Model created in COMSOL Multiphysics 6.4

Scatterer on Substrate

Introduction

A plane TE-polarized electromagnetic wave is incident on a gold nanoparticle on a dielectric substrate. The absorption and scattering cross sections of the particle, and the far-field radiation pattern are computed for a few different polar and azimuthal angles of incidence.

Model Definition

Figure 1 shows the geometry, with the substrate considered to occupy the entire $z < 0$ half-space. A plane electromagnetic wave, with a 500 nm wavelength, is incident at a polar angle θ and an azimuthal angle ϕ. The wave is plane-polarized with the electric field vector tangential to the surface of the substrate.

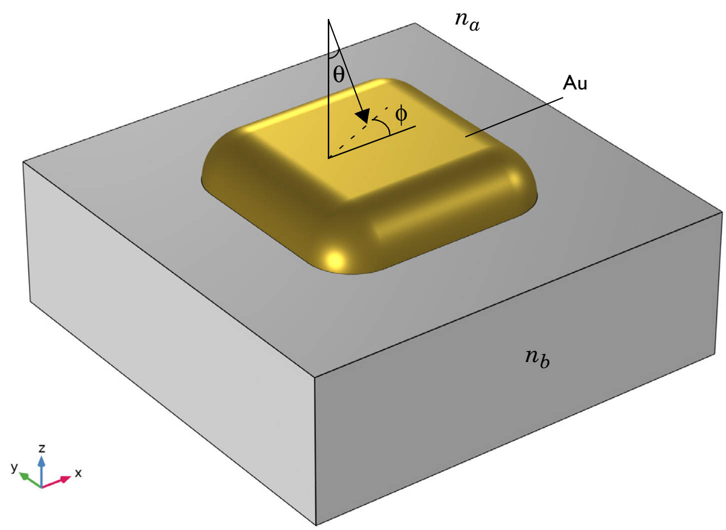


Figure 1: The modeled geometry. The gray domain represents the dielectric substrate. The electric field vector of the incident wave points in the ϕ direction, orthogonal to the plane of incidence.

The model uses $n_a = 1$ for air and $n_b = 1.5$ for the dielectric substrate. The scattering nanoparticle is made of gold. The refractive index is taken from the Optical Material Library.

The model computes the scattering, absorption, and extinction cross sections of the particle on the substrate. The scattering cross section is defined as

$$\sigma_{sc} = \frac{1}{I_0}\iint(\mathbf{n} \cdot \mathbf{S}_{sc})dS$$

Here, $\mathbf{n}$ is the normal vector pointing outward from the scatterer, $\mathbf{S}_{sc}$ is the scattered intensity (Poynting) vector, and I_0 is the incident intensity. The integral is taken over the closed surface of the scatterer. The absorption cross section equals

$$\sigma_{abs} = \frac{1}{I_0}\iiint Q dV$$

where Q is the power loss density in the particle and the integral is taken over its volume. The extinction cross section is simply the sum of the two others:

$$\sigma_{ext} = \sigma_{sc} + \sigma_{abs}$$

The model also calculates the far-field variables.

Results and Discussion

As 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 nanoparticle present.

Figure 2 and Figure 3 show the y-component and the norm of the electric background field, not yet affected by the nanoparticle, for the $\phi = \pi/4$, $\theta = \pi/6$ solution. In the air, this field is a superposition of the incident and reflected plane waves. In the substrate, only a transmitted plane wave exists.

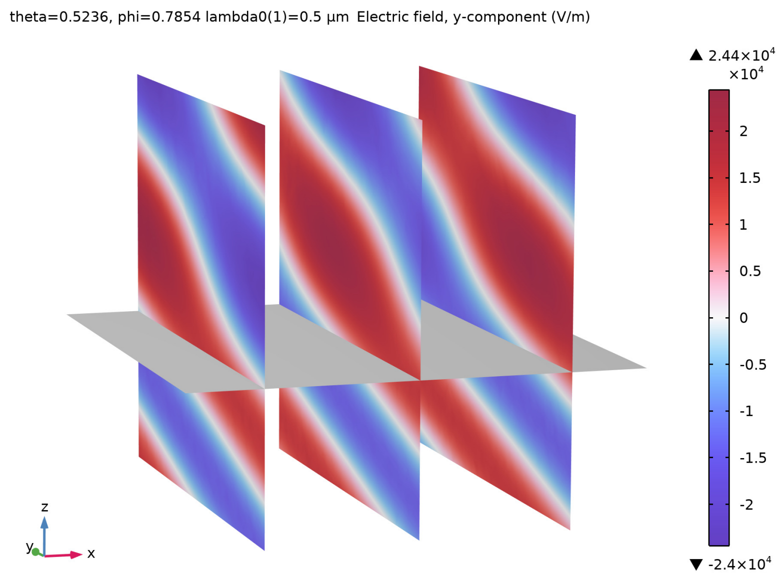


Figure 2: Background electric field, y-component for $\phi = \pi/4$, $\theta = \pi/6$, *on three slices parallel with the* yz*-plane.*

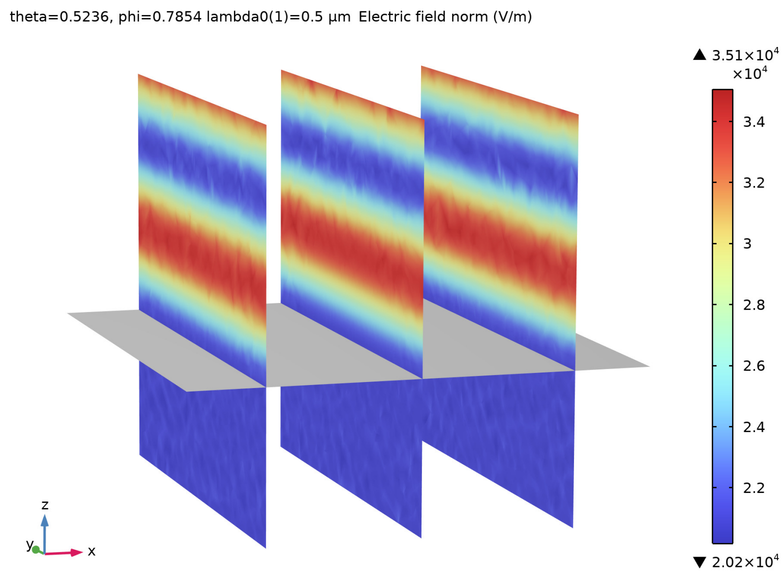


Figure 3: Background electric field norm, for $\phi = \pi/4$, $\theta = \pi/6$.

Figure 4 and Figure 4 show the y-component and norm of the total electric field for the same angles of incidence, after it has been influenced both by the material interface and by the nanoparticle.

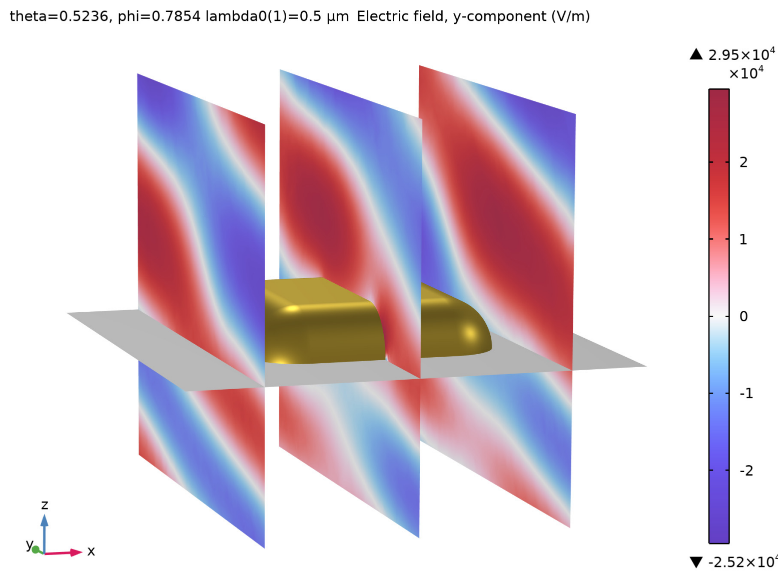


Figure 4: Slice plot of the y-component of the total electric field for $\phi = \pi/4$, $\theta = \pi/6$.

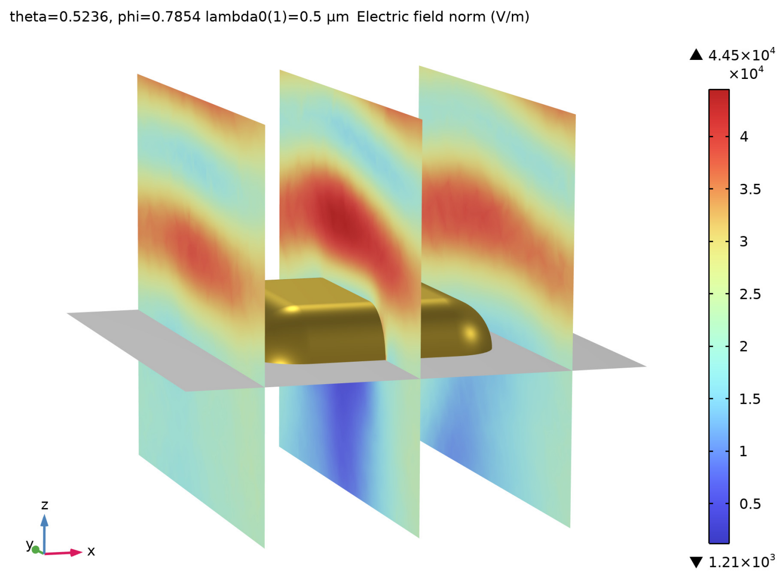


Figure 5: Slice plot of the total electric field norm for $\phi = \pi/4$, $\theta = \pi/6$.

In Figure 6, the power loss density is shown in a horizontal slice through the nanoparticle. No apparent resonance is present and most of the losses take place near the surface of the particle.

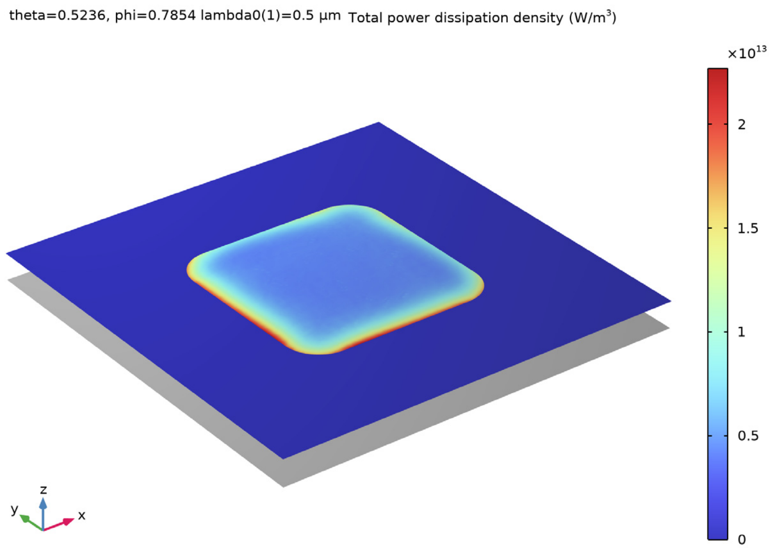


Figure 6: Power loss density in a slice through the nanoparticle.

Table 1 shows the computed cross sections for the set of angles of incidence.

TABLE 1: CROSS SECTIONS.

θ	ϕ	σ_{abs} (m^2)	σ_{sca} (m^2)	σ_{ext} (m^2)
0	0	$9.6 \cdot 10^{-14}$	$1.3 \cdot 10^{-13}$	$2.3 \cdot 10^{-13}$
$\pi/6$	0	$8.2 \cdot 10^{-14}$	$1.2 \cdot 10^{-13}$	$2.0 \cdot 10^{-13}$
$\pi/6$	$\pi/4$	$8.2 \cdot 10^{-14}$	$1.2 \cdot 10^{-13}$	$2.0 \cdot 10^{-13}$
$\pi/4$	$\pi/4$	$6.7 \cdot 10^{-14}$	$9.2 \cdot 10^{-14}$	$1.6 \cdot 10^{-13}$

For this small sample of the angular space, both cross sections indicate a strong dependence on the polar angle but little variation with the azimuthal angle. For a comparison, the nanoparticle covers a geometric area of $1.59 \cdot 10^{-13}$ m^2 of the substrate.

Figure 7 shows the polar plot of the radiation pattern of the far-field norm in the *yz*-plane. The angular distribution shows that for this angle of incidence, maximum radiation occurs in a direction normal to the interface between the dielectric substrate and the air domain.

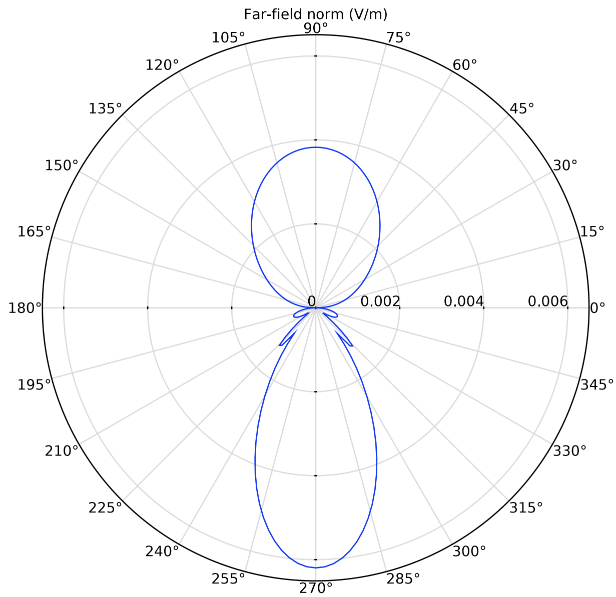


Figure 7: Radiation pattern of the far-field norm in the yz-plane for $\phi = 0$, $\theta = \pi/6$.

Notes About the COMSOL Implementation

The Electromagnetic Waves, Frequency Domain interface features an option to solve for the scattered field, a perturbation to the total field caused by a local scatterer. The incident wave is then entered as a background electric field. This field should be a solution to the wave equation without the presence of the scatterer.

If the scatterer is suspended in free space or any other homogeneous medium, the background field is simply what you are sending in, for example a Gaussian or a plane wave. With the scatterer placed on a substrate, the analytical expression for the background field becomes more complicated. It needs to be the correct superposition of an incident and a reflected wave in the free space domain, and a transmitted wave in the substrate.

A simple and general way to avoid deriving and entering 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 a Periodic Structure node. This node automatically adds two Periodic Port conditions as subnodes. One defines the incident plane wave and allows for specular reflection. The other absorbs the transmitted plane wave. Additionally, the Periodic Structure node automatically adds Floquet Periodic Condition subnodes. These conditions state that the solution on one side of the geometry equals 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 in the xy-plane.

The propagation direction and the polarization of the incident electric field are input parameters for the Periodic Structure node that automatically configures the Periodic Ports and the Floquet Periodic Conditions. Using the coordinate system in Figure 1, the incident wave vector is

$$\mathbf{k}_a = (k_x, k_y, k_{az}) = k_a(\cos\phi_a \sin\theta_a, \sin\phi_a \sin\theta_a, -\cos\theta_a)$$

where k_a is the wave number in the first medium, here vacuum, ϕ_a and θ_a the azimuthal and polar angles of incidence. The expression for the tangentially polarized electric field vector at the plane of incidence becomes

$$\mathbf{E}_0 = E_0(-\sin\phi_a, \cos\phi_a, 0)\exp(-i(k_x x + k_y y))$$

This linear polarization is also known as s-polarization, from the German word *senkrecht* (for perpendicular), as the polarization is orthogonal to the plane of incidence (spanned by the incident wave vector and the port boundary normal).

The Periodic Structure node lets you define a total input power from which the electric field amplitude E_0 is derived. The model uses the value

$$P = I_0 A \cos\theta$$

where $I_0 = 1\ \mathrm{MW/m^2}$ is the intensity of the incident field and A the area of the boundary where the port is set up.

In the substrate, the wave vector is

$$\mathbf{k}_b = (k_x, k_y, k_{bz}) = k_b(\cos\phi_b \sin\theta_b, \sin\phi_b \sin\theta_b, -\cos\theta_b)$$

with

$$k_b = \frac{n_b}{n_a} k_a$$

$$\phi_b = \phi_a$$

$$\sin\theta_b = \frac{n_a}{n_b}\sin\theta_a$$

Notice that the x and y components for the wave vector are the same for the wave in the substrate and the incident wave, due to field continuity.

The electric field vector at the output port is proportional to

$$(-\sin\phi_b, \cos\phi_b, 0)\exp(-i(k_x x + k_y y))\ .$$

Thus, the mode fields and the mode field amplitudes are the same at the output port as at the input port.

Table 2 compares the results for the background field reflectance and the corresponding analytical value. For more information, see (Fresnel Equations).

TABLE 2: COMPUTED AND ANALYTICAL POWER REFLECTION COEFFICIENTS.

θ	ϕ	`ewfd.Rport_0_0`	R
0	0	0.0400	0.0400
π/6	0	0.0579	0.0578
π/6	π/4	0.0578	0.0578
π/4	π/4	0.0920	0.0920

A second Electromagnetic Waves, Frequency Domain interface introduces the gold nanoparticle as the scatterer and surrounds the geometry with PMLs. With the full field solution from the first interface as the background field, only the scattered field needs to be absorbed in the PMLs.

Far-field analysis can be useful to better understand the angular and spatial distribution of the light scattered by the scatterer. In this model, the far field domain has inhomogeneous material properties, as it consists of the air and the substrate. Therefore, a Far-Field Domain, Inhomogeneous node is used to calculate the far field radiation. This node automatically adds three subnodes: Superstrate, Substrate, and Far-Field Calculation. The Superstrate and the Substrate subnodes are used for the selection of the air and the substrate domain group, respectively. The Far-Field Calculation subnode is used to select the boundaries to calculate the far-field variables and define the variable name.

Application Library path: `Wave_Optics_Module/Optical_Scattering/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 **Optics** > **Wave Optics** > **Electromagnetic Waves, Frequency Domain (ewfd)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Optics** > **Wave Optics** > **Electromagnetic Waves, Frequency Domain (ewfd)**.

5 Click **Add**.

After clicking **Add** twice, you should now see two **Electromagnetic Waves, Frequency Domain** entries in the **Added physics interfaces** field.

6 Click **Study**.

7 In the **Select Study** tree, select **Empty Study**.

You will add steps to the study before solving the model.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Define the model parameters. The Description field is optional.

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
w	750[nm]	7.5E-7 m	Width of physical geometry
t_pml	150[nm]	1.5E-7 m	PML thickness
h_air	400[nm]	4E-7 m	Air domain height
h_subs	250[nm]	2.5E-7 m	Substrate domain height
na	1	1	Refractive index, air
nb	1.5	1.5	Refractive index, substrate
lda0	500[nm]	5E-7 m	Wavelength
phi	0	0	Azimuthal angle of incidence in both media
theta	0	0	Polar angle of incidence in air
thetab	asin(na/nb*sin(theta))	0 rad	Polar angle in substrate
I0	1[MW/m^2]	1E6 W/m²	Intensity of incident field
P	I0*w^2*cos(theta)	5.625E-7 W	Port power

The first four parameters will be used in defining the geometry. The azimuthal angle in the substrate remains the same as the angle of incidence. As the polar angle of incidence gets other values in the study, the polar angle in the substrate will automatically be recomputed.

GEOMETRY 1

Import the nanoparticle.

Import 1 (imp1)

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

2 In the **Settings** window for **Import**, locate the **Source** section.

3 Click **Browse**.

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

5 Click **Import**.

Block 1 (blk1)

Draw the air and the substrate using your model parameters.

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 `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

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

10 Clear the Bottom checkbox.

Block 2 (blk2)

1 Right-click **Block 1 (blk1)** and choose **Duplicate**.

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

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

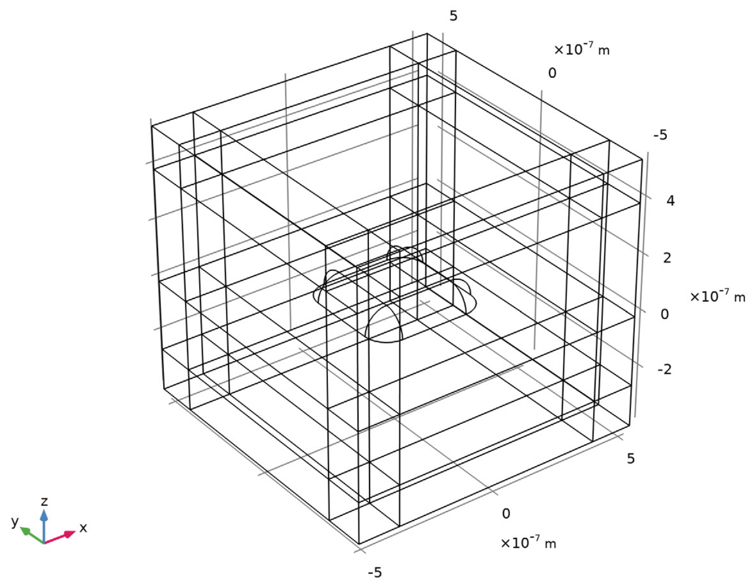
4 Locate the **Position** section. In the **z** text field, type `-(h_subs+t_pml)/2`.

5 Make sure the Left, Right, Front, Back, and Bottom checkboxes are selected. Leave the Top checkbox cleared.

6 Click **Build All Objects**.

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

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



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 separate selections for the nanoparticle.

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, and 25 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**.

Nanoparticle

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

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

3 Select Domain 25 only.

Nanoparticle Surface

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

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

3 Select Domain 25 only.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

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 (ewfd2)**.

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 them to be `0` in the PML domains.

1 In the **Definitions** toolbar, click **Local 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
`ewfd.Ex`	`0`		
`ewfd.Ey`	`0`		
`ewfd.Ez`	`0`		

MATERIALS

Define materials for the air, the substrate, and the nanoparticle.

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 Select Domains 1, 2, 5, 6, 9, 10, 13, 14, 17, 18, 21, 22, 26, 27, 30, 31, 34, and 35 only.

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

ADD MATERIAL

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

Add the material properties of gold from the Optical material library.

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

3 In the tree, select **Optical** > **Inorganic Materials** > **Au - Gold** > **Models and simulations** > **Au (Gold) (Rakic et al. 1998: Brendel-Bormann model; n,k 0.248-6.20 um)**.

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.

MATERIALS

Au (Gold) (Rakic et al. 1998: Brendel-Bormann model; n,k 0.248-6.20 um) (mat3)

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

2 From the **Selection** list, choose **Nanoparticle**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

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.

Define this infinite plane-wave solution using a **Periodic Structure** node. This node will automatically add and configure **Periodic Port** and **Floquet Periodic Condition** subnodes.

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 **Domain Selection** section.

3 From the **Selection** list, choose **Physical Domains**.

Periodic Structure 1

1 In the **Physics** toolbar, click **Domains** and choose **Periodic Structure**.

Add the input power and the angles of incidence.

2 In the **Settings** window for **Periodic Structure**, locate the **Port Mode Settings** section.

3 In the P_{in} text field, type `P`.

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

5 In the α_2 text field, type `phi`.

Wave Equation, Electric 2

1 In the **Physics** toolbar, click **Domains** and choose **Wave Equation, Electric**.

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Domain Selection** section.

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

4 Locate the **Electric Displacement Field** section. From the n list, choose **User defined**. In the associated text field, type `na`.

5 From the k list, choose **User defined**. This redefines the nanoparticle as air.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EWFD2)

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

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain 2 (ewfd2)**.

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

ewfd.Ex	x
ewfd.Ey	y
ewfd.Ez	z

Cross Section Calculation 1

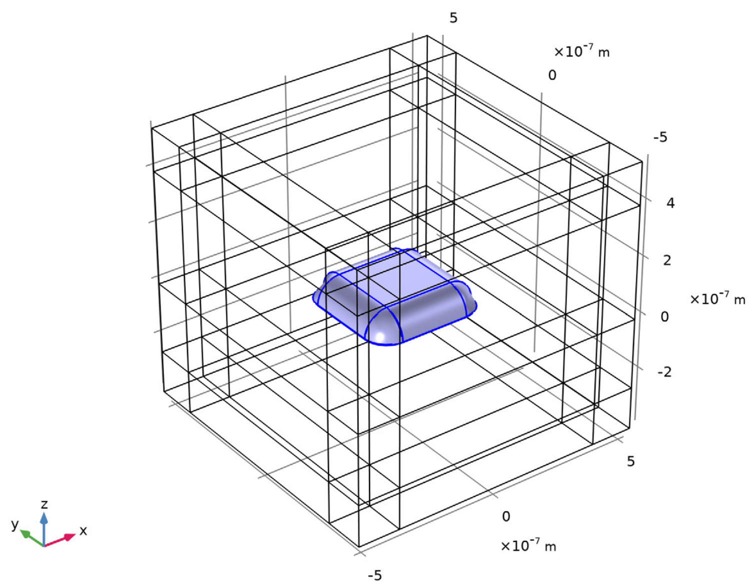
Add **Cross Section Calculation** feature and select the scatterer to calculate the cross section areas.

1 In the **Physics** toolbar, click **Domains** and choose **Cross Section Calculation**.

2 In the **Settings** window for **Cross Section Calculation**, locate the **Domain Selection** section.

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

4 Locate the **Cross Section Calculation** section. In the I_0 text field, type `I0`, which is the intensity of the incident plane wave.



Far-Field Domain, Inhomogeneous 1

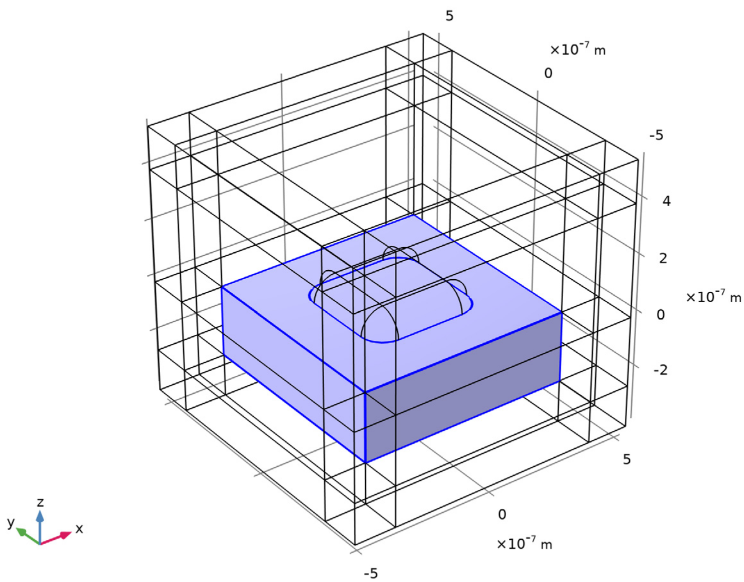
Add **Far-Field Domain, Inhomogeneous** feature to calculate the far-field variables.

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.



MESH 1

The **Physics-controlled mesh** setting creates a mesh with a maximum mesh element size of one sixth of the material wavelength. To resolve the skin depth in the nanoparticle, select **Resolve wave in lossy media** for the second physics interface (**ewfd2**). The periodic boundary conditions get identical triangular meshes and the PML gets a swept mesh.

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

2 In the **Settings** window for **Mesh**, locate the **Electromagnetic Waves, Frequency Domain 2 (ewfd2)** section.

3 Select the **Resolve wave in lossy media** checkbox.

STUDY 1

Add a **Parametric Sweep** for a few different combinations of 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 (Polar angle of incidence in air)	`0 pi/6 pi/6 pi/4`	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
phi (Azimuthal angle of incidence in both media)	`0 0 pi/4 pi/4`	

Step 1: Wavelength Domain

1 In the **Study** toolbar, click **More Study Steps** and choose **Frequency Domain** > **Wavelength Domain**.

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

3 In the **Wavelengths** text field, type `lda0`.

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 (ewfd2)**.

Step 2: Wavelength Domain 2

1 In the **Study** toolbar, click **More Study Steps** and choose **Frequency Domain** > **Wavelength Domain**.

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

3 In the **Wavelengths** text field, type `lda0`.

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 (ewfd)**.

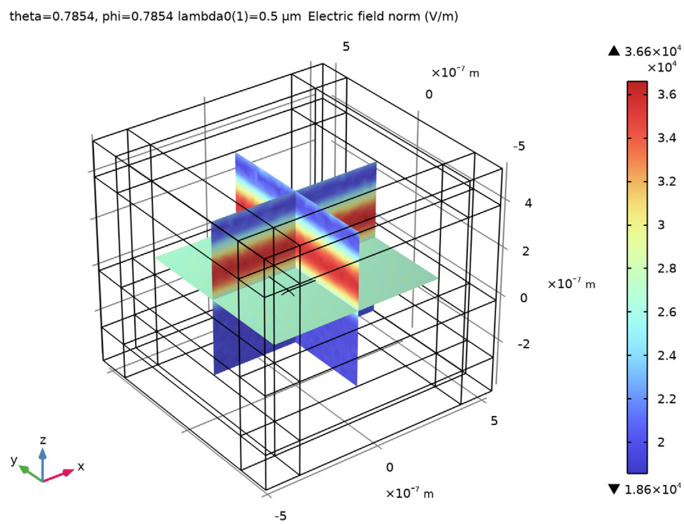
5 In the **Study** toolbar, click **Compute**.

RESULTS

Now, make some small adjustments to the default plots. For the field plots, it is more interesting to see the y-component of the electric field, than the electric field norm.

Electric Field (ewfd)

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



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

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

3 From the **Parameter value (theta,phi)** list, choose **3: theta=0.5236, phi=0.7854**.

Only select the non-PML domains, as the first physics interface is not defined in the PML domains.

4 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 From the **Selection** list, choose **Physical Domains**.

6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

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 3.

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

5 Find the **Z-planes** subsection. In the **Planes** text field, type 0.

Electric Field (ewfd)

Color only the substrate surface to make it clear that you are looking at the field distribution without the nanoparticle.

Surface 1

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

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

3 In the **Expression** text field, type 1.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

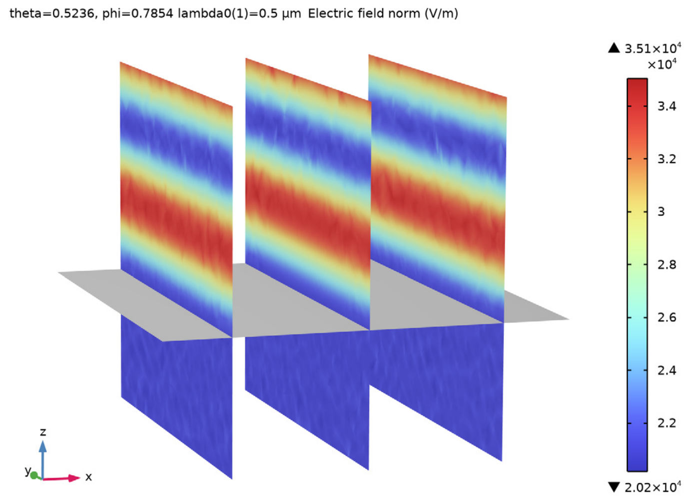
6 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 Select Boundaries 65 and 87 only.

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



. The electric field norm from the first interface confirms that you have a standing wave pattern in the air and a propagating plane wave in the substrate.

Multislice 1

Now, change to plot the *y*-component of the electric field.

1 In the **Model Builder** window, under **Results** > **Electric Field (ewfd)** click **Multislice 1**.

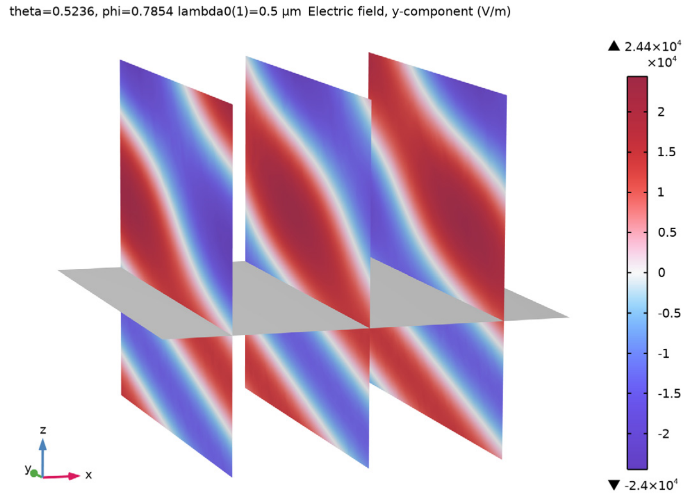
2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `ewfd.Ey`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

5 From the **Scale** list, choose **Linear symmetric**.

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



You can zoom in and rotate the plot you just created, to make it look like the one above.

Background Field, y

1 In the **Model Builder** window, under **Results** click **Electric Field (ewfd)**.

2 In the **Settings** window for **3D Plot Group**, type `Background Field, y` in the **Label** text field.

Reflectance, Transmittance, and Absorptance (ewfd)

To confirm that the first interface was set up correctly, verify that the power reflection at the material interface agrees with the analytical result, by inspecting the first evaluation group.

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

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

The results for the reflectance, `ewfd.Rorder_0_0`, agree reasonably well with the analytical solution, as indicated in Table 2.

Electric Field (ewfd2)

The first default plot for the second physics interface shows the total field norm. First modify this plot group slightly to get a good plot of the field norm. Then change the expression to plot the *y*-component of the field.

1 In the **Model Builder** window, click **Electric Field (ewfd2)**.

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

3 From the **Parameter value (theta,phi)** list, choose **3: theta=0.5236, phi=0.7854**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Field (ewfd2)** 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 3.

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

5 Find the **Z-planes** subsection. In the **Planes** text field, type 0.

Surface 1

In the **Model Builder** window, under **Results** > **Background Field, y** right-click **Surface 1** and choose **Copy**.

Surface 1

In the **Model Builder** window, right-click **Electric Field (ewfd2)** and choose **Paste Surface**.

Include the nanoparticle in this plot, to indicate that this is a result including the scatterer.

Surface 2

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

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Nanoparticle Surface**.

Surface 2

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

Material Appearance 1

1 In the **Electric Field (ewfd2)** toolbar, click **Material Appearance**.

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

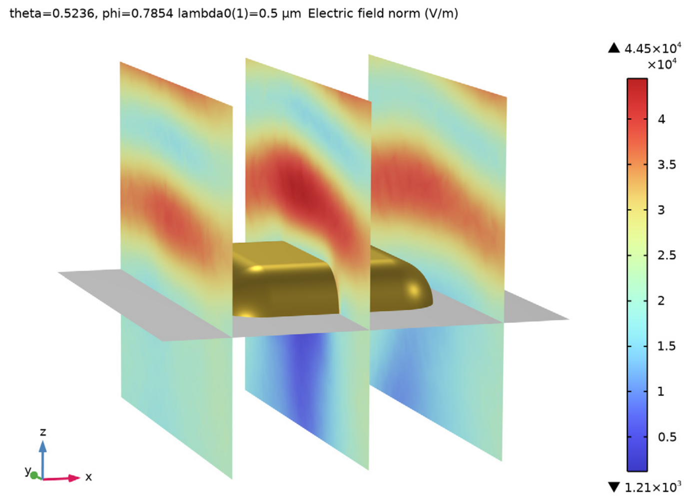
3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Gold**.

Total Field, y

1 In the **Model Builder** window, under **Results** click **Electric Field (ewfd2)**.

2 In the **Electric Field (ewfd2)** toolbar, click **Plot**.



Now, modify this plot group to show the y-component of the total electric field.

3 In the **Settings** window for **3D Plot Group**, type `Total Field, y` in the **Label** text field.

Multislice 1

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

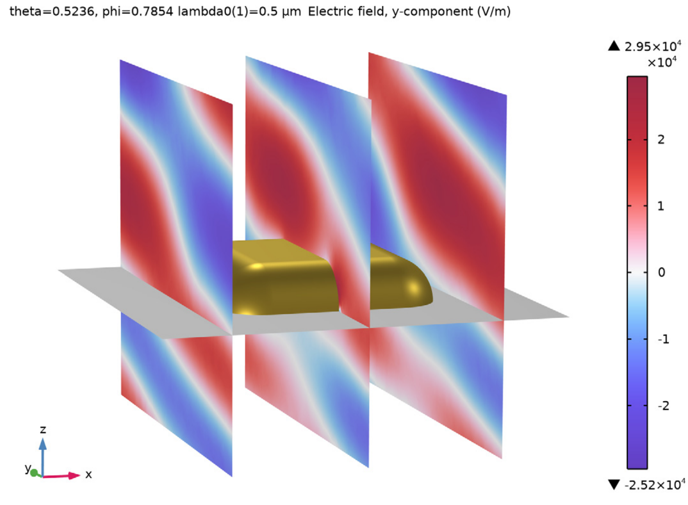
2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `ewfd2.Ey`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

5 From the **Scale** list, choose **Linear symmetric**.

6 In the **Total Field, y** toolbar, click **Plot**.



Electric Field, Background (ewfd2)

Inspect and slightly modify the default plot of the background field, used by the second physics interface.

1 In the **Model Builder** window, under **Results** click **Electric Field, Background (ewfd2)**.

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

3 From the **Parameter value (theta,phi)** list, choose **3: theta=0.5236, phi=0.7854**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

In the **Model Builder** window, under **Results** > **Total Field, y** right-click **Surface 1** and choose **Copy**.

Surface 1

In the **Model Builder** window, right-click **Electric Field, Background (ewfd2)** and choose **Paste Surface**.

Multislice 1

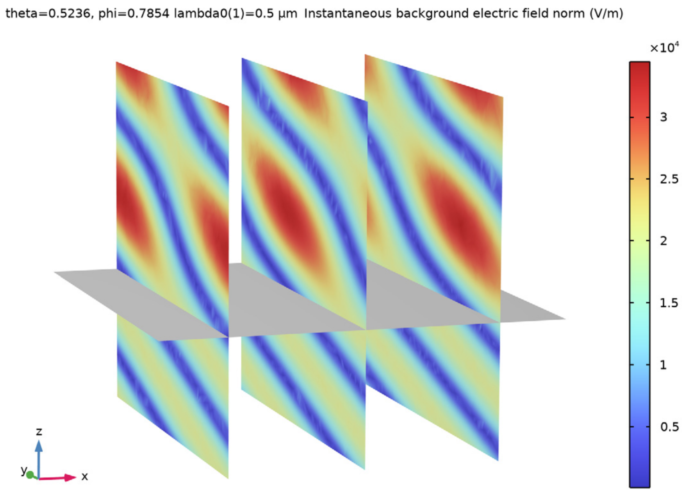
1 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

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

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

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

5 In the **Electric Field, Background (ewfd2)** toolbar, click **Plot**.



This plot shows the instantaneous norm of the background field. The instantaneous norm only includes the norm of the real part of the field. This gives a more wave-like character to the plot. Furthermore, it is also possible to animate this plot, to see how the wave propagates.

2D Far Field (ffi1)

When the **Far-Field Domain, Inhomogeneous** feature is part of the model, by default a 2D polar plot of the total far-field norm radiation pattern is created.

1 In the **Model Builder** window, under **Results** click **2D Far Field (ffi1)**.

2 In the **Settings** window for **Polar Plot Group**, locate the **Data** section.

3 From the **Parameter selection (theta, phi)** list, choose **From list**.

4 In the **Parameter values (theta,phi)** list box, select **2: theta=0.5236, phi=0**.

Radiation Pattern 1

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

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

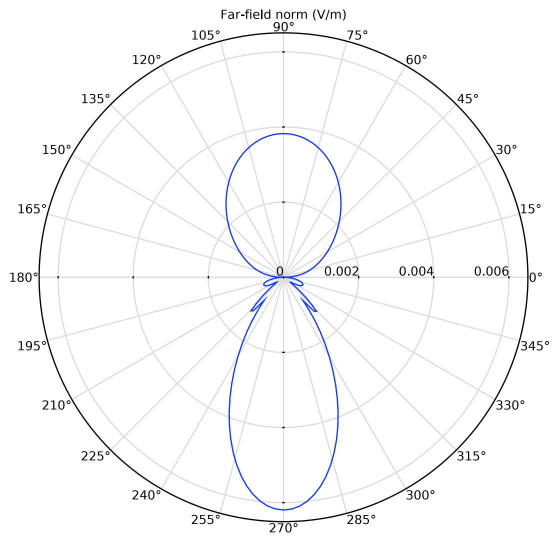
3 Find the **Normal vector** subsection. In the **x** text field, type 1.

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

5 Find the **Reference direction** subsection. In the **x** text field, type 0.

6 In the **y** text field, type 1.

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



Cross Sections (csc1)

The cross-section variables are available for evaluation in an evaluation group.

1 In the **Model Builder** window, under **Results** click **Cross Sections (csc1)**.

2 In the **Cross Sections (csc1)** toolbar, click **Evaluate**.

The results should resemble those in Table 1.

Electric Field, Background (ewfd2)

Finally, create a plot of the power loss in the particle, reproducing Figure 6.

Power Loss

1 In the **Model Builder** window, right-click **Electric Field, Background (ewfd2)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Power Loss` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Power Loss** 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 **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

5 In the **Coordinates** text field, type `50[nm]`.

6 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain 2** > **Heating and losses** > **ewfd2.Qh - Total power dissipation density - W/m³**.

Power Loss

1 Click the **Go to Default View** button in the **Graphics** toolbar.

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

3 In the **Model Builder** window, click **Power Loss**.