Model created in COMSOL Multiphysics 6.4

Embedded Scatterer on a Substrate

Introduction

A plane transverse electric (TE)–polarized electromagnetic wave is incident on a metallic sphere embedded on a substrate. In this electromagnetic scattering problem, the far-field variables are computed for a few different elevation angles of incidence.

Model Definition

Figure 1 shows the schematic of the geometry. One half of the spherical metallic scatterer is embedded within the dielectric substrate. The substrate is considered to occupy the entire $z<0$ half space, while the $z>0$ half space is air. The interface between the air and the substrate is planar and located in the *xy*–plane at $z=0$. A plane electromagnetic wave, with frequency 2.3 GHz, is incident at an elevation angle θ. The wave is plane–polarized with the electric field vector tangential to the interface between the air and the substrate.

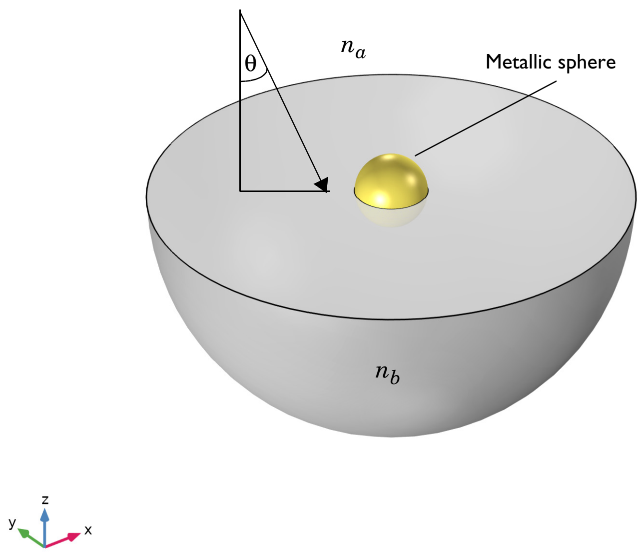


Figure 1: The modeled geometry. The gray domain represents the dielectric substrate, and the gold spherical domain represents the metallic scatterer. The incident wave propagates at an elevation angle θ.

The model uses $n_a = 1$ for air and $n_b = 1.7$ for the dielectric substrate. The metallic sphere is modeled as a perfect electric conductor. The model calculates the S–parameters without the scatterer and the far–field variables in the presence of the scatterer.

Results and Discussion

As is explained in Notes About the COMSOL Implementation, the model first computes a background field from the plane wave incident on the substrate, and then uses that to arrive at the total field with the scatterer present.

Figure 2 shows the instantaneous electric field norm of the total field due to the superposition of the incident plane wave and the field reflected from the substrate, for $\theta = \pi/4$. This total electric field is not affected by the scatterer. In the air, the total field is a superposition of the incident and the reflected plane waves. In the substrate, only the transmitted plane wave exists.

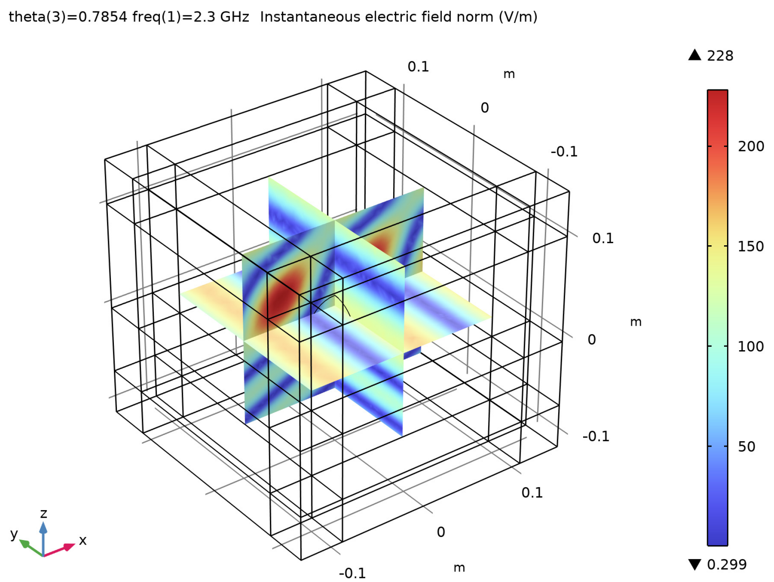


Figure 2: Instantaneous electric field norm of the total field without the scatterer for $\theta = \pi/4$.

The total electric field solved for without the scatterer is used as the background field to solve for the total electric field in the presence of the scatterer. Figure 3 shows the instantaneous norm of the total electric field for $\theta = \pi/4$, after it has been influenced both by the material interface and by the scatterer. Figure 4 shows the instantaneous norm of the background electric field for the same elevation angle of incidence, which is similar to Figure 2.

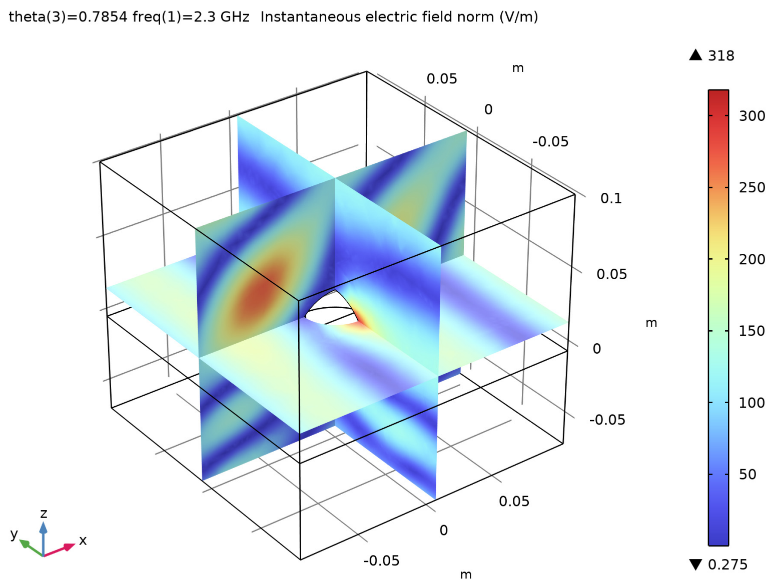


Figure 3: Instantaneous norm of the total electric field in the presence of the scatterer for $\theta = \pi/4$.

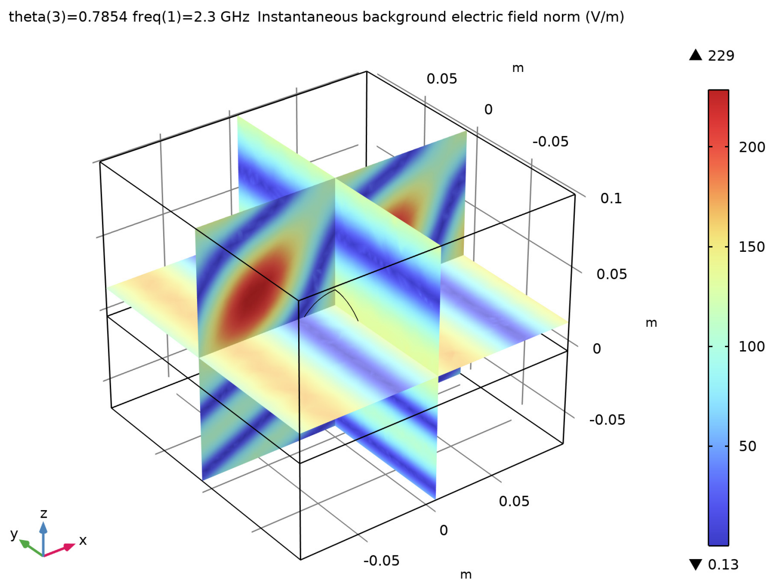


Figure 4: Instantaneous norm of the background electric field for $\theta = \pi/4$.

Figure 5 shows a polar plot of the far–field norm radiation pattern. Radiation patterns are plotted in plane of incidence, *xz*–plane, for different elevation angles of incidence, $\theta = 0$, $\theta = \pi/6$, and $\theta = \pi/4$.

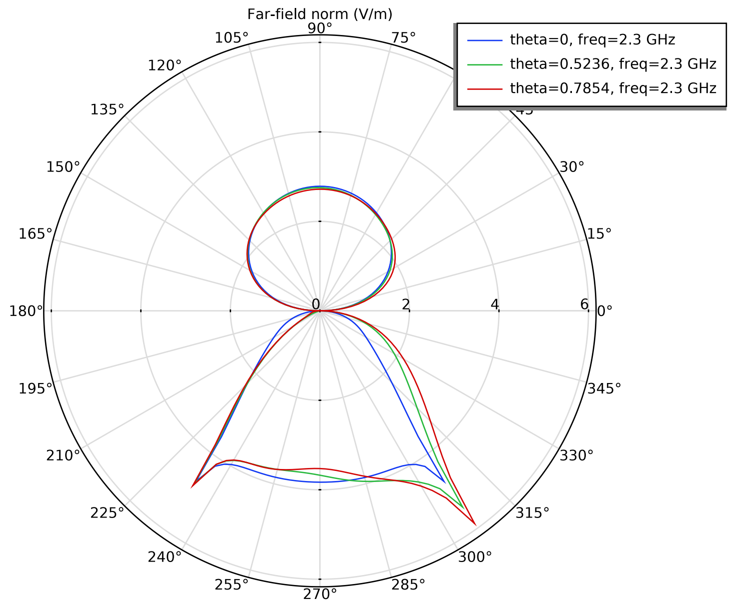


Figure 5: Radiation pattern of the far-field norm in the xz-plane.

Notes About the COMSOL Implementation

The Electromagnetic Waves, Frequency Domain interface provides as an option to solve for the scattered field, a perturbation to the total field caused by a local scatterer embedded on a substrate. If the scatterer is suspended in free space, the incident field that is launched — for instance, a Gaussian or plane wave — is simply entered as the background electric field. With the scatterer embedded on a substrate, the analytical expression for the background field becomes more complicated. It needs to be the correct superposition of an incident field, the field reflected from the substrate, and the transmitted field in the substrate.

A simple and general way to avoid having to derive and enter the analytical background field is to use a full field solution of the problem without the scatterer. To achieve this full field solution, the simulation is set up with two Port and two Periodic Condition nodes. One Port node defines the incident plane wave and allows for specular reflection. The other one absorbs the transmitted plane wave. The Periodic Conditions nodes allow the solution of one side of the geometry to be equal to the solution on the other side multiplied by a complex-valued phase factor. This effectively turns the model into a periodic cell of a geometry that extends indefinitely.

A second Electromagnetic Waves, Frequency Domain interface introduces the scatterer and a Perfect Electric Conductor node is used to define it. The total field solution from the first interface is used as the background electric field. This solves for the total field, including the field scattered by the scatterer, and allows calculating the far-field variables using a Far-Field Domain, Inhomogeneous node.

Application Library path: `RF_Module/Scattering_and_RCS/embedded_scatterer_on_substrate`

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 **Radio Frequency** > **Electromagnetic Waves, Frequency Domain (emw)**.

3 Click **Add**.

4 Click **Add**.

5 Click **Study**.

6 In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

7 Click **Done**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Update the Electric displacement field model settings to Refractive index. This enables you to specify the materials using a Refractive index material model.

Wave Equation, Electric 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain (emw)** 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 **Refractive index**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EMW2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain 2 (emw2)** 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 **Refractive index**.

GLOBAL DEFINITIONS

Define some parameters that are useful for setting up the geometry and the study.

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**
f0	2.3[GHz]	2.3E9 Hz	Frequency
lda0	c_const/f0	0.13034 m	Free space wavelength
w	1.5*lda0	0.19552 m	Width of physical geometry
h_air	lda0*0.8	0.10428 m	Air domain height
h_subs	lda0/2	0.065172 m	Substrate domain height
t_pml	lda0/3	0.043448 m	PML thickness
r_scatterer	3[cm]	0.03 m	Scatterer radius
na	1	1	Refractive index, air
nb	1.7	1.7	Refractive index, substrate
theta	0	0	Elevation angle of incidence in air

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

GEOMETRY 1

Draw the air domain with layers. The outermost layers represent the PMLs.

Air

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

2 In the **Settings** window for **Block**, type `Air` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w+2*t_pml`.

4 In the **Depth** text field, type `w+2*t_pml`.

5 In the **Height** text field, type `h_air+t_pml`.

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

7 In the **z** text field, type `(h_air+t_pml)/2`.

8 Click to expand the **Layers** section. In the table, enter the following settings:

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

Select the Left, Right, Front, Back, and Top checkboxes.

Clear the Bottom checkbox.

9 Find the **Layer position** subsection. Select the **Left** checkbox.

10 Select the **Right** checkbox.

11 Select the **Front** checkbox.

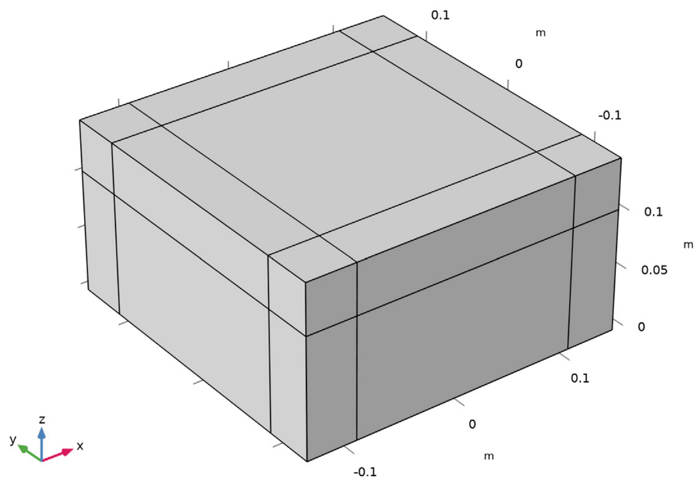
12 Select the **Back** checkbox.

13 Clear the **Bottom** checkbox.

14 Select the **Top** checkbox.

15 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

16 Click **Build Selected**.



Substrate

Now, draw the substrate domain with layers. The outermost layers represent the PMLs.

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

2 In the **Settings** window for **Block**, type `Substrate` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w+2*t_pml`.

4 In the **Depth** text field, type `w+2*t_pml`.

5 In the **Height** text field, type `h_subs+t_pml`.

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

7 In the **z** text field, type `-(h_subs+t_pml)/2`.

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

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

Select the Left, Right, Front, Back, and Bottom checkboxes. Leave the Top checkbox cleared.

9 Find the **Layer position** subsection. Select the **Left** checkbox.

10 Select the **Right** checkbox.

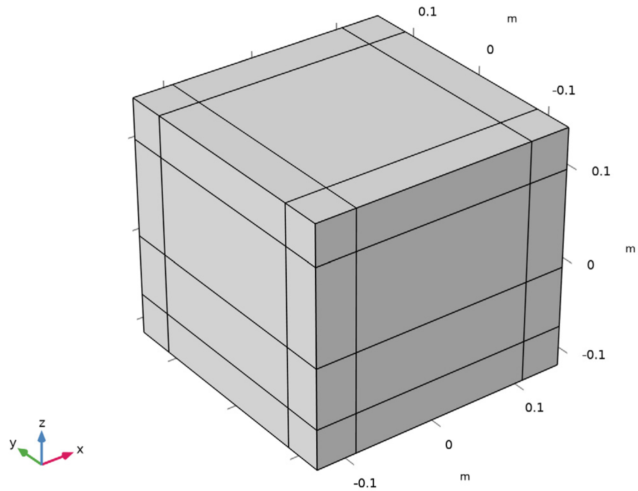
11 Select the **Front** checkbox.

12 Select the **Back** checkbox.

13 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

14 Click **Build Selected**.

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



Scatterer

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

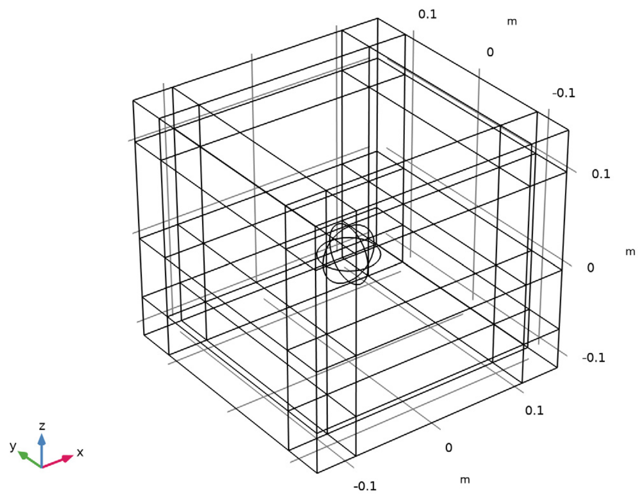
2 In the **Settings** window for **Sphere**, type `Scatterer` in the **Label** text field.

3 Locate the **Size** section. In the **Radius** text field, type `r_scatterer`.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

5 In the **Geometry** toolbar, click **Build All**.

6 Click the **Wireframe Rendering** button in the **Graphics** toolbar, for better visualization of the scatterer.



DEFINITIONS

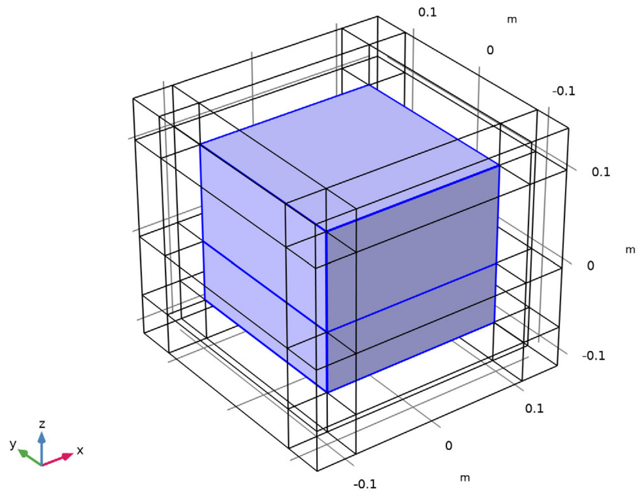
Define selections to separate between the part of your model where you will compute physical results and the part that will constitute the PML. For convenience, add a separate selection for the scatterer.

Physical Domains

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

2 In the **Settings** window for **Explicit**, type `Physical Domains` in the **Label** text field.

3 Select Domains 18, 19, 25, and 26 only.



PML Domains

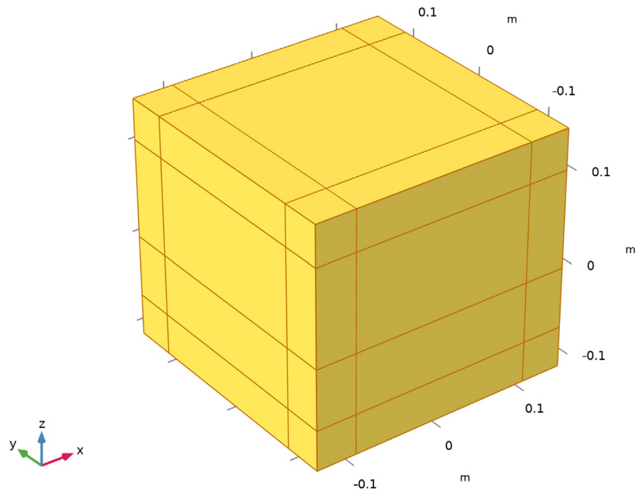
1 In the **Definitions** toolbar, click **Complement**.

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

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

4 In the **Add** dialog, select **Physical Domains** in the **Selections to invert** list.

5 Click **OK**.



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 Domains**.
4. Locate the **Scaling** section. From the **Physics** list, choose **Electromagnetic Waves, Frequency Domain 2 (emw2)**.

Variables 1

Only the second interface will be active in the PML domains. As this interface will use the electric field components from the first interface, define the electric fields to be 0 in the PML domains.

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **PML Domains**.

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

Name	Expression	Unit	Description
`emw.Ex`	0		
`emw.Ey`	0		
`emw.Ez`	0		

MATERIALS

Define materials for the air and the substrate domains. The scatterer will be modeled as a perfect electric conductor in the second interface.

Air

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 `Air` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

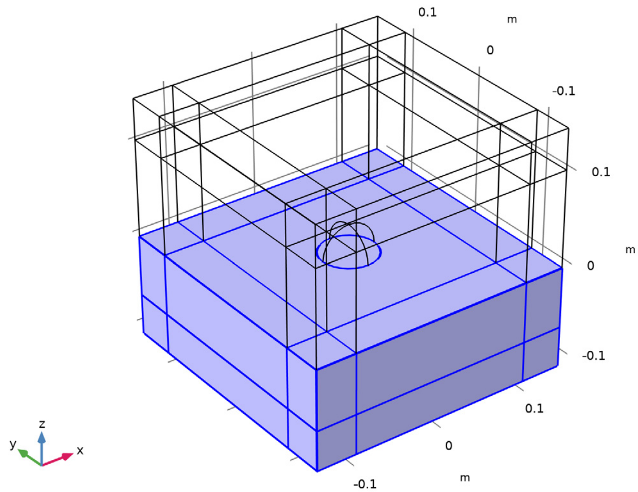
Property	Variable	Value	Unit	Property group
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	na	1	Refractive index

Substrate

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

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

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



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	nb	1	Refractive index

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

You are now ready to specify the physics. Start by setting up the first interface so that it computes the full wave solution to the plane wave falling in on the semi-infinite substrate.

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

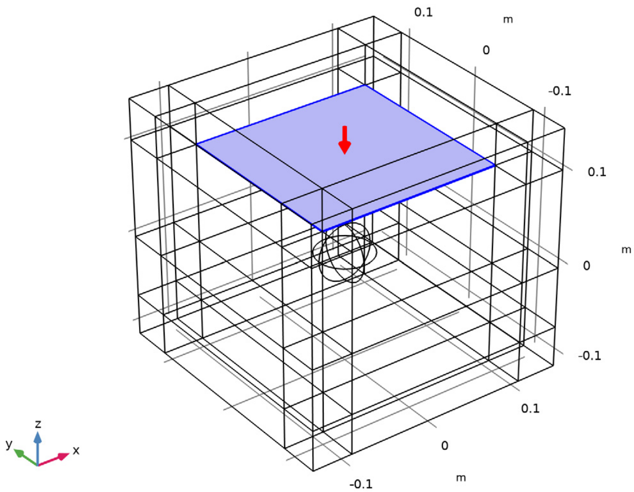
2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Physical Domains**, to exclude the PML domains.

Port 1

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

2 Select Boundary 68 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. From the **Polarization** list, choose **Linear polarization**.

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

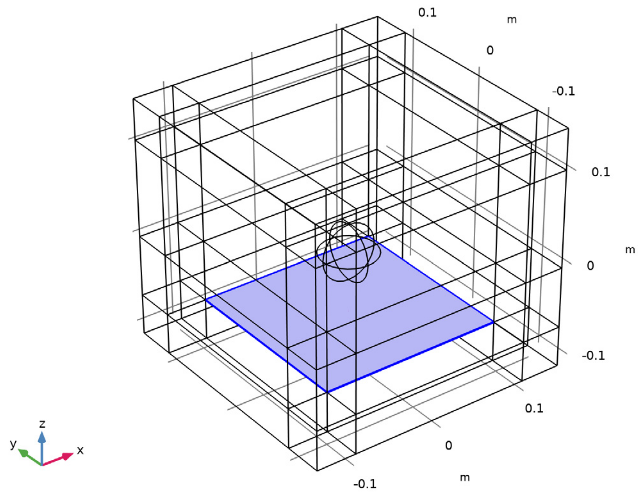
Port 2

1 Right-click **Port 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundary 62 only.

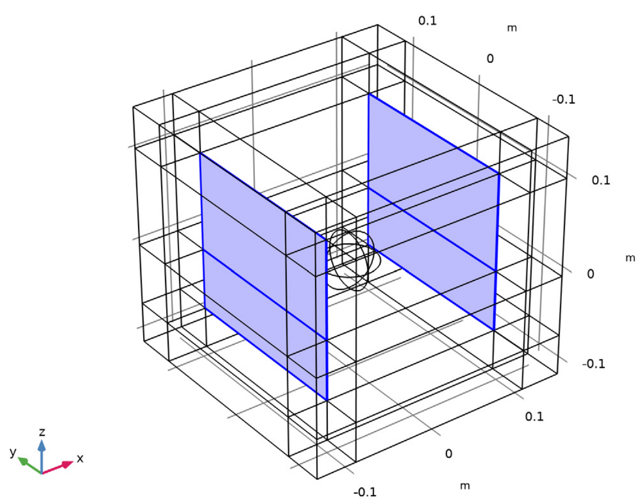


5 Locate the **Port Properties** section. From the **Wave excitation at this port** list, choose **Off**.

Periodic Condition 1

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

2 Select Boundaries 60, 63, 112, and 115 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**.

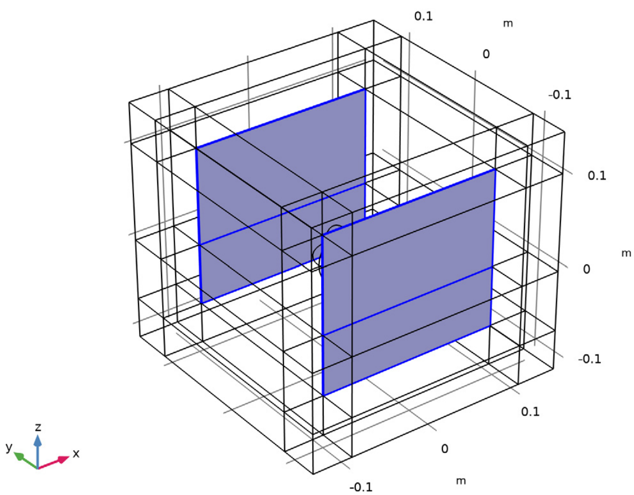
Periodic Condition 2

1 Right-click **Periodic Condition 1** and choose **Duplicate**.

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

3 Click  **Clear Selection**.

4 Select Boundaries 61, 64, 74, and 77 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EMW2)

Set up the second interface to compute how the plane wave solution from the first interface is affected by the scatterer.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain 2 (emw2)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Formulation** section.

3 From the list, choose **Scattered field**.

4 Specify the $\mathbf{E}_\mathrm{b}$ vector as

`emw.Ex`	x
`emw.Ey`	y
`emw.Ez`	z

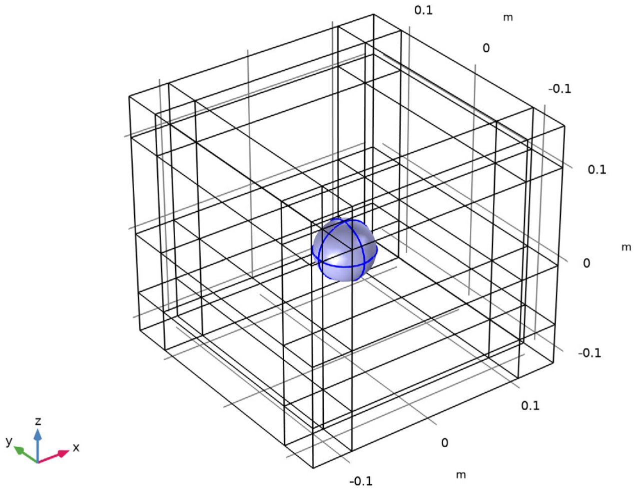
Perfect Electric Conductor 2

Add **Perfect Electric Conductor** node to define the scatterer.

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

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Domain Selection** section.

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



Far-Field Domain, Inhomogeneous 1

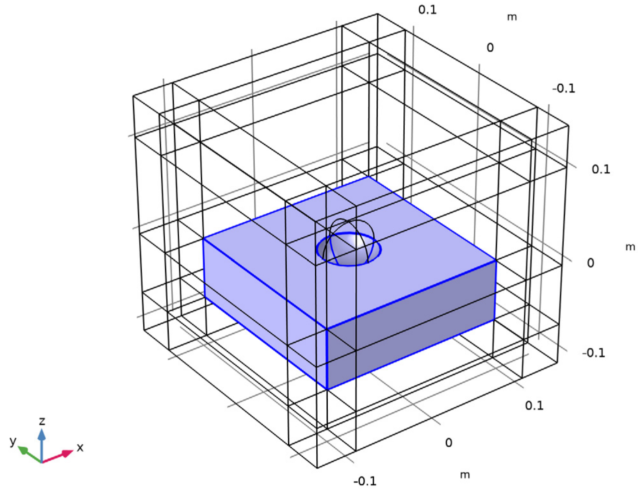
Add **Far-Field Domain, Inhomogeneous** node to calculate the far-field variables.

In the **Physics** toolbar, click **Domains** and choose **Far-Field Domain, Inhomogeneous**.

Substrate 1

1 In the **Model Builder** window, expand the **Far-Field Domain, Inhomogeneous 1** node, then click **Substrate 1**.

2 Select Domain 18 only, to select the substrate domain. The superstrate selection is set to all domains by default.



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`.

4 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Electromagnetic Waves, Frequency Domain 2 (emw2)**.

Step 2: Frequency Domain 1

1 Right-click **Study 1 > Step 1: Frequency Domain** and choose **Duplicate**.

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Electromagnetic Waves, Frequency Domain (emw)**.

4 In the **Solve for** column of the table, under **Component 1 (comp1)**, select the checkbox for **Electromagnetic Waves, Frequency Domain 2 (emw2)**.

Set up the solver for a few elevation angles of incidence. Because the second physics interface depends on the first one but not vice versa, the model can be solved sequentially.

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 of incidence in air)	`0 pi/6 pi/4`	

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

RESULTS

Multislice 1

Create a plot of the instantaneous norm of the total electric field without the scatterer to reproduce Figure 2.

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

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

3 In the **Expression** text field, type `emw.normEi`.

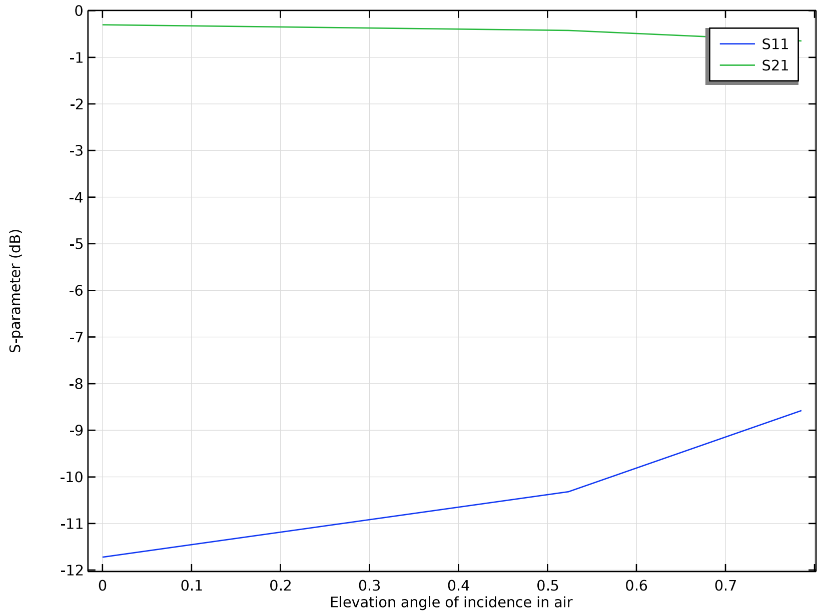
4 In the **Electric Field (emw)** toolbar, click **Plot**.

You have now plotted the instantaneous norm of the electric field from the first interface, for the $\theta = \pi/4$ solution. You can look at the different solutions using the **Parameter Value** list.

S-Parameter (emw)

This default plot shows the S11 and S21 parameters versus the elevation angle of incidence.

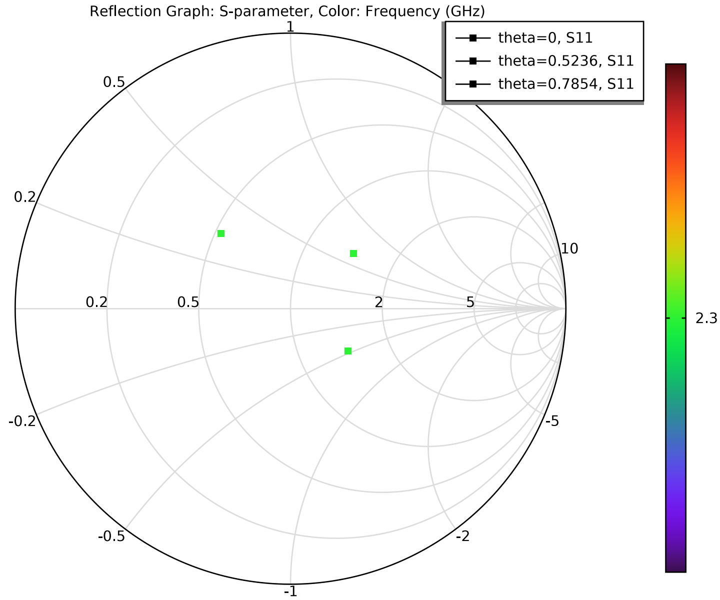
In the **Model Builder** window, under **Results** click **S-Parameter (emw)**.



Smith Plot (emw)

This default plot shows the Smith plot to visualize the S11 parameter.

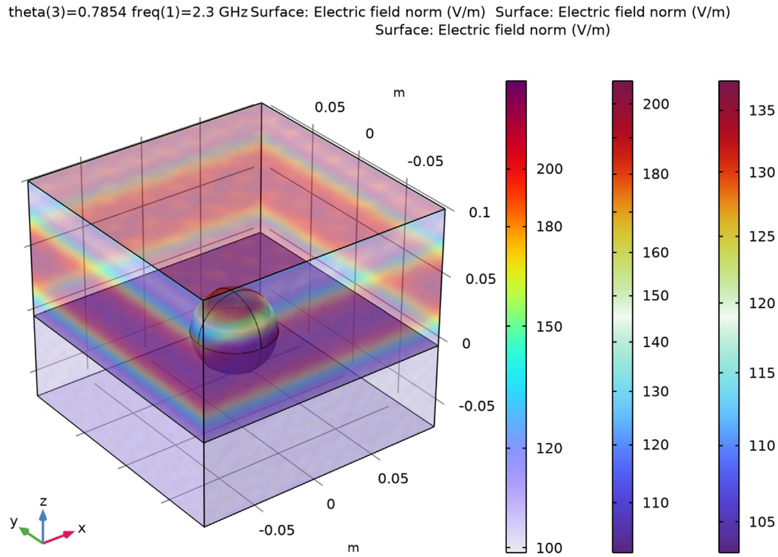
In the **Model Builder** window, click **Smith Plot (emw)**.



Electric Field, Logarithmic (emw)

This default plot shows the norm of the total electric field in logarithmic scale from the first interface, for the $\theta = \pi/4$ solution.

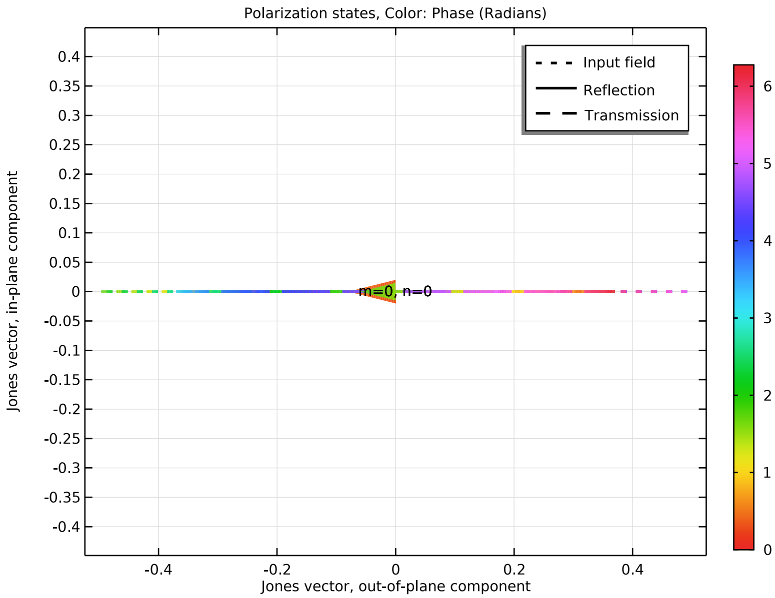
In the **Model Builder** window, click **Electric Field, Logarithmic (emw)**.



Polarization Plot (emw)

This default plot shows the polarization state of the plane wave.

In the **Model Builder** window, click **Polarization Plot (emw)**.



Electric Field (emw2)

Create a plot of the instantaneous norm of the total field in the presence of the scatterer to reproduce Figure 3, for the $\theta = \pi/4$ solution.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Field (emw2)** node, then click **Multislice 1**.

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

3 In the **Expression** text field, type `emw2.normEi`.

4 In the **Electric Field (emw2)** toolbar, click **Plot**.

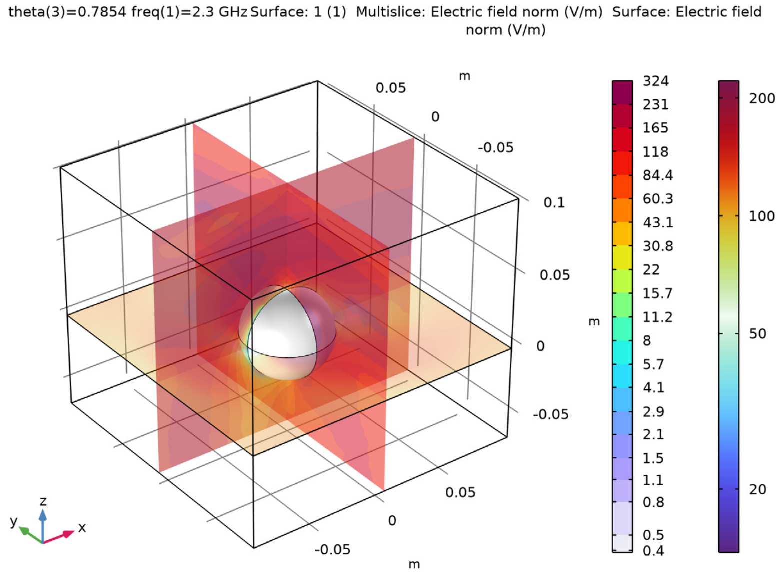
Electric Field, Background (emw2)

This default plot shows the instantaneous norm of the background electric field as is shown in Figure 4, for the $\theta = \pi/4$ solution.

Electric Field, Logarithmic (emw2)

This default plot shows the norm of the total electric field in logarithmic scale from the second interface, for the $\theta = \pi/4$ solution.

In the **Model Builder** window, click **Electric Field, Logarithmic (emw2)**.



2D Far Field (ffi1)

Create a plot of the radiation pattern in the plane of incidence, *xz*-plane, to reproduce Figure 5.

Radiation Pattern 1

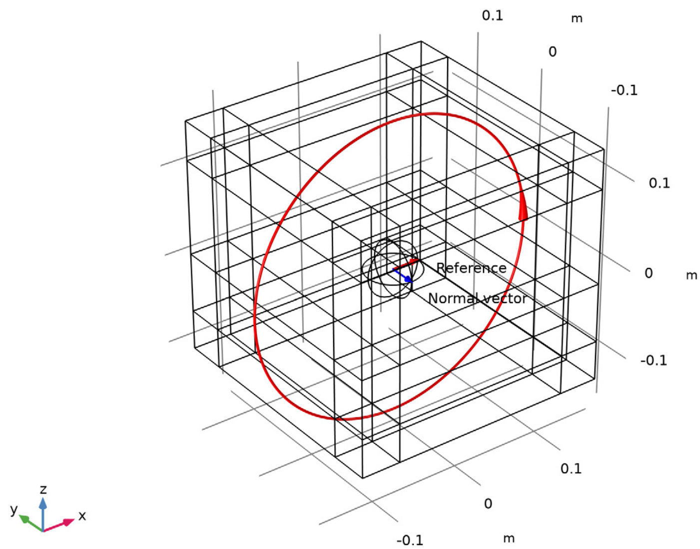
1 In the **Model Builder** window, expand the **Results** > **2D Far Field (ffi1)** node, then click **Radiation Pattern 1**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

3 Find the **Normal vector** subsection. In the **y** text field, type -1.

4 In the **z** text field, type 0.

5 Click **Preview Evaluation Plane**.



6 Click to expand the **Legends** section. Select the **Show legends** checkbox.

7 In the **2D Far Field (ffi1)** toolbar, click **Plot**, to reproduce Figure 5.