# Dipole Antenna with a Quarter-Wave Coaxial Balun

## *Introduction*

An antenna balun blocks undesirable currents on the outside of a cable. A quarter-wave coaxial balun (1:1) is one of the easiest designs: it works as a transformer and makes the input impedance from the inner conductor to the outer conductor infinite. This model simulates an antenna prototype including an SMA Receptacle, an ABS enclosure, and a quarter-wave coaxial balun. The frequency is tuned to GPS range, 1.57542 GHz. Such an antenna can be useful for indoor measurements with a single polarization.

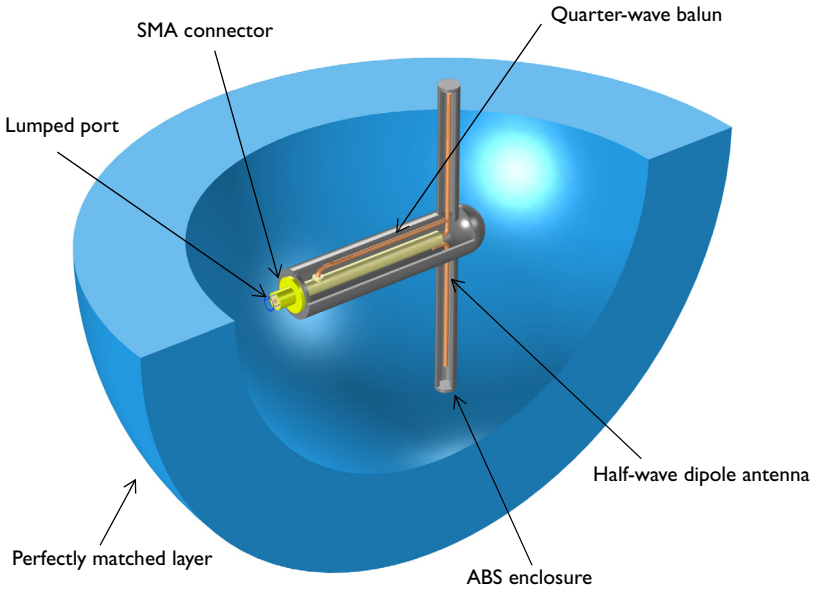



*Figure 1: A quarter-wave balun is added on a half-wave dipole antenna with an ABS enclosure. The entire domain is bounded by a perfectly matched layer.*

## *Model Definition*

The antenna model shown in Figure 1 is composed of a half-wave dipole antenna connected to a coaxial cable, a SMA receptacle (connector), an antenna enclosure, and a perfectly matched layer (PML). Model all metal parts as perfect electric conductors (PEC). The space between the inner and outer conductors of the coaxial cable is filled with polytetrafluoroethylene (PTFE), $\varepsilon_r = 2.1$. Furthermore, model the antenna enclosure using acrylonitrile butadiene styrene (ABS), a plastic material with $\varepsilon_r = 2.4$. The rest of the model space is air and the entire model domain is enclosed by the PML.

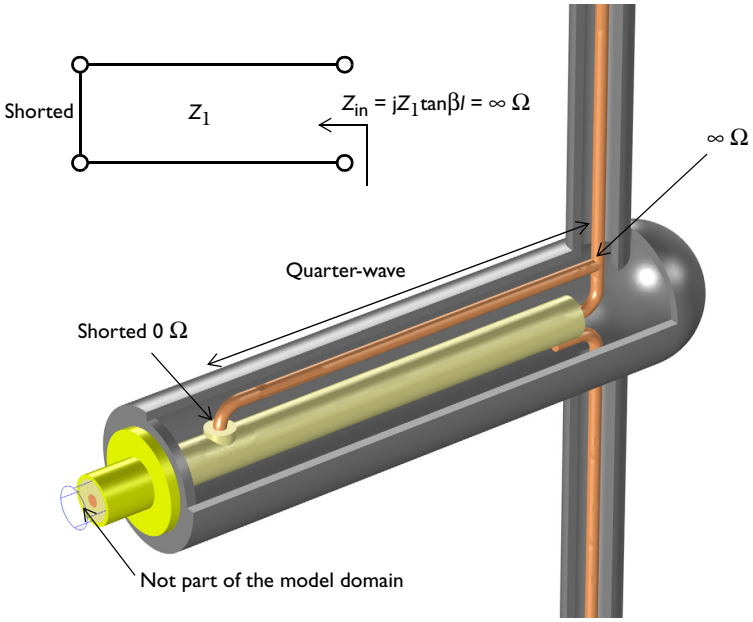



*Figure 2: Zoomed view of the antenna interior shows the configuration of the balun.*

The antenna balun part depicted in Figure 2 can be analyzed using the transmission line theory. The input impedance of the short-circuited transmission line is

$$Z_{\text{in}} = jZ_1 \tan\beta l \tag{1}$$

This can also be calculated via the quarter-wave transformer analysis:

$$Z_{\text{in}} = Z_1^2 / Z_l \tag{2}$$

The input impedance of the short-circuited quarter-wave length transmission line is calculated as infinite from both equations. This infinite input impedance prevents a current flow or coupling from the antenna radiator to the outer conductor of the coaxial cable. Thus, the antenna radiators connected to the inherently unbalanced coaxial cable can be balanced without generating the current on an undesired location.

## *Results and Discussion*

The default plots after the simulation show the norm of the electric fields, 2D far-field polar plot, and 3D far-field radiation pattern. The symmetric and balanced E-fields excited on each antenna radiator are observed in Figure 3.

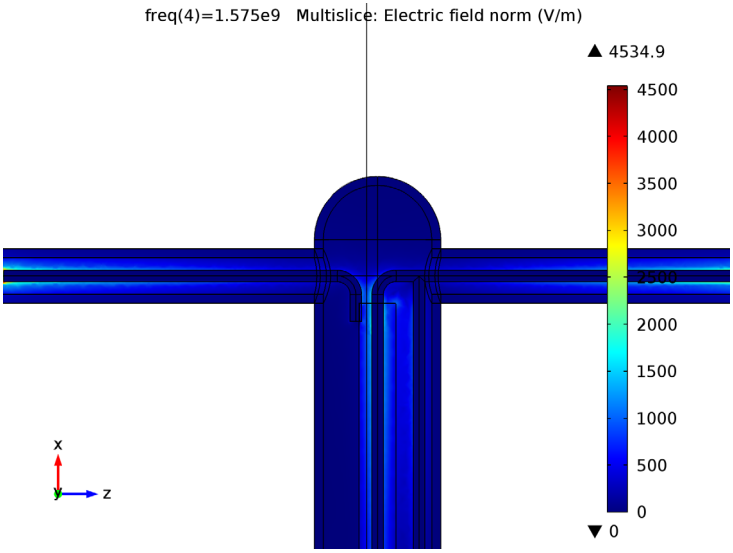



*Figure 3: The E-field norm on each antenna radiator looks symmetric.*

Figure 4 shows the far-field radiation pattern on the H-plane. The pattern looks almost isotropic and omni-directional. The magnitude of the far-field is slightly decreased around 180° where the body of the antenna enclosure and connector are located. The 3D far-field radiation pattern is plotted in Figure 5 and it is very like that of a conventional half-wave dipole antenna.

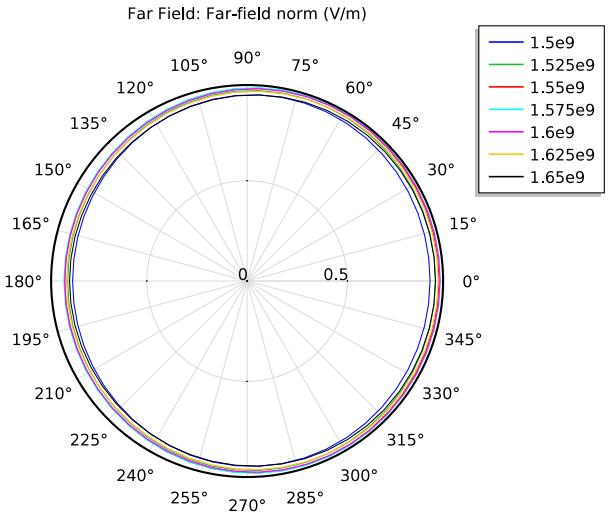



*Figure 4: The far-field pattern on the H-plane is isotropic as expected.*

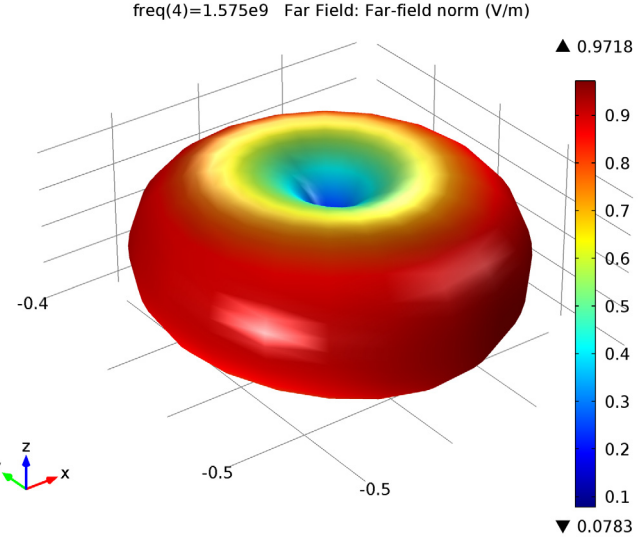



*Figure 5: The visualized 3D far-field pattern is a little bit distorted due to the balun and the antenna enclosure.*

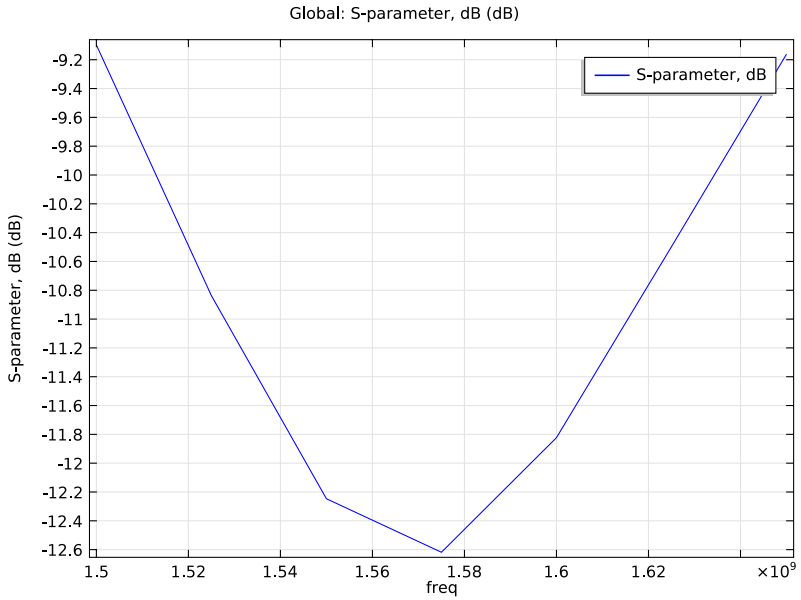



*Figure 6: The S-parameter plot shows the antenna is resonant around the GPS frequency, 1.57542 GHz.*

The calculated S-parameters plotted in Figure 6 are less than –10 dB around the GPS frequency range.

## *References*

1. D.M. Pozar, *Microwave Engineering*, Wiley, 1998

2. C.A. Balanis, *Antenna Theory*, Wiley, 1997.

**Model Library path:** `RF_Module/Antennas/dipole_antenna_balun`

## *Modeling Instructions*

### MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

6 Click **Finish**.

## GLOBAL 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 |
|---|---|---|
| l_wire | 41[mm] | Wire length |
| r_inner | 0.635[mm] | Radius, coax inner |
| r_outer | 2.05[mm] | Radius, coax outer |
| f_min | 1.5[GHz] | Minimum frequency in sweep |
| f_max | 1.65[GHz] | Maximum frequency in sweep |
| lda_min | c_const/f_max | Minimum wavelength in sweep, air |
| h_max | 0.2*lda_min | Maximum mesh size, air |

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

## GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

First, import the antenna enclosure structure.

*Import 1*

1 Right-click **Model 1>Geometry 1** and choose **Import**.

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

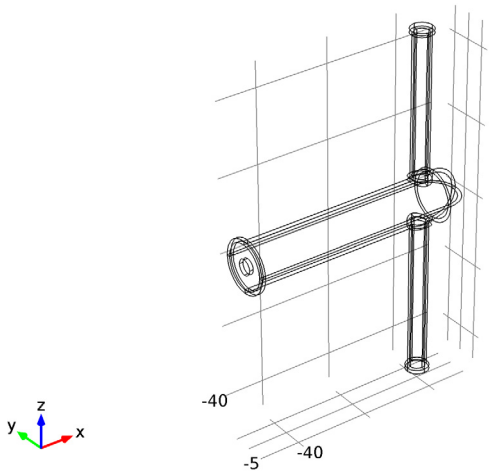
3 Click the **Browse** button.

4 Browse to the model's Model Library folder and double-click the file `dipole_antenna_balun.mphbin`.

5 Click the **Import** button.

Choose wireframe rendering to see the interior of the enclosure.

6 Click the **Wireframe Rendering** button on the Graphics toolbar.




Add a cylinder for the inner conductor of the coaxial cable.

*Cylinder 1*

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `r_inner`.

4 In the **Height** edit field, type `54.95`.

5 Locate the **Position** section. In the **x** edit field, type `-57`.

6 In the **z** edit field, type `1.2`.

7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

8 Right-click **Model 1>Geometry 1>Cylinder 1** and choose **Rename**.

9 Go to the **Rename Cylinder** dialog box and type `Coax inner` in the **New name** edit field.

10 Click **OK**.

Add a cylinder for the outer conductor for the coaxial cable.

*Cylinder 2*

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `r_outer`.

4 In the **Height** edit field, type `54`.

5 Locate the **Position** section. In the **x** edit field, type `-57`.

6 In the **z** edit field, type `1.2`.

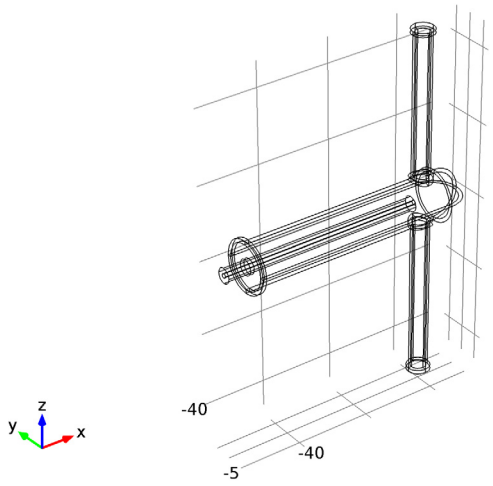
7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

8 Right-click **Model 1>Geometry 1>Cylinder 2** and choose **Rename**.

9 Go to the **Rename Cylinder** dialog box and type `Coax outer` in the **New name** edit field.

10 Click **OK**.

11 Right-click **Model 1>Geometry 1>Cylinder 2** and choose **Build Selected**.




Add a one-quarter of a torus for the 90-degree bend part connecting the antenna radiator and the inner conductor of the coaxial cable.

*Torus 1*

1 Right-click **Geometry 1** and choose **More Primitives>Torus**.

2 In the **Torus** settings window, locate the **Size and Shape** section.

3 In the **Major radius** edit field, type `r_outer`.

4 In the **Minor radius** edit field, type `r_inner`.

5 In the **Revolution angle** edit field, type `90`.

6 Locate the **Position** section. In the **x** edit field, type `-r_outer`.

7 In the **z** edit field, type `3.25`.

8 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

9 Locate the **Rotation Angle** section. In the **Rotation** edit field, type `270`.

10 Right-click **Model 1>Geometry 1>Torus 1** and choose **Rename**.

11 Go to the **Rename Torus** dialog box and type `90 degree bend` in the **New name** edit field.

12 Click **OK**.

Add a cylinder for an antenna radiator.

*Cylinder 3*

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `r_inner`.

4 In the **Height** edit field, type `l_wire`.

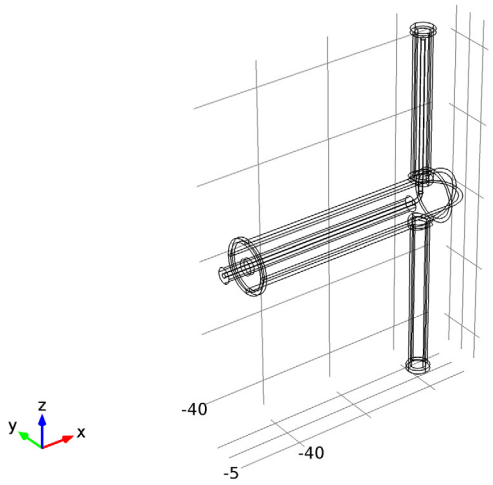
5 Locate the **Position** section. In the **z** edit field, type `3.25`.

6 Right-click **Model 1>Geometry 1>Cylinder 3** and choose **Rename**.

7 Go to the **Rename Cylinder** dialog box and type `Radiator` in the **New name** edit field.

8 Click **OK**.

**9** Right-click **Model 1>Geometry 1>Cylinder 3** and choose **Build Selected**.




Create a second antenna radiator by mirroring one side.

*Mirror 1*

**1** Right-click **Geometry 1** and choose **Transforms>Mirror**.

**2** Select the objects **cyl3** and **tor1** only.

**3** In the **Mirror** settings window, locate the **Input** section.

**4** Select the **Keep input objects** check box.

Add a cylinder to connect another 90 degree bend with the outer conductor of the coaxial cable.

*Cylinder 4*

**1** Right-click **Geometry 1** and choose **Cylinder**.

**2** In the **Cylinder** settings window, locate the **Size and Shape** section.

**3** In the **Radius** edit field, type `r_inner`.

**4** In the **Height** edit field, type `2.95`.

**5** Locate the **Position** section. In the **x** edit field, type `-5`.

**6** In the **z** edit field, type `-1.2`.

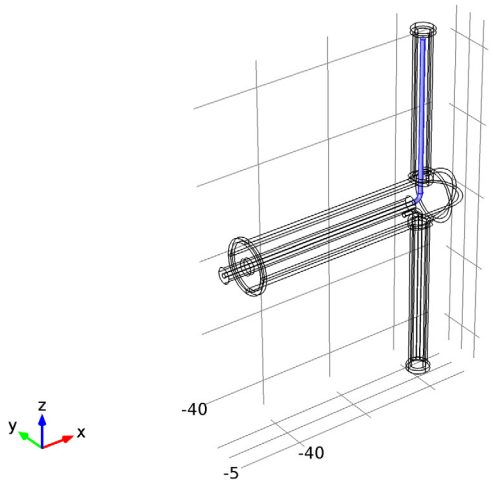
**7** Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

**8** Right-click **Model 1>Geometry 1>Cylinder 4** and choose **Rename**.

9 Go to the **Rename Cylinder** dialog box and type `Solder to ground: radiator` in the **New name** edit field.

10 Click **OK**.

11 Right-click **Model 1>Geometry 1>Cylinder 4** and choose **Build Selected**.




Create the quarter-wave coaxial balun by performing a couple of geometry transform operations.

*Rotate 1*

1 Right-click **Geometry 1** and choose **Transforms>Rotate**.

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

3 In the **Rotate** settings window, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Rotation Angle** section. In the **Rotation** edit field, type `-90`.

6 Locate the **Axis of Rotation** section. From the **Axis type** list, choose **y-axis**.

*Move 1*

1 Right-click **Geometry 1** and choose **Transforms>Move**.

2 Select the objects **rot1(1)** and **rot1(2)** only.

3 In the **Move** settings window, locate the **Displacement** section.

4 In the **x** edit field, type `-l_wire-3.25`.

5 In the **z** edit field, type `5.8`.

Add a cylinder to connect the balun and the outer conductor of the coaxial cable.

*Cylinder 5*

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `1.5`.

4 In the **Height** edit field, type `1.25`.

5 Locate the **Position** section. In the **x** edit field, type `-l_wire-r_outer`.

6 In the **z** edit field, type `2.5`.

7 Right-click **Model 1>Geometry 1>Cylinder 5** and choose **Rename**.

8 Go to the **Rename Cylinder** dialog box and type `Solder to ground: balun` in the **New name** edit field.

9 Click **OK**.

Create unions of some of the objects to remove unnecessary edges that would increase the mesh complexity.

*Union 1*

1 Right-click **Geometry 1** and choose **Boolean Operations>Union**.

2 Select the objects **cyl2** and **cyl5** only.

3 In the **Union** settings window, locate the **Union** section.

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

*Union 2*

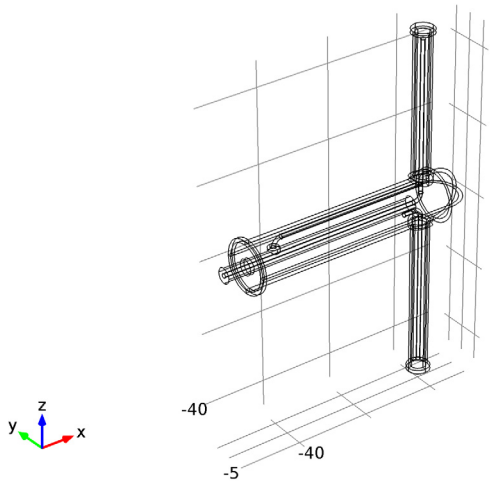
1 Right-click **Geometry 1** and choose **Boolean Operations>Union**.

2 Select the objects **cyl3** and **mov1(1)** only.

3 In the **Union** settings window, locate the **Union** section.

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

5 Click the **Build Selected** button.




Add a couple of cylinders for the SMA connector.

*Cylinder 6*

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `2.45`.

4 In the **Height** edit field, type `5`.

5 Locate the **Position** section. In the **x** edit field, type `-57`.

6 In the **z** edit field, type `1.2`.

7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

8 Right-click **Model 1>Geometry 1>Cylinder 6** and choose **Rename**.

9 Go to the **Rename Cylinder** dialog box and type `SMA` in the **New name** edit field.

10 Click **OK**.

*Cylinder 7*

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `5`.

4 Locate the **Position** section. In the **x** edit field, type `-52`.

5 In the **z** edit field, type `1.2`.

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

7 Right-click **Model 1>Geometry 1>Cylinder 7** and choose **Rename**.

8 Go to the **Rename Cylinder** dialog box and type `SMA flange` in the **New name** edit field.

9 Click **OK**.

Add a cylinder on the back side of the coaxial port. This domain is not a part of the model space.

*Cylinder 8*

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `r_outer`.

4 In the **Height** edit field, type `2`.

5 Locate the **Position** section. In the **x** edit field, type `-59`.

6 In the **z** edit field, type `1.2`.

7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

8 Right-click **Model 1>Geometry 1>Cylinder 8** and choose **Rename**.

9 Go to the **Rename Cylinder** dialog box and type `Connector cap` in the **New name** edit field.

10 Click **OK**.

Add a cylinder to tune the antenna for generating balanced current distributions on each wire radiator.

*Cylinder 9*

1 Right-click **Geometry 1** and choose **Cylinder**.

2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `r_inner`.

4 In the **Height** edit field, type `0.1*l_wire`.

5 Locate the **Position** section. In the **z** edit field, type `-l_wire-3.25`.

6 Right-click **Model 1>Geometry 1>Cylinder 9** and choose **Rename**.

7 Go to the **Rename Cylinder** dialog box and type `Tuning wire` in the **New name** edit field.

8 Click **OK**.

*Difference 1*

1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.

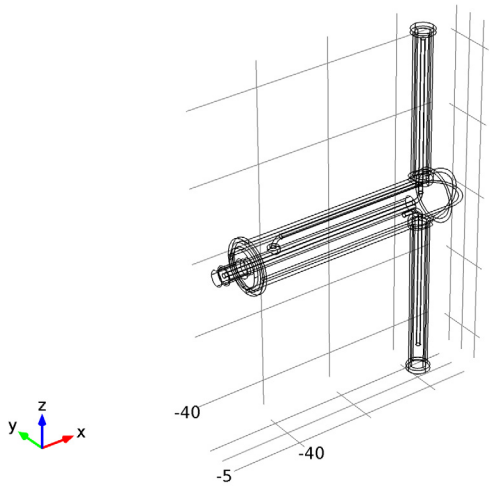
2 Select the object **mir1(1)** only.

3 In the **Difference** settings window, locate the **Difference** section.

4 Under **Objects to subtract**, click **Activate Selection**.

5 Select the object **cyl9** only.

6 Click the **Build Selected** button.




Add a sphere for the PML.

*Sphere 1*

1 Right-click **Geometry 1** and choose **Sphere**.

2 In the **Sphere** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 100.

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

| Layer name | Thickness (mm) |
|---|---|
| Layer 1 | 30 |

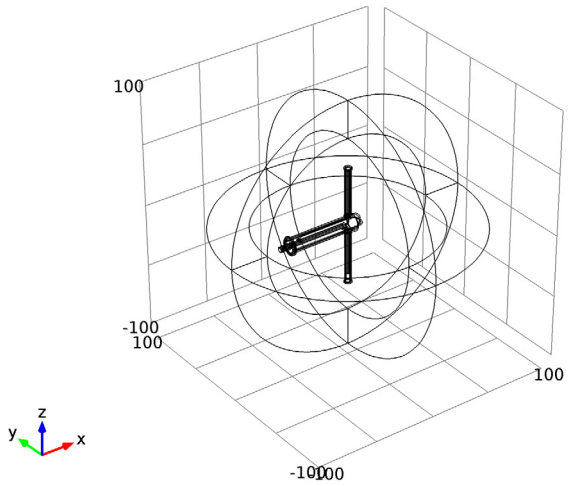
5 Right-click **Sphere 1** and choose **Rename**.

6 Go to the **Rename Sphere** dialog box and type PML in the **New name** edit field.

7 Click **OK**.

8 Click the **Build All** button.

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




10 Click the **Zoom In** button on the Graphics toolbar two or three times.

## DEFINITIONS

Create a set of selections for use before setting up the physics. First, create selections for the conductor boundaries.

*Explicit 1*

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

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

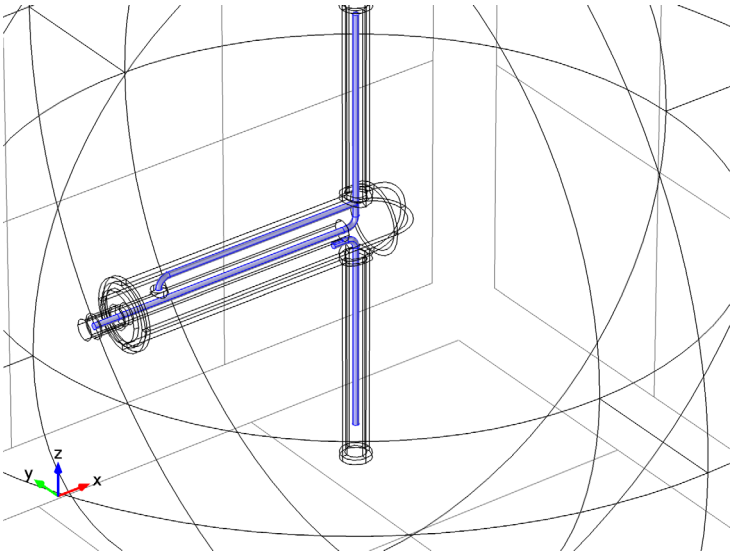
3 From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 Go to the **Paste Selection** dialog box.

6 In the **Selection** edit field, type `24-26, 29-30, 41-42, 45-46, 62-63, 69-70, 102-103, 105-106, 109-117, 119-120, 122, 125-127, 143-144, 161-162, 164-169, 180-182, 184-185, 187-189, 217-218, 227-228`.

**7** Click the **OK** button.




**8** Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.

**9** Go to the **Rename Explicit** dialog box and type `Coax inner and radiator boundaries` in the **New name** edit field.

**10** Click **OK**.

*Explicit 2*

**1** Right-click **Definitions** and choose **Selections>Explicit**.

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

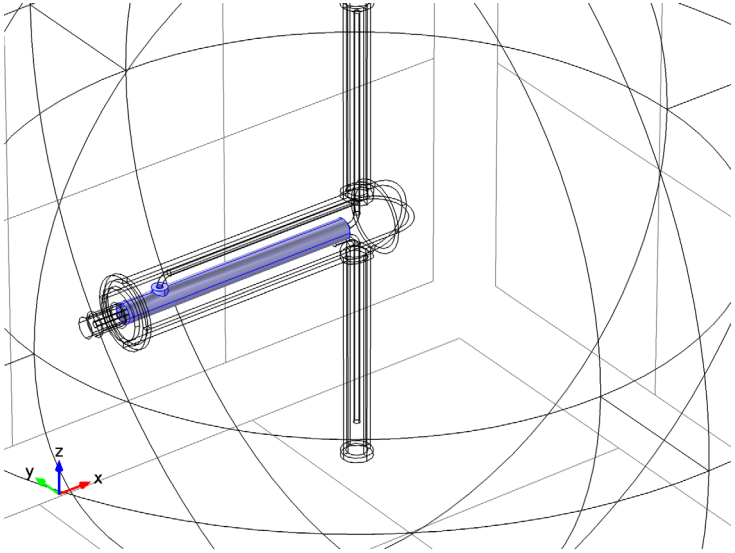
**3** From the **Geometric entity level** list, choose **Boundary**.

**4** Click **Paste Selection**.

**5** Go to the **Paste Selection** dialog box.

**6** In the **Selection** edit field, type `59-60, 68, 71, 85-86, 91-92, 98-100, 104, 107`.

**7** Click the **OK** button.




**8** Right-click **Model 1>Definitions>Explicit 2** and choose **Rename**.

**9** Go to the **Rename Explicit** dialog box and type `Coax outer boundaries` in the **New name** edit field.

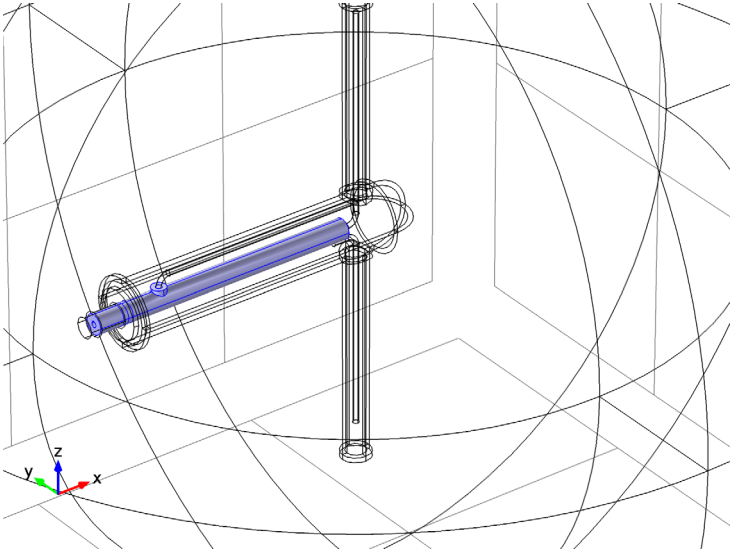
**10** Click **OK**.

Add a selection for the dielectric material inside the coaxial cable.

*Explicit 3*

**1** Right-click **Definitions** and choose **Selections>Explicit**.

2 Select Domains 8, 11, 16, and 28 only.




3 Right-click **Model 1>Definitions>Explicit 3** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `Coax dielectric` in the **New name** edit field.

5 Click **OK**.

Add a selection for the SMA connector.

*Explicit 4*

1 Right-click **Definitions** and choose **Selections>Explicit**.

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

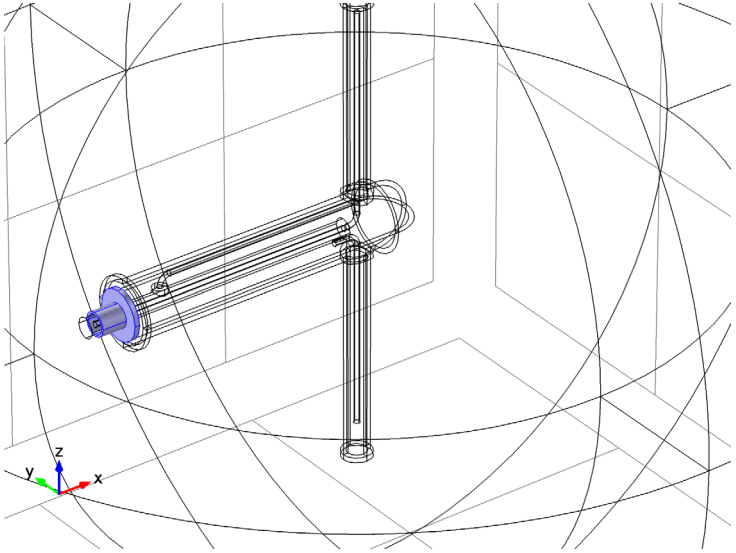
3 From the **Geometric entity level** list, choose **Boundary**.

4 Click **Paste Selection**.

5 Go to the **Paste Selection** dialog box.

6 In the **Selection** edit field, type `18-20, 22-23, 27-28, 31-35, 38-39, 43-44, 47-48, 57`.

**7** Click the **OK** button.




**8** Right-click **Model 1>Definitions>Explicit 4** and choose **Rename**.

**9** Go to the **Rename Explicit** dialog box and type `SMA boundaries` in the **New name** edit field.

**10** Click **OK**.

Add a selection for the antenna enclosure.

*Explicit 5*

**1** Right-click **Definitions** and choose **Selections>Explicit**.

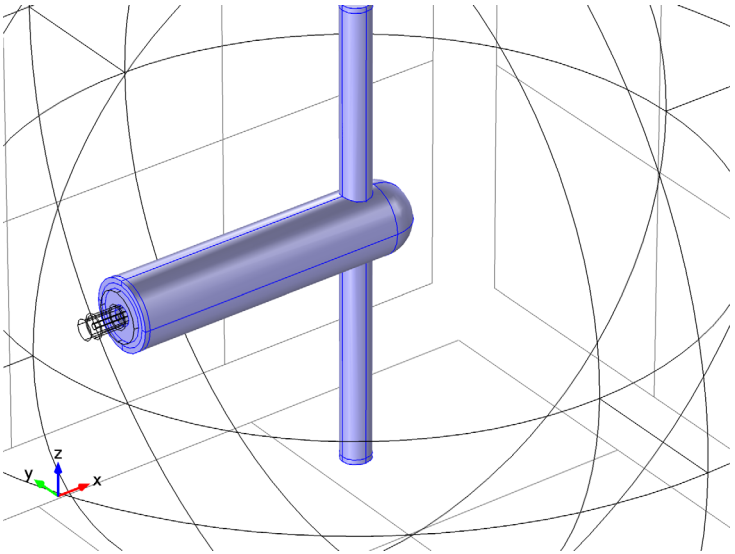
**2** In the **Explicit** settings window, locate the **Input Entities** section.

**3** Click **Paste Selection**.

**4** Go to the **Paste Selection** dialog box.

**5** In the **Selection** edit field, type `13-15, 18-21, 23-24, 30-39, 45-48, 51-58`.

**6** Click the **OK** button.




**7** Right-click **Model 1>Definitions>Explicit 5** and choose **Rename**.

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

**9** Click **OK**.

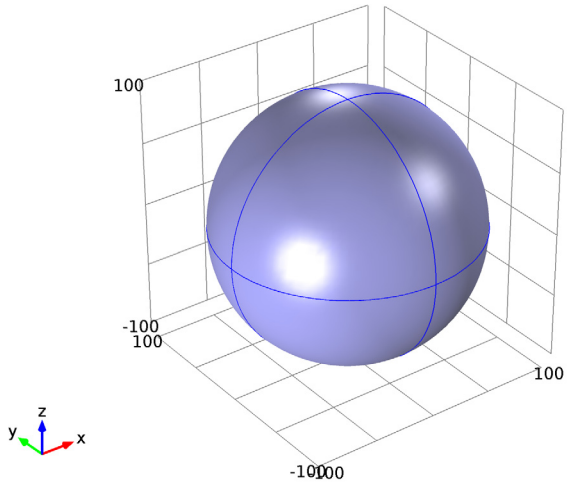
**10** Click the **Zoom Extents** button on the Graphics toolbar.

Add a selection for the PML.

*Explicit 6*

**1** Right-click **Definitions** and choose **Selections>Explicit**.

2 Select Domains 1–4, 43, 44, 49, and 50 only.




3 Right-click **Model 1>Definitions>Explicit 6** and choose **Rename**.

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

5 Click **OK**.

Add a selection for the scattering boundaries. These are the outermost boundaries of the PML.

*Explicit 7*

1 Right-click **Definitions** and choose **Selections>Explicit**.

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

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

4 Select Boundaries 5–8, 193, 194, 221, and 226 only.

5 Right-click **Model 1>Definitions>Explicit 7** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Scattering boundaries` in the **New name** edit field.

7 Click **OK**.

Add a selection for the far-field domain. Use the complement of the PML.

*Complement 1*

1 Right-click **Definitions** and choose **Selections>Complement**.

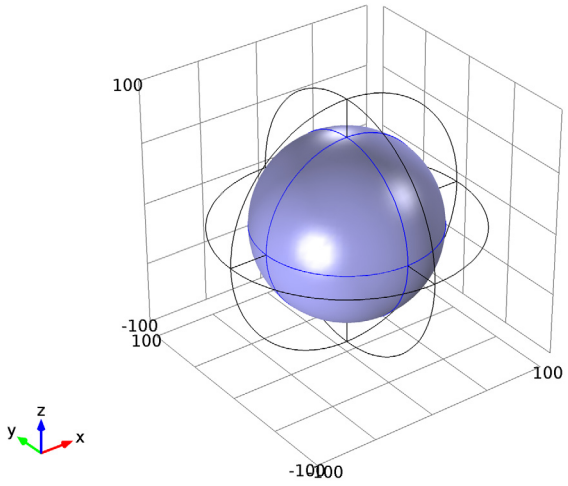
2 In the **Complement** settings window, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 Go to the **Add** dialog box.

5 In the **Selections to invert** list, select **PML**.

6 Click the **OK** button.




7 Right-click **Model 1>Definitions>Complement 1** and choose **Rename**.

8 Go to the **Rename Complement** dialog box and type `Far-field domain` in the **New name** edit field.

9 Click **OK**.

Add a selection for the Far-Field Calculation boundaries. These are the outermost boundaries of the far-field domain.

*Explicit 8*

1 Right-click **Definitions** and choose **Selections>Explicit**.

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

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

4 Select Boundaries 9–12, 195, 196, 222, and 225 only.

5 Right-click **Model 1>Definitions>Explicit 8** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Far-field calc. boundaries` in the **New name** edit field.

7 Click **OK**.

Add a selection for the connector cap.

*Explicit 9*

1 Right-click **Definitions** and choose **Selections>Explicit**.

2 Select Domain 6 only.

3 Right-click **Model 1>Definitions>Explicit 9** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `Coax cap` in the **New name** edit field.

5 Click **OK**.

Add a selection for the model domain. Use the complement of the connector cap selection.

*Complement 2*

1 Right-click **Definitions** and choose **Selections>Complement**.

2 In the **Complement** settings window, locate the **Input Entities** section.

3 Under **Selections to invert**, click **Add**.

4 Go to the **Add** dialog box.

5 In the **Selections to invert** list, select **Coax cap**.

6 Click the **OK** button.

7 Right-click **Model 1>Definitions>Complement 2** and choose **Rename**.

8 Go to the **Rename Complement** dialog box and type `Model domain` in the **New name** edit field.

9 Click **OK**.

Add the PML using the PML selection you defined earlier.

*Perfectly Matched Layer 1*

1 Right-click **Definitions** and choose **Perfectly Matched Layer**.

2 In the **Perfectly Matched Layer** settings window, locate the **Domain Selection** section.

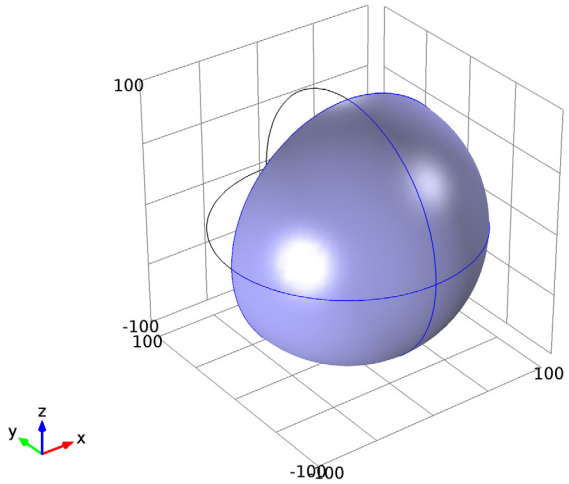
3 From the **Selection** list, choose **PML**.

4 Locate the **Geometry** section. From the **Type** list, choose **Spherical**.

*View 1*

1 In the **Model Builder** window, under **Model 1>Definitions** right-click **View 1** and choose **Hide Geometric Entities**.

2 Select Domains 1, 2, 5, 43, and 44 only.




## ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Now, set up the physics. Choose an appropriate selection when you assign each boundary condition.

1 In the **Model Builder** window, under **Model 1** click **Electromagnetic Waves, Frequency Domain**.

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

3 From the **Selection** list, choose **Model domain**.

4 Click the **Zoom In** button on the Graphics toolbar two or three times.

*Perfect Electric Conductor 2*

1 Right-click **Model 1>Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.

2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Coax inner and radiator boundaries**.

*Perfect Electric Conductor 3*

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.

2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Coax outer boundaries**.

*Perfect Electric Conductor 4*

1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Electric Conductor**.

2 In the **Perfect Electric Conductor** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **SMA boundaries**.

*Lumped Port 1*

1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Lumped Port**.

2 Select Boundary 21 only.

3 In the **Lumped 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 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Scattering Boundary Condition**.

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

3 From the **Selection** list, choose **Scattering boundaries**.

*Far-Field Domain 1*

1 Right-click **Electromagnetic Waves, Frequency Domain** and choose **Far-Field Domain**.

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

3 From the **Selection** list, choose **Far-field domain**.

*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 **Far-field calc. boundaries**.

## MATERIALS

Assign material properties. Use air for all domains, first.

*Material Browser*

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.

Override the coax dielectric domains with a dielectric material of $\varepsilon_r = 2.1$.

*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 **Coax dielectric**.

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

| Property | Name | Value |
|---|---|---|
| Relative permittivity | epsilonr | 2.1 |
| Relative permeability | mur | 1 |
| Electrical conductivity | sigma | 0 |

5 Right-click **Material 2** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type PTFE in the **New name** edit field.

7 Click **OK**.

Override the antenna enclosure with a dielectric material of $\varepsilon_r = 2.4$.

*Material 3*

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

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

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

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

| Property | Name | Value |
|---|---|---|
| Relative permittivity | epsilonr | 2.4 |
| Relative permeability | mur | 1 |
| Electrical conductivity | sigma | 0 |

5 Right-click **Material 3** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type ABS in the **New name** edit field.

7 Click **OK**.

### MESH 1

Choose the maximum mesh size in the air domain smaller than 0.2 wavelengths using the parameter `h_max` that you defined in parameters. For the dielectric materials, scale the mesh size by the inverse of the square root of the relative dielectric constant.

*Size 1*

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

2 In the **Size** settings window, locate the **Element Size** section.

3 Click the **Custom** button.

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

5 In the associated edit field, type `h_max`.

6 Select the **Maximum element growth rate** check box.

7 In the associated edit field, type `1.8`.

*Size 2*

1 In the **Model Builder** window, right-click **Mesh 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 **Coax 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 `h_max/sqrt(2.1)`.

*Size 3*

1 Right-click **Mesh 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 **Enclosure**.

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 `h_max/sqrt(2.4)`.

8 Select the **Minimum element size** check box.

9 In the associated edit field, type `1`.

Specify the minimum mesh size to get the 2nd-order polynomials on the thin wires.

*Size 4*

1 Right-click **Mesh 1** and choose **Size**.

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

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

4 From the **Selection** list, choose **Coax inner and radiator boundaries**.

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

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

7 In the associated edit field, type `0.8`.

*Size 5*

1 Right-click **Mesh 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 Select Domains 6 and 7 only.

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

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

7 In the associated edit field, type `1`.

*Free Tetrahedral 1*

1 Right-click **Mesh 1** and choose **Free Tetrahedral**.

2 In the **Free Tetrahedral** settings window, locate the **Domain Selection** section.

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

4 From the **Selection** list, choose **Far-field domain**.

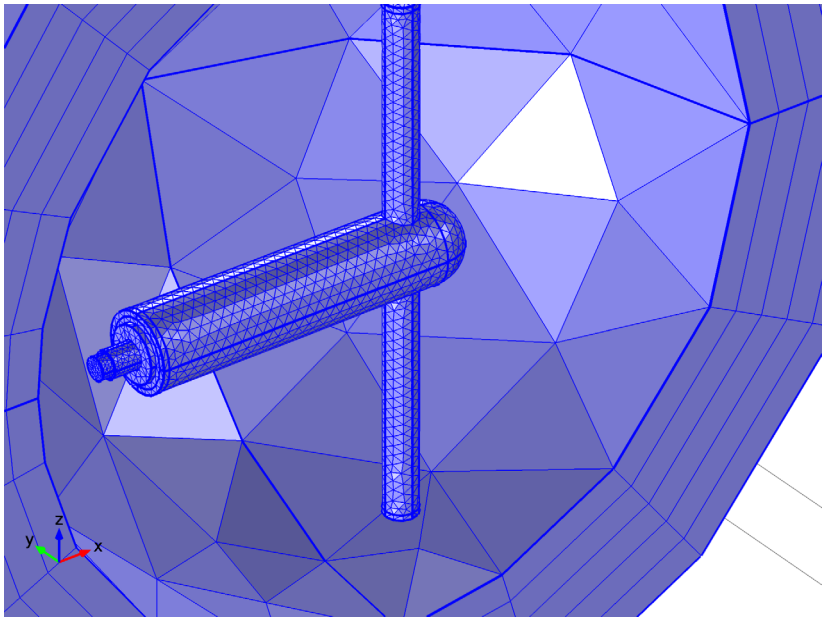
*Swept 1*

Right-click **Mesh 1** and choose **Swept**.

*Distribution 1*

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window, click **Build All**.




## 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(f_min,0.025[GHz],f_max)`.

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

## RESULTS

The default plot shows the E-field norm, 2D far-field polar plot, and 3D far-field radiation pattern. Switch the parameter value to the center frequency in the sweep to plot the results on the resonant frequency.

### *Electric field*

1 In the **Model Builder** window, under **Results** click **Electric field**.

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

3 From the **Parameter value (freq)** list, choose **1.575e9**.

4 Click the **Plot** button.

5 Click the **Go to ZX View** button on the Graphics toolbar.

6 Click the **Zoom In** button on the Graphics toolbar two or three times.

The plot shows the symmetric E-field norm distribution on the wires; compare with Figure 3.

### *Polar Plot Group 2*

Adjust the axis range.

1 In the **Model Builder** window, under **Results** click **Polar Plot Group 2**.

2 In the **Polar Plot Group** settings window, click to expand the **Axis** section.

3 Select the **Manual axis limits** check box.

4 In the ***r* minimum** edit field, type `0`.

5 Click the **Plot** button.

The plotted H-plane patterns are almost isotropic as shown in Figure 4.

### *3D Plot Group 3*

1 In the **Model Builder** window, under **Results** click **3D Plot Group 3**.

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

3 From the **Parameter value (freq)** list, choose **1.575e9**.

4 Click the **Plot** button.

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

6 Click the **Go to Default 3D View** button on the Graphics toolbar.

The visualized 3D radiation pattern looks like the radiation pattern of a typical dipole antenna. Compare the reproduced plot with Figure 5.

Finish the result analysis by evaluating the S-parameters of the antenna.

*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, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB (emw.S11dB)**.

4 Click the **Plot** button.

Compare the result with the S-parameter plot in Figure 6.