Created in COMSOL Multiphysics 5.6

Sierpinski Fractal Monopole Antenna

Introduction

A fractal is a mathematical form showing self-repeating patterns. By virtue of its geometrical properties, a fractal structure can generate multiple resonances in RF applications. This antenna model uses a 3rd-order Sierpinski triangle, and the calculated S-parameters show good input matching at the higher-order resonances.

Model Definition

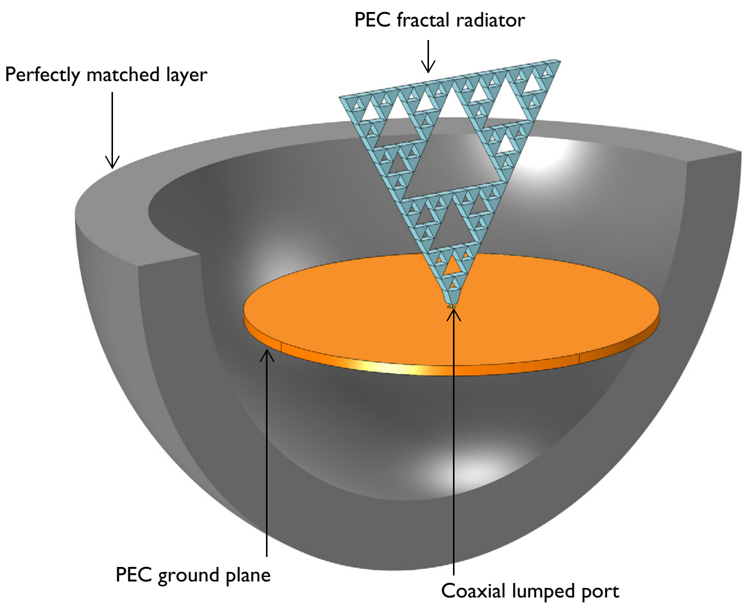


Figure 1: Sierpinski fractal monopole antenna with PEC ground plane enclosed by PML.

Figure 1 depicts the Sierpinski fractal monopole antenna model composed of the fractal radiator, ground plane, coaxial feed, and perfectly matched layer. The fractal radiator part, with the shape of an inverted triangle, is modeled as a perfect electric conductor (PEC) placed on top of the ground plane. The bottom end of the fractal structure, which is originally a pointed triangular geometry, is modified to be flat to connect to the dielectric-filled coaxial cable feed. The diameter of the ground plane is larger than the half-wave of the antenna's 2nd resonance. All metal parts, including the ground plane as well as the coax inner and outer conductors, are modeled as perfect electric conductors. The entire antenna structure is enclosed by a perfectly matched layer.

Results and Discussion

The calculated S-parameters around the 2nd and 3rd resonant frequencies are less than –10 dB. Additional matching parts are not required for using the antenna at the higher-order resonances with the given fractal structure. The E-field norm plot on the PEC surface in Figure 2 describes the hotspot, about a quarter-wavelength, corresponding to its resonant frequency.

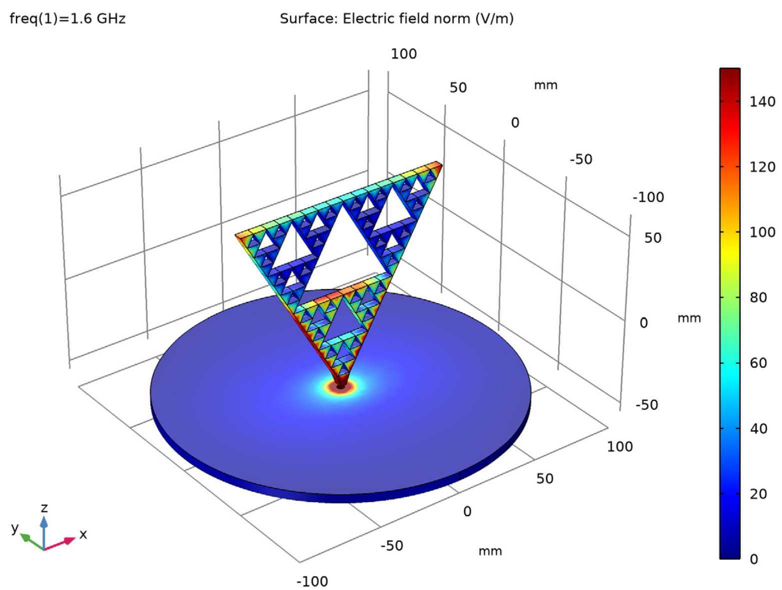


Figure 2: The visualized E-field on the radiator shows the hotspot at the 2nd resonance.

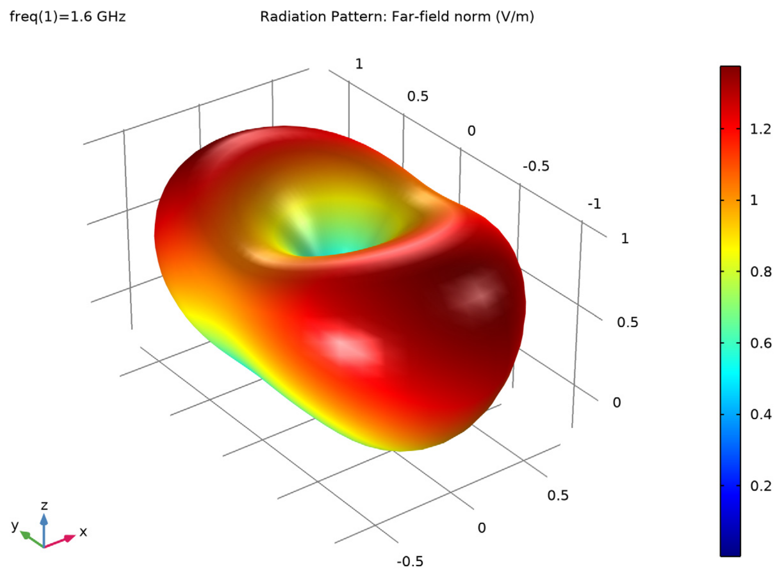


Figure 3: 3D far-field radiation pattern at the 2nd resonance. Some distortion at the lower half space is observed due to the ground plane.

Application Library path: `RF_Module/Antennas/sierpinski_fractal_monopole_antenna`

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 `1.6[GHz] 3[GHz]`.

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 Click **Load from File**.

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

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 **mm**.

Start by activating wireframe rendering to get a better view of the coaxial cable parts which are inside the ground plane.

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

Create a cylinder for the ground plane.

Ground plane

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

2 In the **Settings** window for **Cylinder**, type `Ground plane` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `100`.

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

5 Locate the **Position** section. In the **z** text field, type `-55`.

Add two cylinders for the inner outer conductor of the coaxial cable.

Coax inner

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

2 In the **Settings** window for **Cylinder**, type `Coax inner` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_inner`.

4 In the **Height** text field, type `thickness+2`.

5 Locate the **Position** section. In the **z** text field, type `-55`.

Coax outer

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

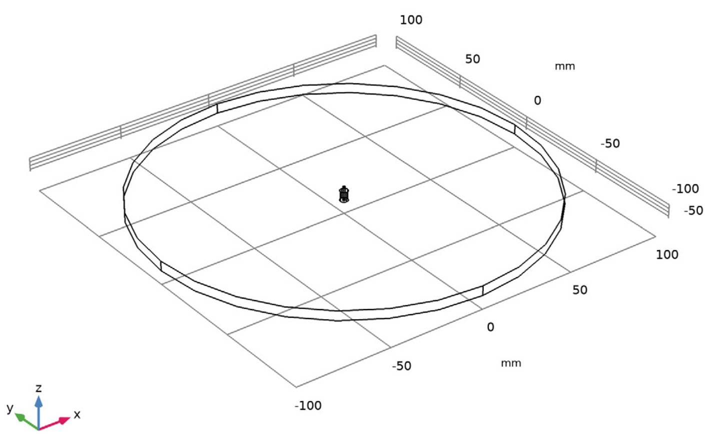
2 In the **Settings** window for **Cylinder**, type `Coax outer` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_outer`.

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

5 Locate the **Position** section. In the **z** text field, type `-55`.

6 Click **Build Selected**.



Add a sphere for the PML. Use the layer definition to create a shell type structure.

PMLs

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

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

3 Locate the **Size** section. In the **Radius** text field, type `180[mm]`.

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

Layer name	Thickness (mm)
Layer 1	35

Next, add a work plane where you will create the Sierpinski fractal triangle.

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 **zx-plane**.

4 In the **y-coordinate** text field, type `-thickness/2`.

5 Click **Show Work Plane**.

Work Plane 1 (wp1)>Plane Geometry

Use polygon to create a triangle. The initial triangle is not equilateral but isosceles to get integer numbers on the grid for easier and simpler copy operations.

Work Plane 1 (wp1)>Polygon 1 (pol1)

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

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

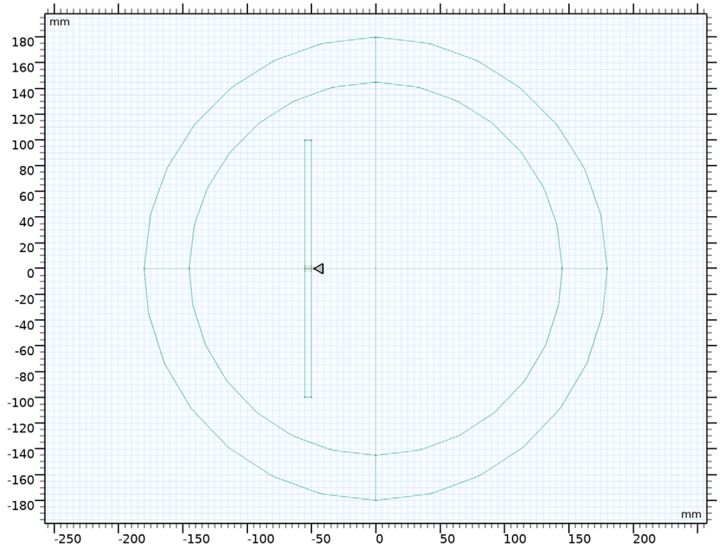
3 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

4 In the table, enter the following settings:

xw (mm)	yw (mm)
x1_tri	y1_tri
x2_tri	y2_tri
x3_tri	y3_tri

5 Click **Build Selected**.

6 Click the **Zoom In** button in the **Graphics** toolbar, a couple of times.



This triangle is a seed object for the Sierpinski fractal triangle. Use Copy and Union repeatedly as shown below until it reaches 3rd order.

Work Plane 1 (wp1)>Copy 1 (copy1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **pol1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **xw** text field, type `7,7`.

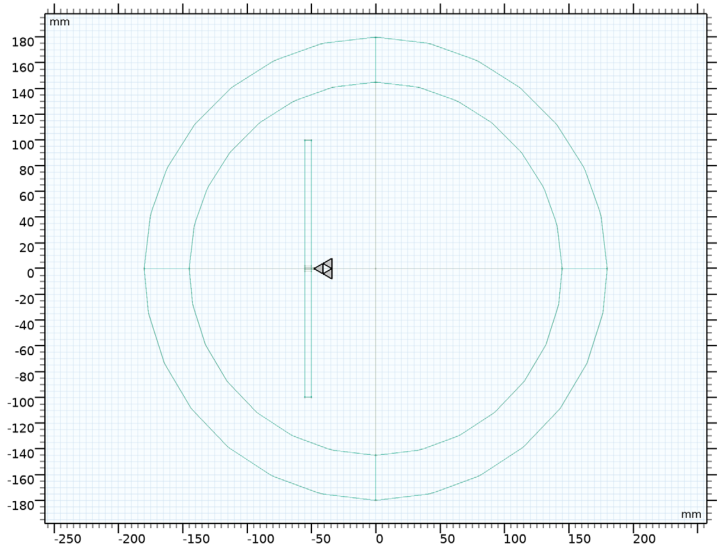
5 In the **yw** text field, type `4,-4`.

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

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

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

3 In the **Settings** window for **Union**, click **Build Selected**.



Work Plane 1 (wp1)>Copy 2 (copy2)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **xw** text field, type `14,14`.

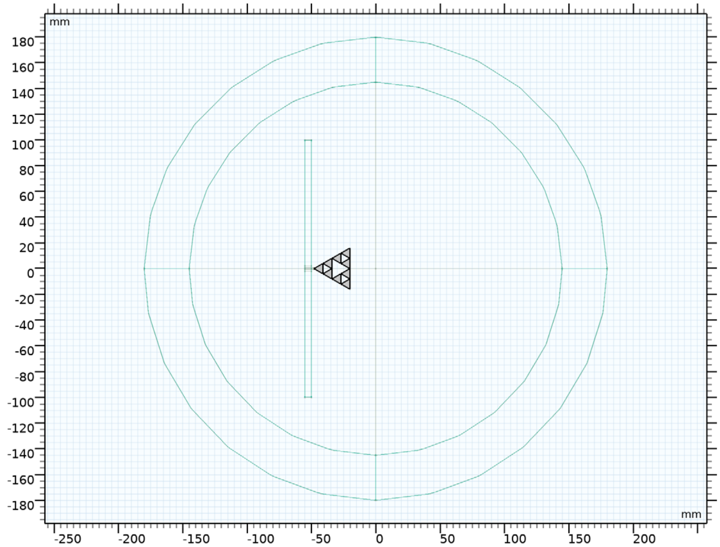
5 In the **yw** text field, type `8,-8`.

Work Plane 1 (wp1)>Union 2 (uni2)

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

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

3 In the **Settings** window for **Union**, click **Build Selected**.



Work Plane 1 (wp1)>Copy 3 (copy3)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **uni2** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **xw** text field, type `28,28`.

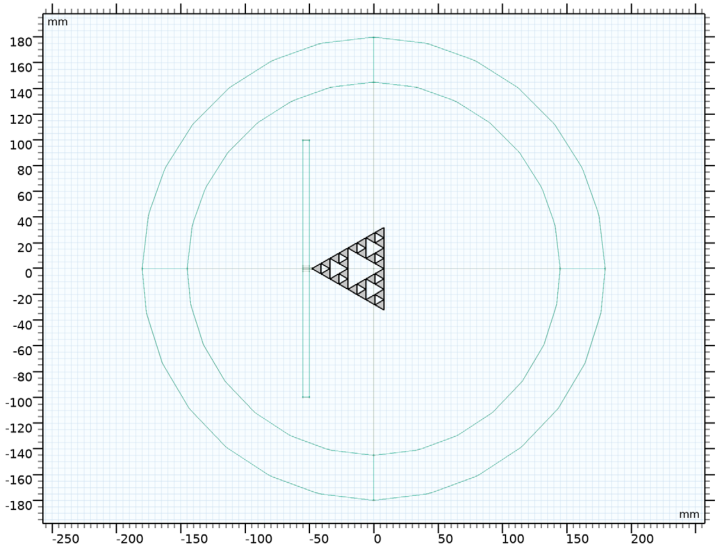
5 In the **yw** text field, type `16,-16`.

Work Plane 1 (wp1)>Union 3 (uni3)

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

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

3 In the **Settings** window for **Union**, click **Build Selected**.



Work Plane 1 (wp1)>Copy 4 (copy4)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **uni3** only.

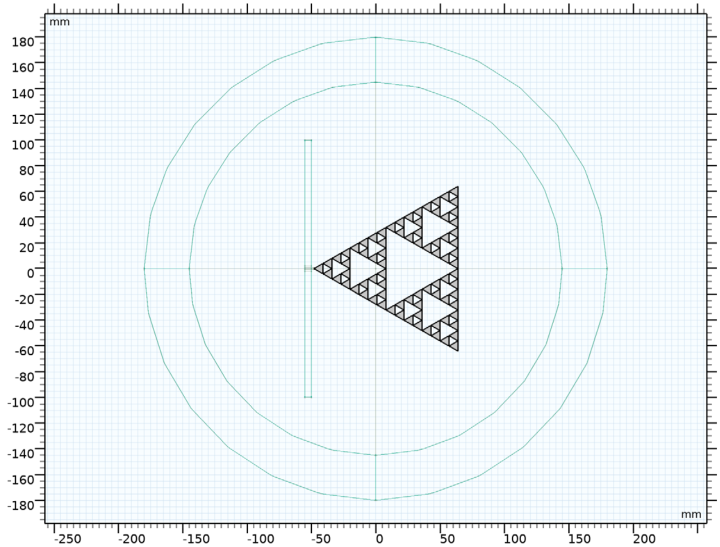
3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **xw** text field, type `56,56`.

5 In the **yw** text field, type `32,-32`.

6 Click the **Zoom Out** button in the **Graphics** toolbar.

7 Click **Build Selected**.



This is the 3rd-order Sierpinski fractal triangle.

Work Plane 1 (wp1)>Plane Geometry

Add a polygon on the 1st triangle. The flat end is preferred to connect the coaxial pin (inner conductor).

Work Plane 1 (wp1)>Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

xw (mm)	yw (mm)
-41	4
-48	1.6
-48	-1.6
-41	-4

Work Plane 1 (wp1)>Union 4 (uni4)

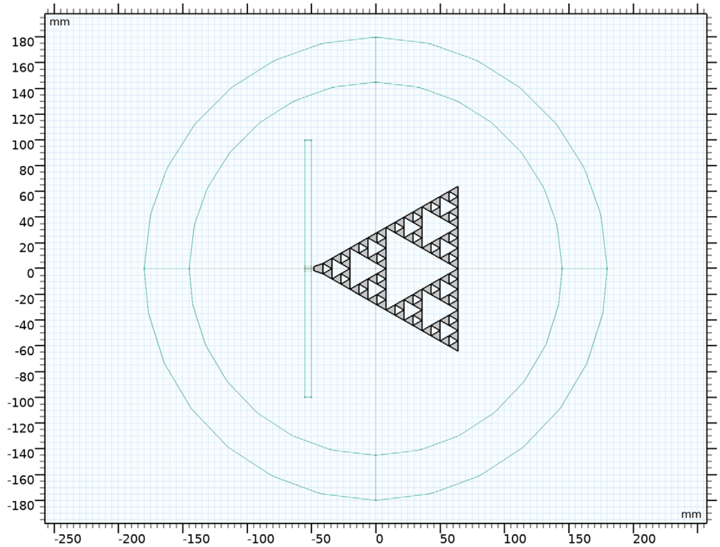
1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Union**.

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

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

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

5 Click **Build Selected**.



Work Plane 1 (wp1)>Plane Geometry

Scale the entire triangle structure to generate equilateral triangles.

Work Plane 1 (wp1)>Scale 1 (sca1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Scale**.

2 Select the object **uni4** only.

3 In the **Settings** window for **Scale**, locate the **Scale Factor** section.

4 From the **Scaling** list, choose **Anisotropic**.

5 In the **xw** text field, type `1/1.75*sqrt(3)`.

6 Locate the **Center of Scaling** section. In the **xw** text field, type `-48`.

Extrude 1 (ext1)

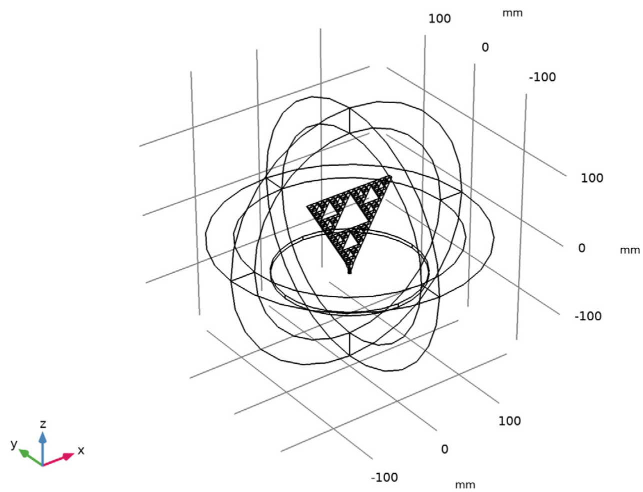
1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (mm)
thickness

4 Click **Build All Objects**.



The finished geometry represents the Sierpinski fractal monopole antenna inside the PML.

DEFINITIONS

Create a set of selections before setting up the physics. Add a selection for the domains that are not part of the model analysis. These are the inside domains of the fractal radiator and ground plane. Because these parts are modeled as PECs, the inside does not need to be analyzed.

Excluded

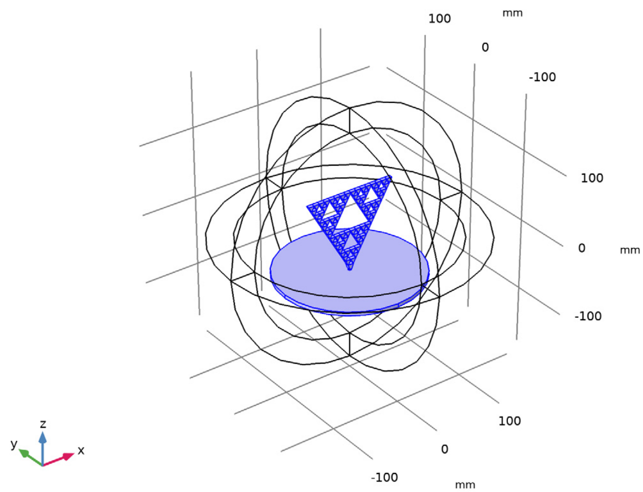
1 In the **Definitions** toolbar, click **Explicit**.

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

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `6-47, 49-50, 53-56, 59-94` in the **Selection** text field.

5 Click **OK**.



Add a selection for the model domain. Use the complement of the **Excluded** domain.

Model space

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

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

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

4 In the **Add** dialog box, select **Excluded** in the **Selections to invert** list.

5 Click **OK**.

Perfectly Matched Layer 1 (pml1)

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

2 Select Domains 1–4, 51, 52, 57, and 58 only.

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

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

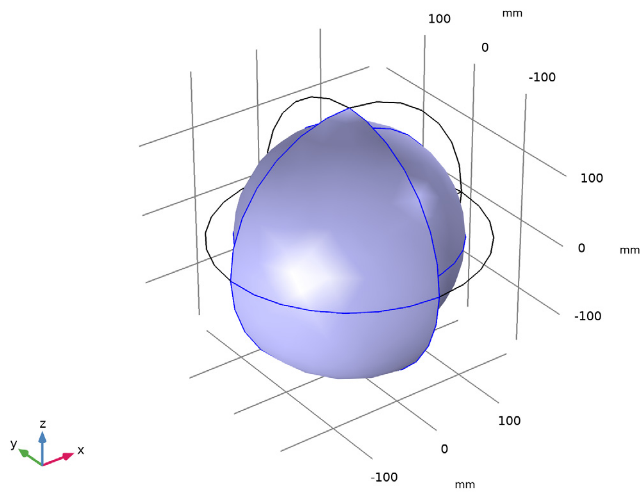
View 1

Suppress three domains to get a better view when setting up the physics and reviewing the meshed results.

Hide for Physics 1

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

2 Select Domains 1, 2, and 5 only.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. Use the model domain selection for the current physics. The boundaries of excluded domains will be set to perfect electric conductor by default.

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 **Model space**.

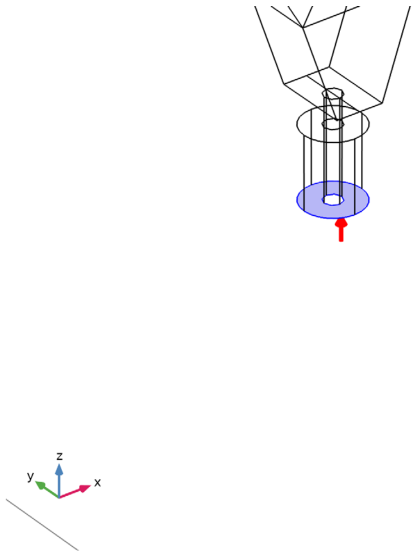
Locate the coaxial lumped port on the bottom of the outer conductor. Zoom in a couple of times to get a clear view of the coaxial cable.

Lumped Port 1

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

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

3 Select Boundary 223 only.



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

5 From the **Type of lumped port** list, choose **Coaxial**.

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

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

Far-Field Domain 1

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

MATERIALS

Assign material properties. First, use air for all domains.

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

Override the substrate with the dielectric material of $\varepsilon_r = 2.1$.

PTFE

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 PTFE in the **Label** text field.

3 Select Domain 48 only.

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

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

MESH 1

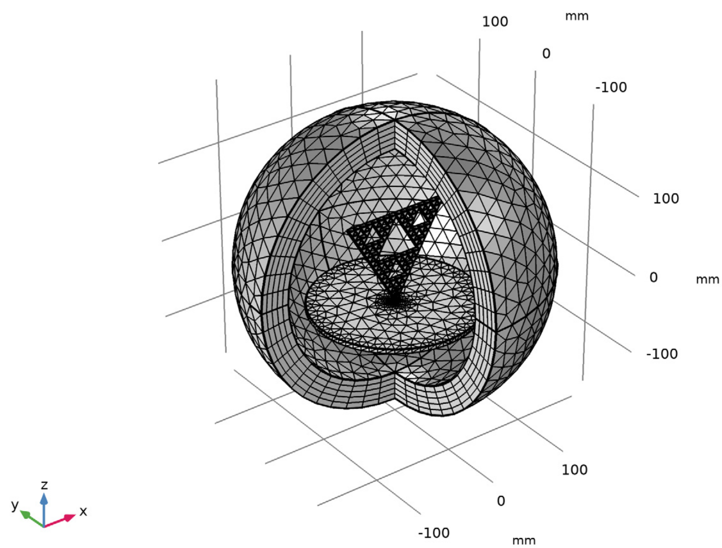
1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

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

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.

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



STUDY 1

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

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

3 Clear the **Generate default plots** check box.

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

RESULTS

Begin the result analysis by evaluating the S-parameters.

Global Evaluation 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results>Derived Values** and choose **Global Evaluation**.

3 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Ports>emw.S11dB - S11**.

4 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

The calculated S-parameters for both frequencies are less than -10 dB.

Add a surface plot for the E-field norm. Before adding the surface plot, add a selection on the solution to specify the visualization area.

RESULTS

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

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 From the **Selection** list, choose **Excluded**.

3D Plot Group 1

1 In the **Results** toolbar, click **3D Plot Group**.

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

3 From the **Parameter value (freq (GHz))** list, choose **1.6**.

Surface 1

1 Right-click **3D Plot Group 1** and choose **Surface**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Maximum** text field, type 150.

5 In the **3D Plot Group 1** toolbar, click **Plot**.

Compare the plot with Figure 2.

3D Plot Group 2

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Clear the **Plot dataset edges** check box.

4 Locate the **Data** section. From the **Parameter value (freq (GHz))** list, choose **1.6**.

Radiation Pattern 1

1 In the **3D Plot Group 2** toolbar, click **More Plots** and choose **Radiation Pattern**.

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

3 Find the **Angles** subsection. In the **Number of elevation angles** text field, type 60.

4 In the **Number of azimuth angles** text field, type 60.

5 In the **3D Plot Group 2** toolbar, click **Plot**.

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

Compare this plot with that shown in Figure 3.