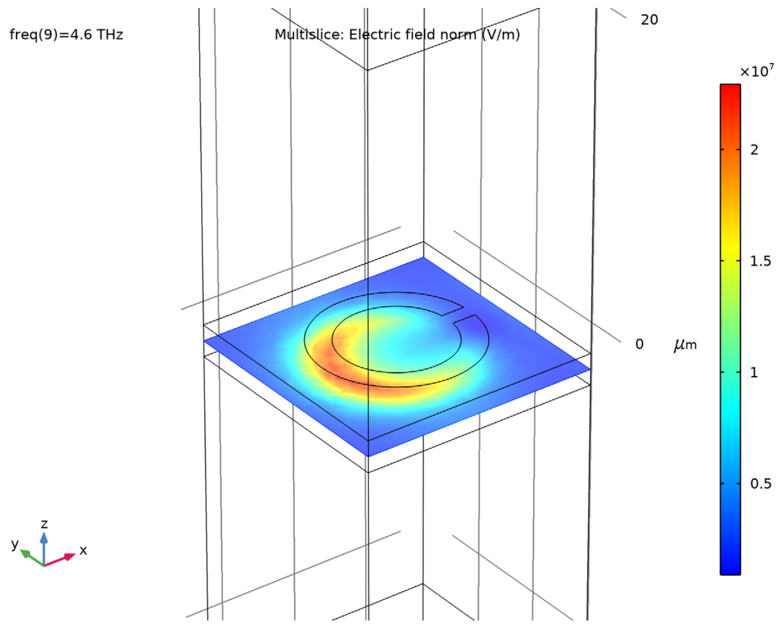
5 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `-1`.

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

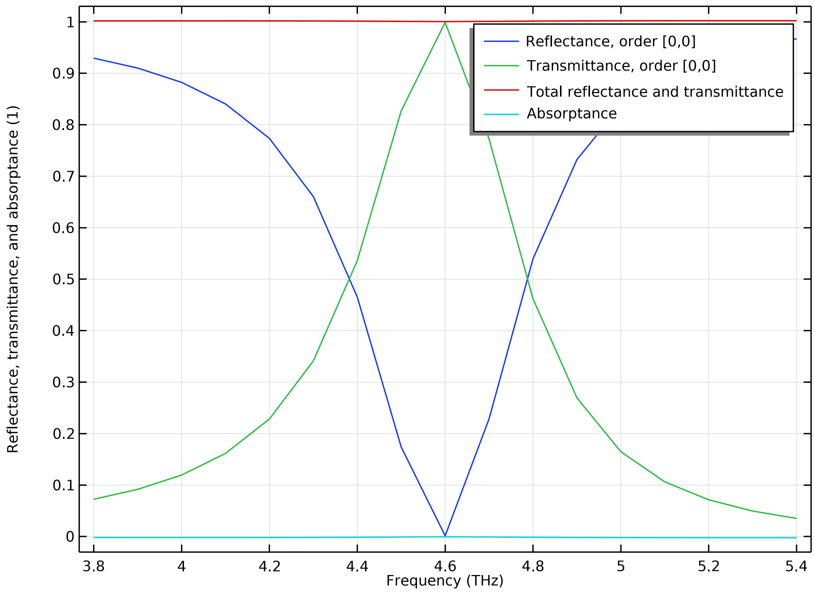
8 Click the **Zoom In** button in the **Graphics** toolbar.

9 Click the **Zoom In** button in the **Graphics** toolbar.



This reproduces Figure 2.

Reflectance, Transmittance, and Absorptance (ewfd)



Identify the resonant frequency of the periodic structure from the S-parameter plot Figure 3.

Next, evaluate the reflectivity and transmittivity performance of the model with different incident angles.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

STUDY 1

Solver Configurations

1 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

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

If you want to clear the Add Study window after adding, click Add Study again in the Home toolbar.

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

STUDY 2

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta (Elevation angle)	`range(0[deg],5[deg], 85[deg])`	`rad`

Step 1: Frequency Domain

1 In the **Model Builder** window, 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 `4.6[THz]`.

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

RESULTS

Multislice 1

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

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

3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.

5 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `-1`.

7 In the **Electric Field (ewfd) 1** toolbar, click **Plot**.

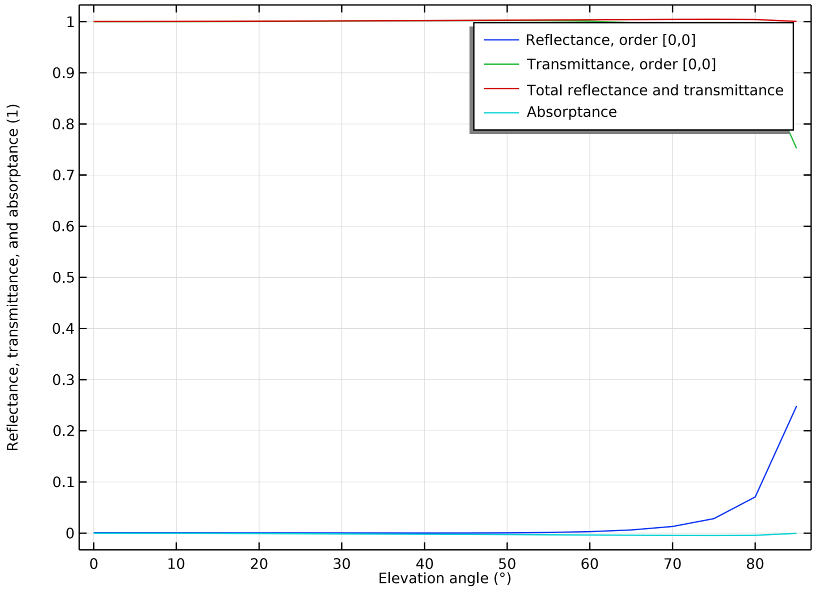
Global 1

1 In the **Model Builder** window, expand the **Results>Reflectance, Transmittance, and Absorptance (ewfd) 1** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.

3 From the **Unit** list, choose **°**.

4 In the **Reflectance, Transmittance, and Absorptance (ewfd) 1** toolbar, click **Plot**.



This is the S-parameter plot as a function of incident angle shown in Figure 4.

Analyze the same model with a much finer frequency resolution using **Adaptive Frequency Sweep** based on asymptotic waveform evaluation (AWE). When a device presents a slowly varying frequency response, the AWE provides a faster solution time when running the simulation on many frequency points. The following example with the AWE can be computed 50 times faster than regular Frequency Domain sweeps with a same finer frequency resolution.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

Port 1

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

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

3 Click **Create Selection**.

4 In the **Create Selection** dialog box, type `Port 1` in the **Selection name** text field.

5 Click **OK**.

Port 2

1 In the **Model Builder** window, click **Port 2**.
2 In the **Settings** window for **Port**, locate the **Boundary Selection** section.
3 Click **Create Selection**.
4 In the **Create Selection** dialog box, type `Port 2` in the **Selection name** text field.
5 Click **OK**.

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>Adaptive Frequency Sweep**.
4 Click **Add Study** in the window toolbar.
5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Adaptive Frequency Sweep

1 In the **Settings** window for **Adaptive Frequency Sweep**, locate the **Study Settings** section.
2 In the **Frequencies** text field, type `range(3.8[THz],0.01[THz],5.4[THz])`.

Use a ten times finer frequency resolution.

A slowly varying scalar value curve works well for AWE expressions. When **AWE expression type** is set to **Physics controlled** in the **Adaptive Frequency Sweep** study settings, `abs(comp1.ewfd.S21)` is used automatically for two-port devices.

Because such a fine frequency step generates a memory-intensive solution, the model file size will increase tremendously when it is saved. When only the frequency response of port related variables are of interest, it is not necessary to store all of the field solutions. By selecting the **Store fields in output** check box in the **Values of Dependent Variables** section, we can control the part of the model on which the computed solution is saved. We only add the selection containing these boundaries where the port variables are calculated. The port size is relatively small compared to the entire modeling domain, and the saved file size with the fine frequency step is more or less that of the regular discrete frequency sweep model when only the solutions on the port boundaries are stored.

3 Locate the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

4 Under **Selections**, click **Add**.

5 In the **Add** dialog box, in the **Selections** list, choose **Port 1** and **Port 2**.

6 Click **OK**.

It is necessary to include the port boundaries to calculate S-parameters. By choosing only the port boundaries for **Store fields in output** settings, it is possible to reduce the size of a model file a lot.

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

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Electric Field (ewfd) 2** node.

2 Right-click **Multislice 1** and choose **Delete**.

Surface 1

In the **Model Builder** window, right-click **Electric Field (ewfd) 2** and choose **Surface**.

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.

2 Select Boundaries 6 and 15 only.

3 In the **Electric Field (ewfd) 2** toolbar, click **Plot**.

Reflectance, Transmittance, and Absorptance (ewfd) 2

1 In the **Model Builder** window, click **Reflectance, Transmittance, and Absorptance (ewfd) 2**.

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

3 From the **Position** list, choose **Lower right**.

Global 1

1 In the **Model Builder** window, expand the **Reflectance, Transmittance, and Absorptance (ewfd) 2** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
ewfd.Rorder_0_0	1	S11 Adaptive Frequency Sweep
ewfd.Torder_0_0	1	S21 Adaptive Frequency Sweep

Global 2

1 Right-click **Results>Reflectance, Transmittance, and Absorptance (ewfd) 2>Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
ewfd.Rorder_0_0	1	S11 Regular Sweep
ewfd.Torder_0_0	1	S21 Regular Sweep

4 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 From the **Positioning** list, choose **In data points**.

8 In the **Reflectance, Transmittance, and Absorptance (ewfd) 2** toolbar, click **Plot**.