Conical Antenna

Introduction

Conical antennas are useful for many applications due to their broadband characteristics and relative simplicity. This example includes an analysis of the antenna impedance and the radiation pattern as functions of the frequency for a monoconical antenna with a finite ground plane and a 50 Ω coaxial feed. The rotational symmetry makes it possible to model this in axially symmetric 2D. When modeling in 2D, you can use a dense mesh, giving an excellent accuracy for a wide range of frequencies.

Model Definition

The antenna geometry consists of a 0.2 m tall metallic cone with a top angle of 90 degrees on a finite ground plane of a 0.282 m radius. The coaxial feed has a central conductor of 1.5 mm radius and an outer conductor (screen) of 4.916 mm radius separated by a teflon dielectric of relative permittivity of 2.07. The central conductor of the coaxial cable is connected to the cone, and the screen is connected to the ground plane.

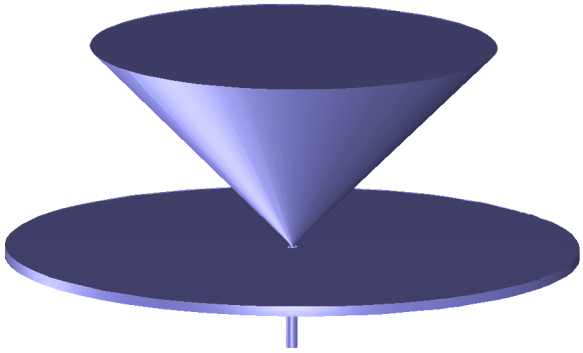


Figure 1: The geometry of the antenna. The central conductor of the coaxial cable is connected to the metallic cone, and the cable screen is connected to the finite ground plane.

The model takes advantage of the rotational symmetry of the problem, which allows modeling in 2D using cylindrical coordinates. You can then use a very fine mesh to achieve an excellent accuracy.

©2012 COMSOL

DOMAIN EQUATIONS

An electromagnetic wave propagating in a coaxial cable is characterized by transverse electromagnetic fields (TEM). Assuming time-harmonic fields with complex amplitudes containing the phase information, you have:

$$\mathbf{E} = \mathbf{e}_r \frac{C}{r} e^{j(\omega t - kz)}$$

$$\mathbf{H} = \mathbf{e}_\varphi \frac{C}{rZ} e^{j(\omega t - kz)}$$

where z is the direction of propagation and r, φ, and z are cylindrical coordinates centered on axis of the coaxial cable. Z is the wave impedance in the dielectric of the cable, and C is an arbitrary constant. The angular frequency is denoted by ω. The propagation constant, k, relates to the wavelength in the medium λ as

$$k = \frac{2\pi}{\lambda}$$

In the air, the electric field also has a finite axial component whereas the magnetic field is purely azimuthal. Thus it is possible to model the antenna using an axisymmetric transverse magnetic (TM) formulation, and the wave equation becomes scalar in H_φ:

$$\nabla \times \left(\frac{1}{\varepsilon} \nabla \times H_\varphi \right) - \mu \omega^2 H_\varphi = 0$$

BOUNDARY CONDITIONS

The boundary conditions for the metallic surfaces are:

$$\mathbf{n} \times \mathbf{E} = 0$$

At the feed point, a matched coaxial port boundary condition is used to make the boundary transparent to the wave. The antenna is radiating into free space, but you can only discretize a finite region. Therefore, truncate the geometry some distance from the antenna using a scattering boundary condition allowing for outgoing spherical waves to pass with very little reflections. A symmetry boundary condition is automatically applied on boundaries at $r = 0$.

Results and Discussion

Figure 2 shows the antenna impedance as a function of frequency. Ideally, the antenna impedance should be matched to the characteristic impedance of the feed, 50 Ω, to

 ©2012 COMSOL

obtain maximum transmission into free space. This is quite well fulfilled in the high frequency range.

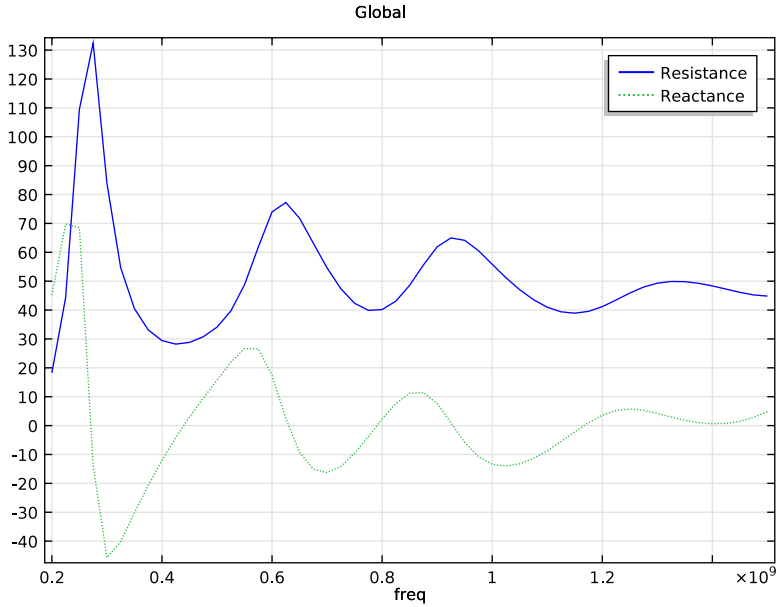


Figure 2: The antenna impedance in Ω as a function of frequency from 200 MHz to 1.5 GHz. The solid line shows the radiation resistance and the dotted line represents the reactance.

Figure 3 shows the antenna radiation pattern in the near-field for three different frequencies. The effect of the finite diameter of the ground plane is to lift the main lobe from the horizontal plane. For an infinite ground plane or in the high-frequency limit, the radiation pattern is symmetric around zero elevation. This is easy to understand, as an infinite ground plane can be replaced by a mirror image of the monocone below the plane. Such a biconical antenna is symmetric around zero elevation and has its main lobe in the horizontal direction. The decreased lobe lifting at higher frequencies is just about visible in Figure 3.

Figure 4 shows the antenna radiation pattern in the far field for the same frequencies as the radiation pattern at the boundary in Figure 3.

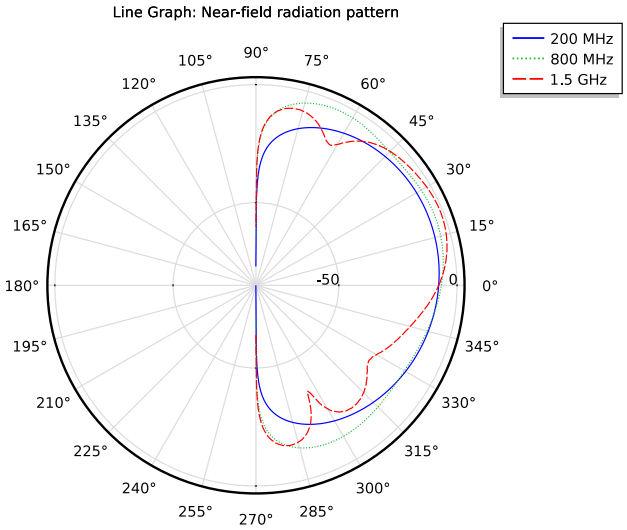


Figure 3: Polar plot of the antenna radiation pattern in the near field versus the elevation angle for 200 MHz, 800 MHz, and 1.5 GHz. The scale is logarithmic.

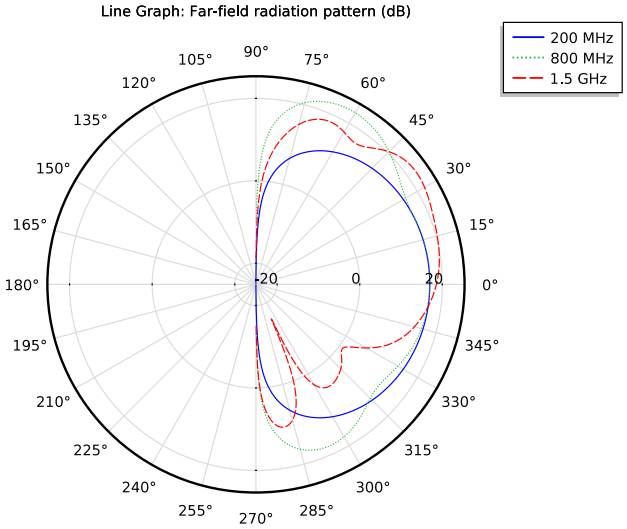


Figure 4: Polar plot of the antenna radiation pattern for the far field versus the elevation angle for 200 MHz, 800 MHz, and 1.5 GHz. This plot is normalized differently but has a shape resembling the near field.

©2012 COMSOL

As the frequency increases the antenna impedance gets closer to 50 Ω, which means that a voltage generator connected to the input of the antenna should have an output impedance of 50 Ω.

Model Library path: `RF_Module/RF_and_Microwave_Engineering/conical_antenna`

Modeling Instructions

MODEL WIZARD

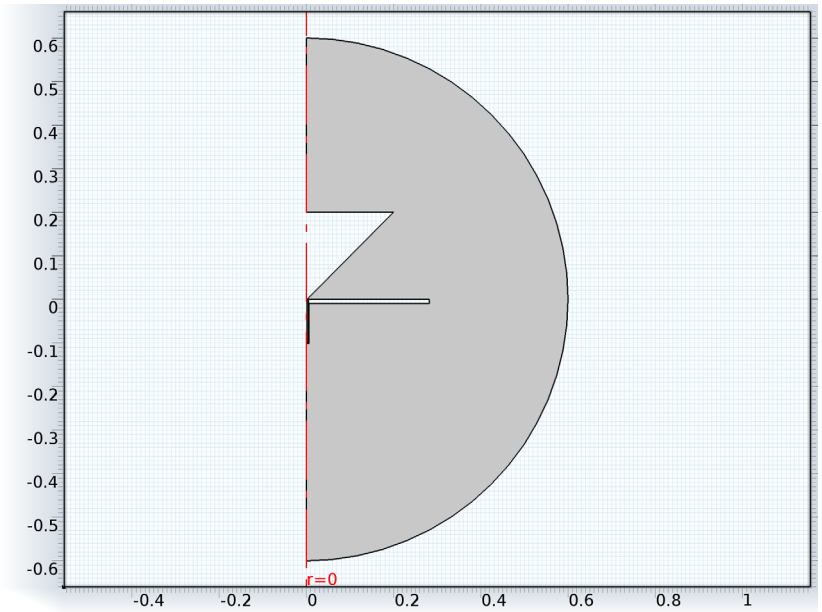
1. Go to the **Model Wizard** window.
2. Click the **2D axisymmetric** button.
3. Click **Next**.
4. In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
5. Click **Next**.
6. Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
7. Click **Finish**.

GEOMETRY 1

Import 1

1. In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Import**.
2. In the **Import** settings window, locate the **Import** section.
3. Click the **Browse** button.
4. Browse to the model's Model Library folder and double-click the file `conical_antenna.mphbin`.

5 Click the **Import** button.



The imported geometry is effectively a half circle with the metal areas removed. You model the electromagnetic waves in the air and the dielectric material inside the coaxial cable. There is no need to include the metal as a domain in the model because the fields in it are essentially zero except on its surface.

GLOBAL DEFINITIONS

Prepare for the impedance computation by making a few definitions.

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
`Z_tl`	`50[ohm]`	`Characteristic transmission line impedance`

©2012 COMSOL

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
`Z`	`Z_tl*(1+emw.S11)/(1-emw.S11)`	`Antenna impedance`

`emw.S11` is the name of the automatically computed reflection S-parameter.

DEFINITIONS

Define the following selections in order to get easy access to some frequently used domains and boundaries.

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 1** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Air` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 1 only.

Explicit 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 2** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Dielectric` in the **New name** edit field.

4 Click **OK**.

The dielectric domain is inside the coaxial cable just below the cone. It is easier to select it if you zoom in a little.

5 Select Domain 2 only.

Explicit 3

1 In the **Model Builder** window, right-click **Definitions** and choose **Selections>Explicit**.

2 Right-click **Explicit 3** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `Outer Air Boundaries` in the **New name** edit field.

4 Click **OK**.

5 In the **Explicit** settings window, locate the **Input Entities** section.

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

7 Select Boundaries 14 and 15 only.

With all selections and expressions now defined, it is time to set up the materials and the physics of the model.

MATERIALS

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

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

4 Right-click and choose **Add Material to Model** from the menu.

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

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

PROPERTY	NAME	VALUE
Relative permittivity	epsilonr	2.07
Relative permeability	mur	1
Electric conductivity	sigma	0

©2012 COMSOL

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Port 1

1 In the **Model Builder** window, right-click **Model 1>Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Port**.

Set up Boundary 6, at the bottom of the coaxial cable, to be a Port. You can zoom in on this part of the geometry to easier find and select this boundary.

2 Select Boundary 6 only.

3 In the **Port** settings window, locate the **Port Properties** section.

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

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

Scattering Boundary Condition 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Scattering Boundary Condition**.

2 In the **Scattering Boundary Condition** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outer Air Boundaries**.

The Scattering boundary condition is a simple way of letting the waves undergo only minor artificial reflections as they leave the computational domain through the exterior boundaries. To minimize these reflections, but at a greater computational cost, you can use Perfectly Matched Layers (see the section Perfectly Matched Layers (PMLs) in the *RF Module User's Guide*).

Perfect Electric Conductor 1

As you can see if you click the Perfect Electric Conductor node under Electromagnetic Waves, the physical boundaries to which you have not assigned any boundary condition will by default be considered perfect electric conductors. This is a good approximation for most metals throughout the frequency range considered in this model.

Far-Field Domain 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the domain setting **Far-Field Domain**.

2 Select Domain 1 only.

Far-Field Calculation 1

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

2 In the **Far-Field Calculation** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outer Air Boundaries**.

Adding a Far Field Calculation feature does not affect the physics of the model, but makes it possible to study the far field generated by the antenna. Select the boundaries to use for this computation so that, in the physical (3D) geometry, they surround all sources and reflecting objects. The outer air boundaries are a convenient choice.

MESH 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Free Triangular**.

Because this is a relatively small 2D model, you could easily use one of the finer mesh defaults and get a good accuracy. The following settings however are designed to get you a fine mesh mostly where it is needed.

Size

1 In the **Size** settings window, locate the **Element Size Parameters** section.

2 In the **Maximum element size** edit field, type `25[mm]`.

This global maximum element size makes sure that the mesh everywhere resolves the wavelength.

Size 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Dielectric**.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type `0.5[mm]`.

You now have an especially fine mesh inside the coaxial cable, where the wave is produced.

Size 2

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

 ©2012 COMSOL

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 4 and 8 only.

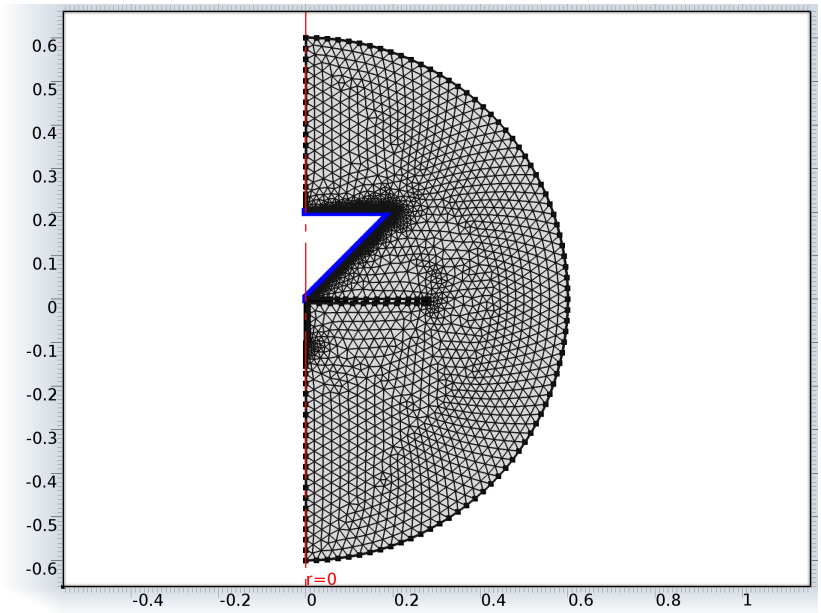
5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated edit field, type `2.5[mm]`.

This gives a fine mesh on the surface of the antenna.

8 Click the **Build All** button.



STUDY 1

Step 1: Frequency Domain

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

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

3 In the **Frequencies** edit field, type `range(200e6,25e6,1.5e9)`.

The frequency range you just entered runs from 200 MHz to 1.5 GHz in steps of 25 MHz.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Electric field

The plot that appears once the solution process is finished shows the norm of the electric field at 1.5 GHz. The reason it is mostly dark blue is because the range is dominated by the high values in and near the coaxial cable. To better see how the wave propagates, try plotting the instantaneous value of the H-field using a manual range.

1 In the **Model Builder** window, expand the **Electric field** node, then click **Surface**.

2 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field>Magnetic field, phi component (emw.Hphi)**.

3 Click to expand the **Range** section. Select the **Manual color range** check box.

4 In the **Minimum** edit field, type `-0.5`.

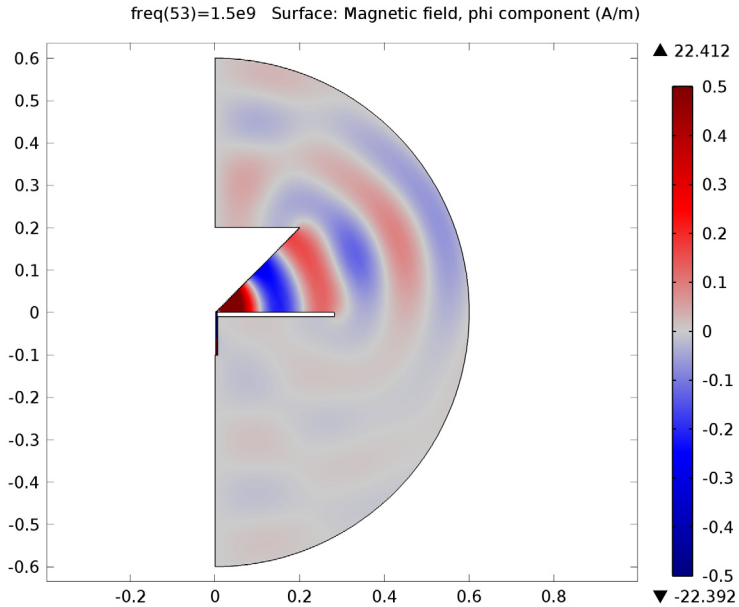
5 In the **Maximum** edit field, type `0.5`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

7 Click the **Plot** button.

 ©2012 COMSOL

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



To plot the impedance as a function of the frequency, set up a 1D plot.

1D Plot Group 4

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

2 Right-click **1D Plot Group 4** and choose **Global**.

3 In the **Global** settings window, locate the **y-Axis Data** section.

4 In the table, enter the following settings:

EXPRESSION	DESCRIPTION
`real(Z)`	`Resistance`
`imag(Z)`	`Reactance`

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

6 Click the **Plot** button.

You have now reproduced Figure 2. Next, visualize the near-field and far-field radiation patterns using polar plots.

Polar Plot Group 5

1 In the **Model Builder** window, right-click **Results** and choose **Polar Plot Group**.

Select a few of the frequencies from the list of parameter values. Showing the radiation pattern for all of them would take a bit of time and lead to a cluttered plot.

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

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

4 In the **Parameter values (freq)** list, choose **2e8**, **8e8**, and **1.5e9**.

5 Right-click **Results>Polar Plot Group 5** and choose **Line Graph**.

6 In the **Line Graph** settings window, locate the **Selection** section.

7 From the **Selection** list, choose **Outer Air Boundaries**.

8 Click **Replace Expression** in the upper-right corner of the **r-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Energy and power>Power outflow, time average (emw.nPoav)**.

9 Locate the **r-Axis Data** section. In the **Expression** edit field, type `10*log10(emw.nPoav)`.

The variable `emw.nPoav` represents the outgoing power flow through the boundaries where it is evaluated. The expression you just entered gives you the same in a logarithmic scale.

10 Select the **Description** check box.

11 In the associated edit field, type `Near-field radiation pattern`.

12 Locate the θ **Angle Data** section. From the **Parameter** list, choose **Expression**.

13 In the **Expression** edit field, type `atan2(z,r)`.

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

15 Click to collapse the **Coloring and Style** section. Click to expand the **Legends** section. Select the **Show legends** check box.

16 From the **Legends** list, choose **Manual**.

17 In the table, enter the following settings:

LEGENDS
200 MHz
800 MHz
1.5 GHz

 ©2012 COMSOL

18 Click to collapse the **Legends** section. Click the **Zoom Extents** button on the Graphics toolbar.

Your near-field radiation plot should look like that in Figure 3.

Finally, visualize the far-field radiation pattern.

Polar Plot Group 6

1 In the **Model Builder** window, right-click **Polar Plot Group 5** and choose **Duplicate**.

2 In the **Model Builder** window, expand the **Results>Polar Plot Group 6** node, then click **Line Graph 1**.

3 In the **Line Graph** settings window, click **Replace Expression** in the upper-right corner of the **r-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Global>Far-field norm, dB (emw.normdBEfar)**.

4 Locate the **r-Axis Data** section. In the **Description** edit field, type `Far-field radiation pattern`.

5 Click the **Plot** button.

The plot should look like that in Figure 4.

©2012 COMSOL