Created in COMSOL Multiphysics 6.1

Simulating Wireless Power Transfer in Circular Loop Antennas

Introduction

This example addresses the concept of wireless power transfer by studying the energy coupling between two circular loop antennas tuned for UHF RFID frequency whose size is reduced using chip inductors. While the orientation of a transmitting antenna is fixed, a receiving antenna is rotating and the best coupling configuration is investigated in terms of S-parameters.

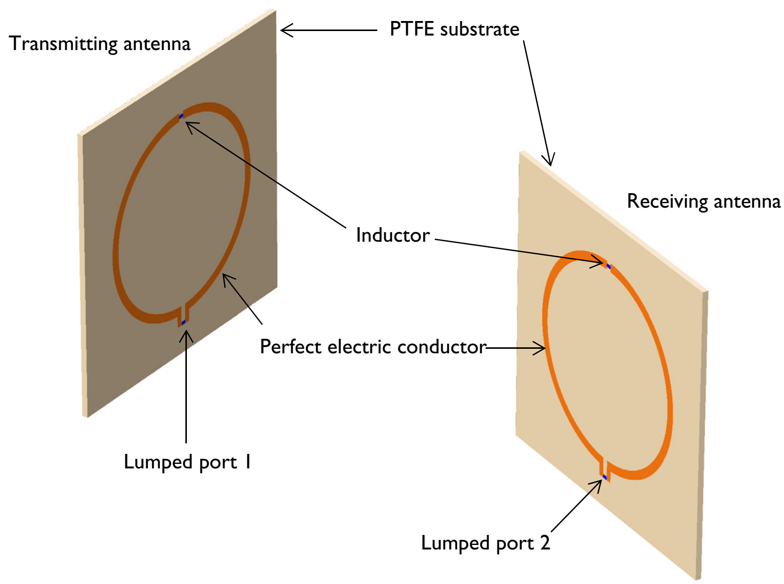


Figure 1: Model set up to compute the coupling effect between two circular loop antennas based on the receiving antenna orientation. The air domain and perfectly matched layers are not shown in this figure.

Model Definition

The model consists of two printed circular loop antennas enclosed by an air domain with perfectly matched layers (PML). The operating frequency of the antennas is 915 MHz for the UFH RFID communication.

A thin copper layer is patterned on a 2 mm Polytetrafluoroethylene (PTFE) board. The thickness of the copper layer is geometrically very thin, but much thicker than the copper skin depth, $\delta_s = (2/\omega\mu\sigma)^{1/2} = 2.15\ \mu\text{m}$ at this frequency, so it is modeled as a perfect electric conductor (PEC). The antenna diameter is reduced down to ~0.22 λ_0 by inserting a lumped inductor representing a 0805 surface mount device in the middle of each circular copper trace. On the split section of each trace configured as PEC, a lumped port with 50 Ω reference impedance is assigned to excite or terminate the antennas.

The surrounding PMLs are necessary to absorb the radiation from the transmitting antenna and describe the antenna coupling in infinite free space.

Results and Discussion

Figure 2 shows E-field norm distribution on the *xy*-plane and an arrow plot of the power flow from the transmitting antenna as a function of the receiving antenna rotation angle. When the two antennas are facing each other; the angle of rotation of the receiving antenna is 0 degrees and the fields are strongly coupled. When the angle of rotation of the receiving antenna is 90 degrees, there is no hot coupling area around the receiving antenna that can be visualized. The red arrows describing the power flow are penetrating the receiving antenna without noticeable distortion.

The computed input matching characteristic of the transmitting antenna via S_{11} is below -20 dB regardless of the receiving antenna orientation.

The coupling relation is summarized by approximating S_{21} in table below:

TABLE 1: S-PARAMETER AS A FUNCTION OF ROTATION ANGLE

ANGLE (DEGREE)	**0**	**22.5**	**45**	**67.5**	**90**
S_{21}	-12.5	-13	-15.2	-20.1	-51.6

The computed S_{21} value also shows almost no coupling at 90 degrees.

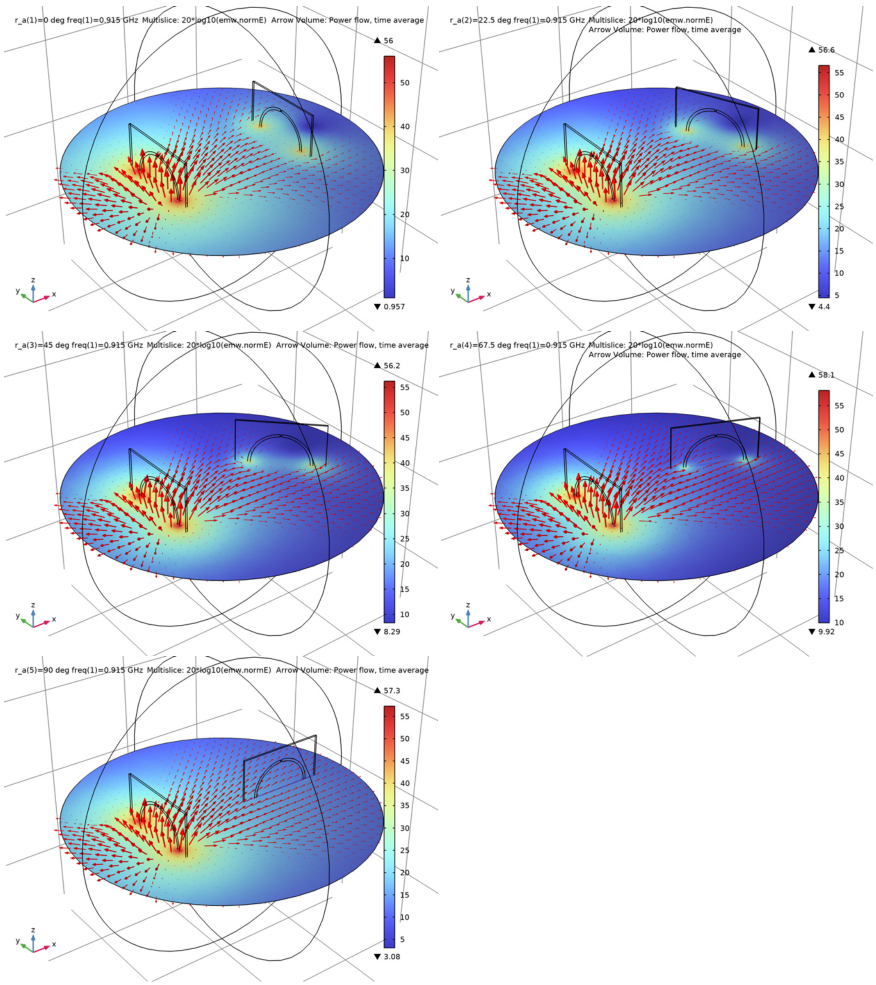


Figure 2: Plot of E-field norm and power flow at z = 0 while the receiving antenna is rotating from 0 to 90 degrees with a step of 22.5 degrees.

Application Library path: `RF_Module/Antennas/uhf_wireless_power_transfer`

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 **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 `915[MHz]`.

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
r_a	0[deg]	0 rad	Rotation 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 **cm**.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 In the **x-coordinate** text field, type `-8`.

5 Click **Show Work Plane**.

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 `3.6`.

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 `3.3`.

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 `0.2`.

4 In the **Height** text field, type `8`.

5 Locate the **Position** section. In the **xw** text field, type `-0.1`.

6 In the **yw** 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 Click the **Zoom Extents** button in the **Graphics** toolbar.

3 Select the object **c1** only.

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

5 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

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

7 Click **Build Selected**.

Work Plane 1 (wp1)>Rectangle 2 (r2)

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 `0.2`.

4 In the **Height** text field, type `0.6`.

5 Locate the **Position** section. In the **xw** text field, type `-0.3`.

6 In the **yw** text field, type `-4`.

Work Plane 1 (wp1)>Rectangle 3 (r3)

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 `0.2`.

4 In the **Height** text field, type `0.6`.

5 Locate the **Position** section. In the **xw** text field, type `0.1`.

6 In the **yw** text field, type `-4`.

Work Plane 1 (wp1)>Union 1 (uni1)

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

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

3 Clear the **Keep interior boundaries** check box.

4 Click in the **Graphics** window and then press Ctrl+A to select all objects.

Work Plane 1 (wp1)>Rectangle 4 (r4)

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 `0.2`.

4 In the **Height** text field, type `0.125`.

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

6 In the **yw** text field, type `-3.9375`.

Work Plane 1 (wp1)>Rectangle 5 (r5)

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 `0.2`.

4 In the **Height** text field, type `0.125`.

5 Locate the **Position** section. In the **yw** text field, type `3.45`.

6 From the **Base** list, choose **Center**.

Block 1 (blk1)

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 `0.2`.

4 In the **Depth** text field, type `10`.

5 In the **Height** text field, type `10`.

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

7 In the **x** text field, type `-8.1`.

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

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Click the **Select All** button in the **Graphics** toolbar.

3 In the **Settings** window for **Mirror**, locate the **Normal Vector to Plane of Reflection** section.

4 In the **x** text field, type `1`.

5 In the **z** text field, type `0`.

6 Locate the **Input** section. Select the **Keep input objects** check box.

7 Click **Build Selected**.

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

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **mir1(1)** and **mir1(2)** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `r_a`.

5 Locate the **Point on Axis of Rotation** section. In the **x** text field, type `8`.

Sphere 1 (sph1)

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

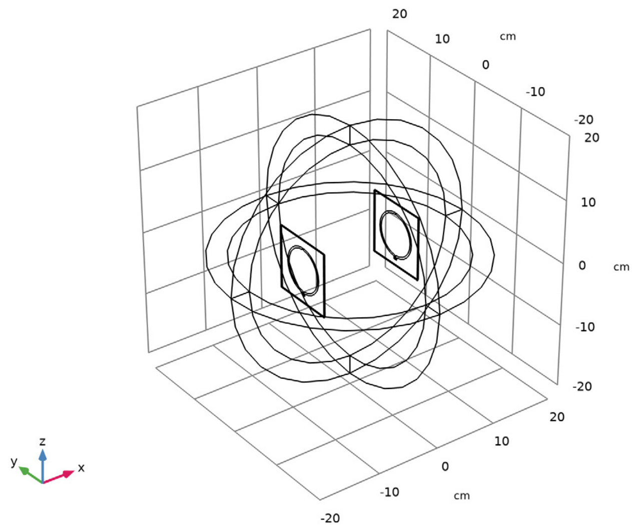
2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `20`.

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

Layer name	Thickness (cm)
`Layer 1`	3

5 Click **Build All Objects**.



DEFINITIONS

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 **Geometry** section.

3 From the **Type** list, choose **Spherical**.

4 Select Domains 1–4 and 7–10 only.

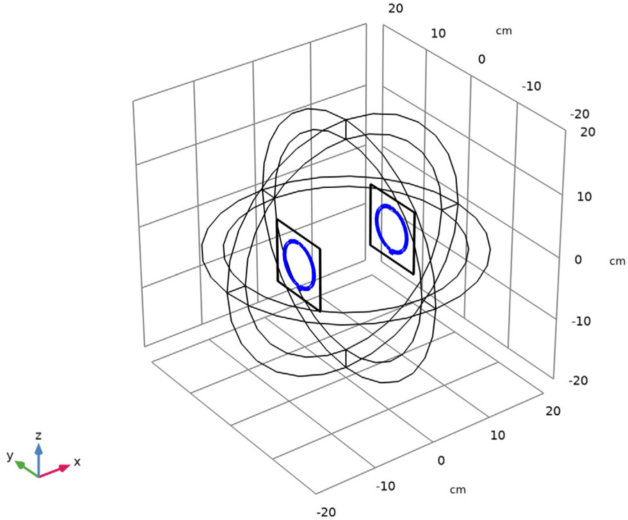
These are the outermost shell domains of the sphere.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Perfect Electric Conductor 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose the boundary condition **Perfect Electric Conductor**.

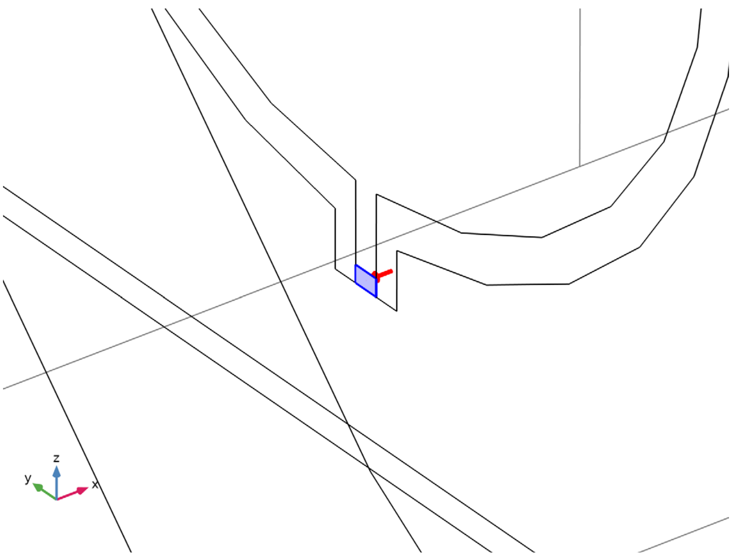
2 Select Boundaries 19, 23, 44, and 48 only.



Lumped Port 1

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

2 Select Boundary 21 only.



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

Lumped Port 2

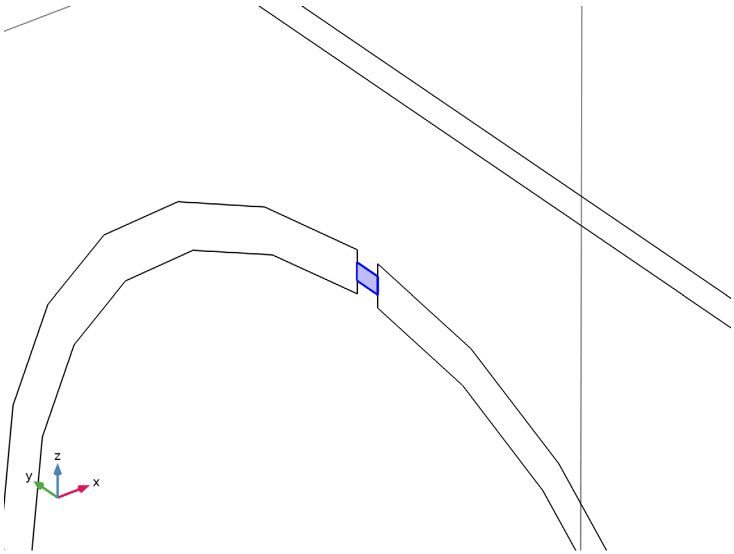
1 In the **Physics** toolbar, click **Boundaries** and choose **Lumped Port**.

2 Select Boundary 46 only.

Lumped Element 1

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

2 Select Boundary 22 only.



3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Inductor**.

5 In the L_{element} text field, type `66[nH]`.

Lumped Element 2

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

2 Select Boundary 47 only.

3 In the **Settings** window for **Lumped Element**, locate the **Settings** section.

4 From the **Lumped element device** list, choose **Inductor**.

5 In the L_{element} text field, type `66[nH]`.

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 Domains 6 and 11 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

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

DEFINITIONS

Hide for Physics 1

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

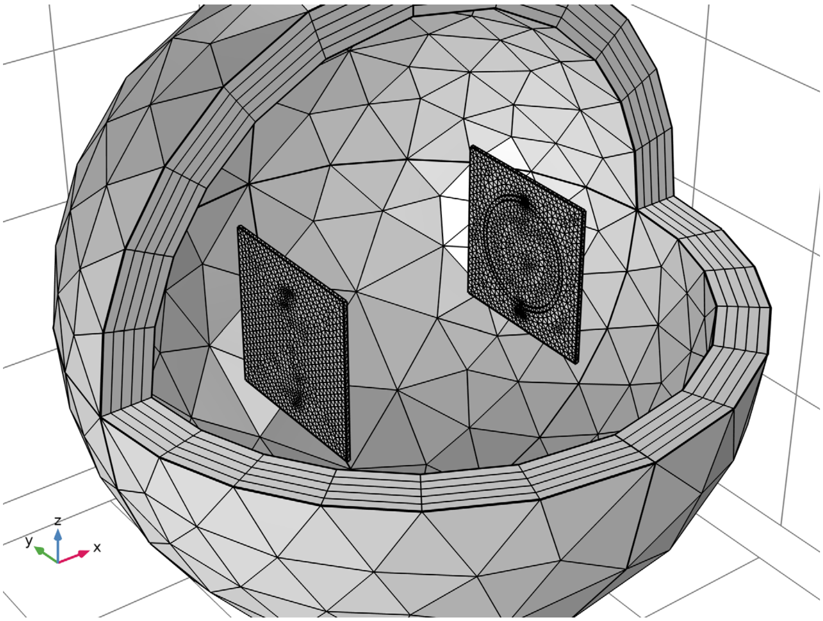
2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 6, 10, 25, 28, and 30 only.

You can define the above selection using the **Paste Selection** button in the setting window.

MESH 1



STUDY 1

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
r_a (Rotation angle)	`range(0[deg],22.5[deg],90[deg])`	`deg`

Step 1: Frequency Domain

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

RESULTS

Multislice

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

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

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

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

Visualize the norm of E-field in dB scale.

5 Locate the **Expression** section. In the **Expression** text field, type `20*log10(emw.normE)`.

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

The field distribution of the PML domains is not of interest, so exclude them from the plot.

Study 1/Parametric Solutions 1 (sol2)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/ Parametric Solutions 1 (sol2)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `5, 6, 11` in the **Selection** text field.

6 Click **OK**.

Electric Field (emw)

Add an arrow volume plot of the power flow.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Energy and power>emw.Poavx,...,emw.Poavz - Power flow, time average**.

3 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. In the **Points** text field, type `31`.

4 Find the **Y grid points** subsection. In the **Points** text field, type `31`.

5 Find the **Z grid points** subsection. In the **Points** text field, type 1.

6 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

7 In the **Range quotient** text field, type 1000.

Electric Field (emw)

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

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

3 From the **Parameter value (r_a (deg))** list, choose **0**.

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

5 From the **Parameter value (r_a (deg))** list, choose **22.5**.

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

7 From the **Parameter value (r_a (deg))** list, choose **45**.

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

9 From the **Parameter value (r_a (deg))** list, choose **67.5**.

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

11 From the **Parameter value (r_a (deg))** list, choose **90**.

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

Compare all reproduced plots with Figure 2.

S-parameter (emw)

The computed S_{11} should be below -10 dB and the computed S_{21} should decrease as the receiving loop antenna rotates.