Created in COMSOL Multiphysics 6.1

Transient Analysis of a Printed Dual-Band Strip Antenna

Introduction

A wideband antenna study, such as an S-parameter or far-field pattern analysis, can be obtained by performing a transient response analysis and a time-to-frequency fast Fourier transform (FFT). This model runs a time dependent study first and then transforms the dependent variable, the magnetic vector potential A, and a voltage signal at a lumped port from time domain to frequency domain. S-parameters and far-field radiation data are generated from the frequency domain data. The computed S-parameters shows two resonance in the given frequency range, as expected for this dual-band antenna design.

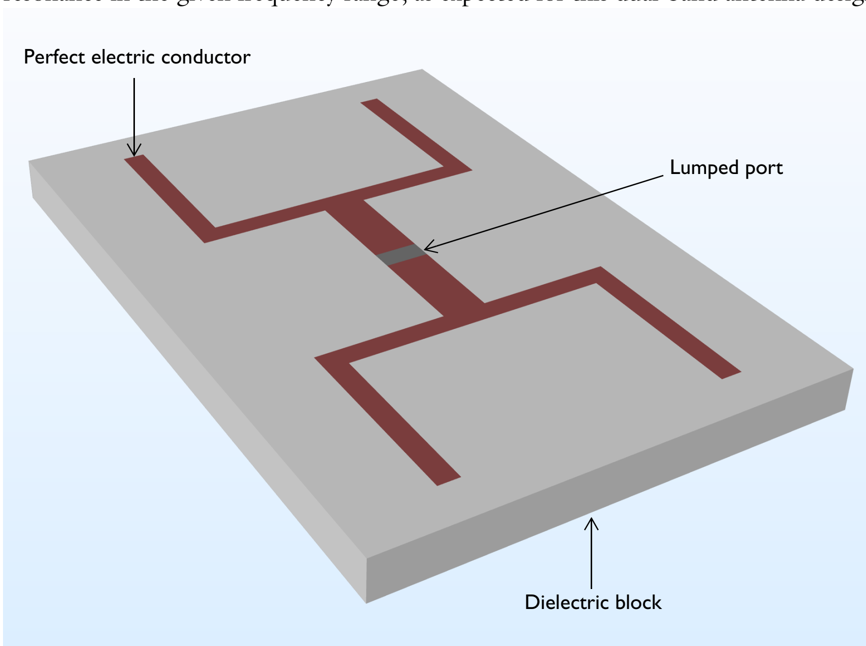


Figure 1: Printed dual-band antenna strip. The surrounding air domain is not included for visualization purposes.

Model Definition

A metallic strip modeled as perfect electric conductor (PEC) is patterned on a dielectric block. A low permittivity value of $\varepsilon_r = 1.05$ is used for the dielectric block to describe a polystyrene foam board. A lumped port with 50 Ω characteristic impedance is added on a small rectangular surface in the middle of the strip to excite the antenna structure. The antenna is enclosed by a spherical air domain. The exterior surfaces of the air are finished by a scattering boundary condition that is an absorbing boundary to describe an open radiating space. On these surfaces, the near field to far field transformation is executed in the frequency domain.

In the lumped port settings window, by clicking the check box “Calculate S-parameter” on the excitation port, the voltage excitation type is set to the modulated Gaussian and the center frequency (f_0) of the modulating sinusoidal function can be specified. The modulated Gaussian excitation voltage is defined as:

$$\frac{1}{\sigma\sqrt{2\pi}}\exp\left(-\frac{\left(t-\frac{2}{f_0}\right)^2}{2\sigma}\right)\sin(2\pi f_0(1+\eta_{\mathrm{f}})t)$$

where σ is the standard deviation $1/2f_0$, f_0 is the center frequency and η_{f} is the modulating frequency shift ratio. A small ratio value, for example 3 %, of η_{f} may enhance the frequency response around the highest frequency.

The frequency here has to be matched to the center frequency of the S-parameter calculation used in the Time to Frequency FFT study step. The end time of the Time Dependent study step is set to 100 times of the period of the modulating sinusoidal function, which is long enough in this model to ensure that the input energy is fully decayed. This would work for a typical passive circuit except for closed cavity type devices, where the energy decay time can be much longer. The stop condition is automatically added under the Time-Dependent solver (the **Calculate S-parameter** check box activates this stop condition in the solver settings). When the sum of total electric and magnetic energy in the modeling domain is less than 70 dB compared to the input energy, the Time Dependent study is terminated by the stop condition and all time-domain data will be passed to the FFT step. To generate the frequency-domain data without significant distortion in the frequency range between 0 and $2f_0$, the time step, satisfying the Nyquist criterion, is set to $1/4f_0 = 1/2B$, where B is the bandwidth $2f_0$.

To provide a fine frequency resolution, the end time of the FFT study step is much longer than that of the Time Dependent study. Zero-padding is applied to the Time Dependent study data before the FFT study step.

Results and Discussion

The electric field norm on top of the dielectric block is plotted at the first antenna resonance frequency in Figure 2. It is observed that the electric field reacts with the entire metallic strip structure when the frequency is around the first resonance. The S-parameters up to 3 GHz in Figure 3 shows two resonance regions where the computed S_{11} is below

-10 dB. While the far-field radiation pattern around the low frequency resonance is isotropic on the H-plane, which resembles that of a half-wave dipole antenna, the radiation characteristic at the high frequency resonance shows a higher directivity toward the up and downside of the dielectric block as shown in Figure 4.

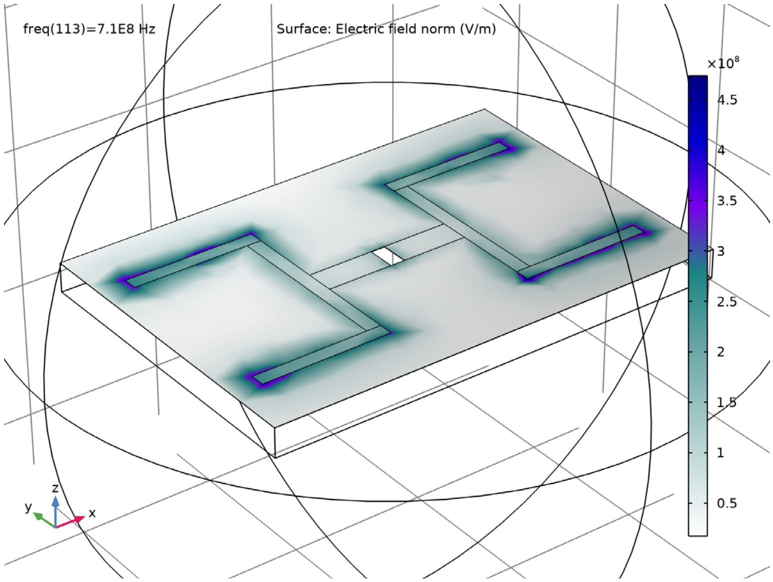


Figure 2: The electric field norm distribution on top of the dielectric board. The resonance behavior is observed at each tip of the printed strip.

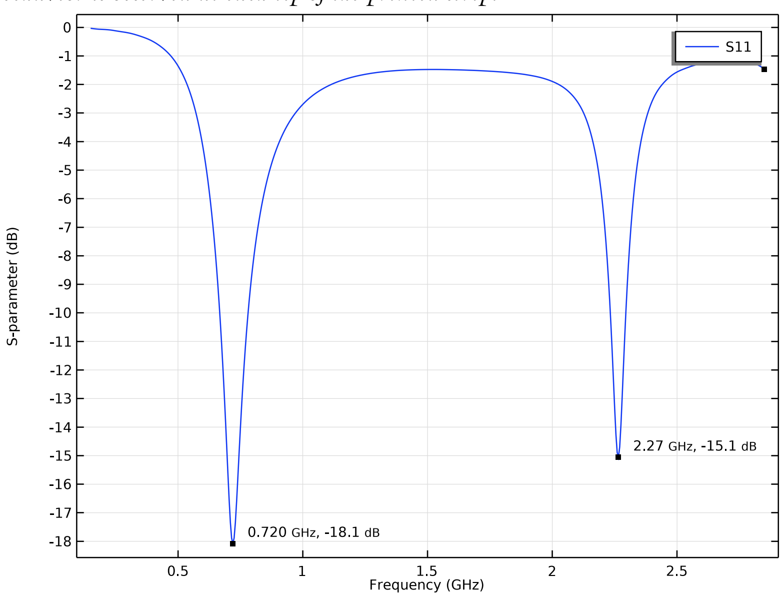
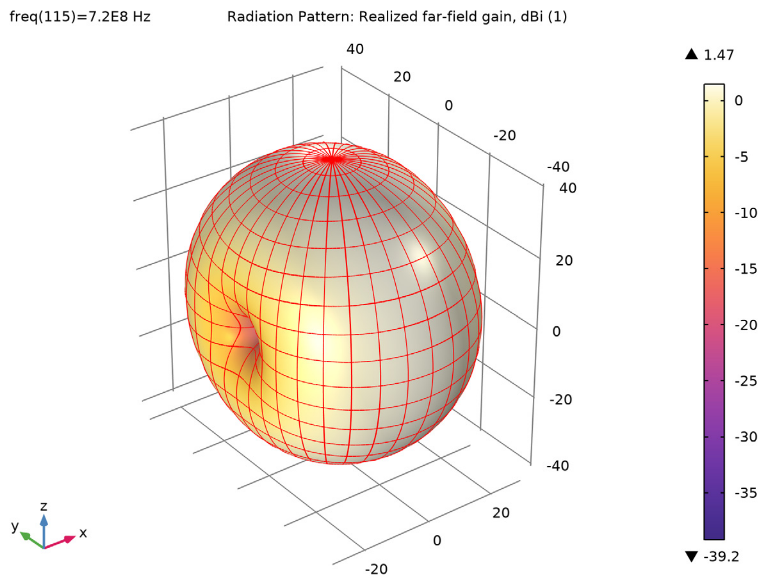


Figure 3: The S-parameter plot describes two resonances where S_{11} is less than -10 dB.



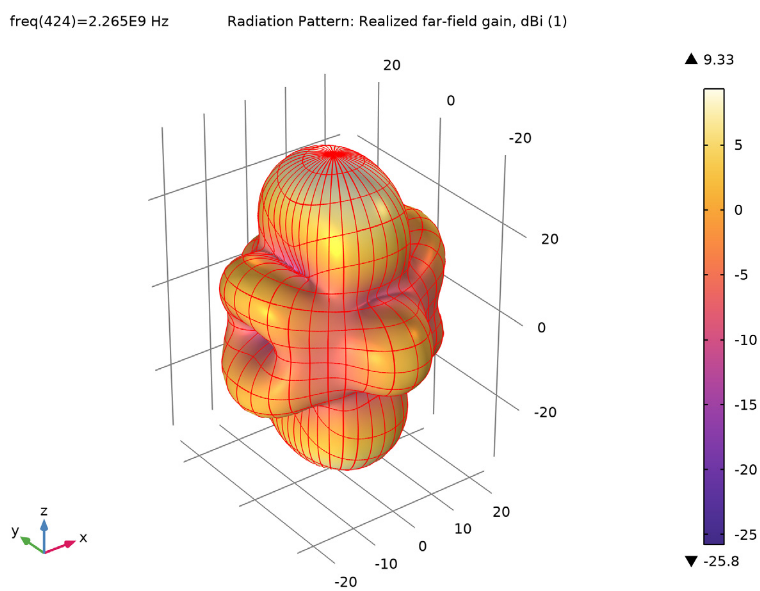


Figure 4: The 3D far-field radiation pattern around the first (top) and second resonance (bottom). The directivity at the second resonance is stronger than that at the first resonance.

Notes About the COMSOL Implementation

The time to frequency fast Fourier transform (FFT) study step transforms the solution of dependent variables in time domain to frequency domain. After running the FFT, only the postprocessing variables that can be expressed with the dependent variables and far-field variables in the frequency domain are valid for the result analysis. Since the solutions are typically transformed with a very small frequency step, it is recommended to use the **Store field in output** option to reduce the size of the model.

Application Library path: `RF_Module/Antennas/dual_band_antenna_transient`

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, Transient (temw)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent with FFT**.

6 Click **Done**.

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
f0	1.5[GHz]	1.5E9 Hz	Center frequency
T0	1/f0	6.6667E-10 s	Period
Tend	100*T0	6.6667E-8 s	End time

GEOMETRY 1

Block 1 (blk1)

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

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

3 In the **Settings** window for **Geometry**, locate the **Units** section.

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

First, create a block for a dielectric layer.

5 In the **Model Builder** window, click **Block 1 (blk1)**.

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

7 In the **Width** text field, type 160.

8 In the **Depth** text field, type 110.

9 In the **Height** text field, type 10.

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

Add a work plane where the printed antenna will be placed.

Work Plane 1 (wp1)

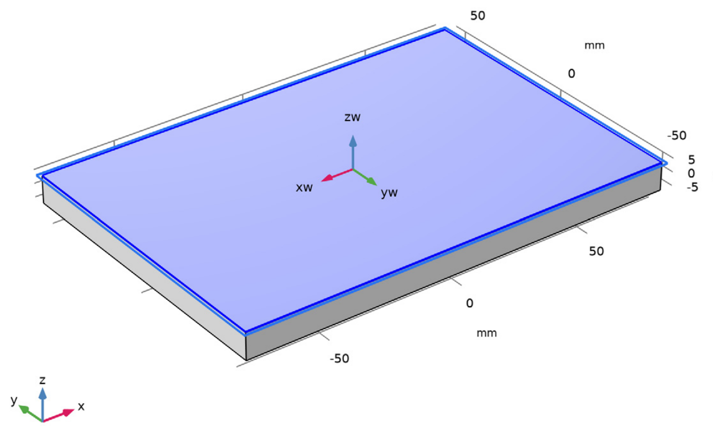
1 In the **Geometry** toolbar, click **Work Plane**.

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

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **blk1**, select Boundary 4 only.

It might be easier to select the correct boundary by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)



Work Plane 1 (wp1)>Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Draw the antenna strip.

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 5.

4 In the **Height** text field, type 10.

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

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 50.

4 In the **Height** text field, type 10.

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

6 Click **Build Selected**.

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 5.

4 In the **Height** text field, type 60.

5 Locate the **Position** section. In the **xw** text field, type -30.

6 In the **yw** text field, type -30.

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 45.

4 In the **Height** text field, type 5.

5 Locate the **Position** section. In the **xw** text field, type -70.

6 In the **yw** text field, type 30.

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.

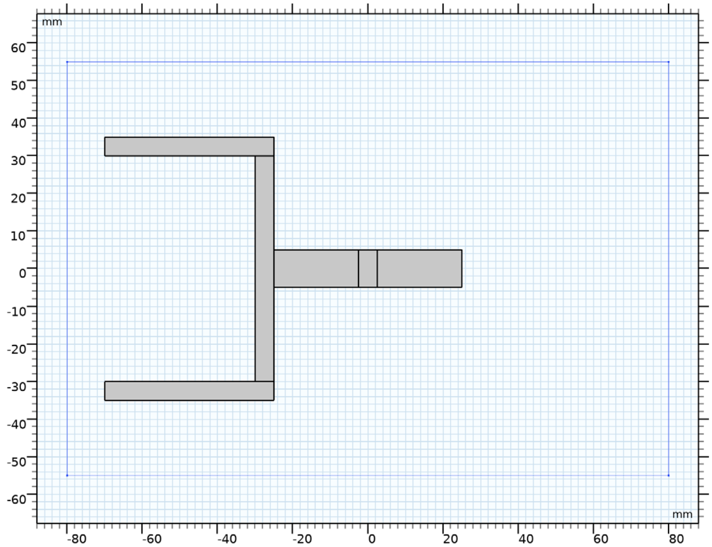
3 In the **Width** text field, type 45.

4 In the **Height** text field, type 5.

5 Locate the **Position** section. In the **xw** text field, type -70.

6 In the **yw** text field, type -35.

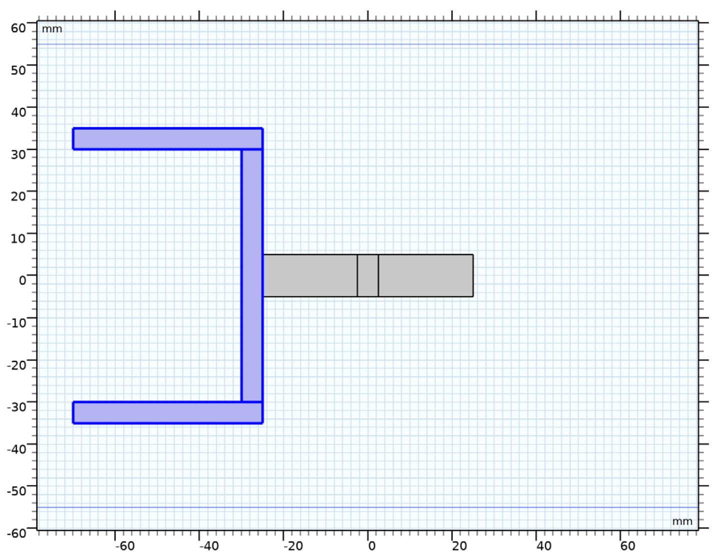
7 Click **Build Selected**.



Work Plane 1 (wp1)>Mirror 1 (mir1)

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

2 Select the objects **r3**, **r4**, and **r5** only.



3 In the **Settings** window for **Mirror**, locate the **Input** section.

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

5 Click **Build Selected**.

Add a sphere for the surrounding air domain.

Sphere 1 (sph1)

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

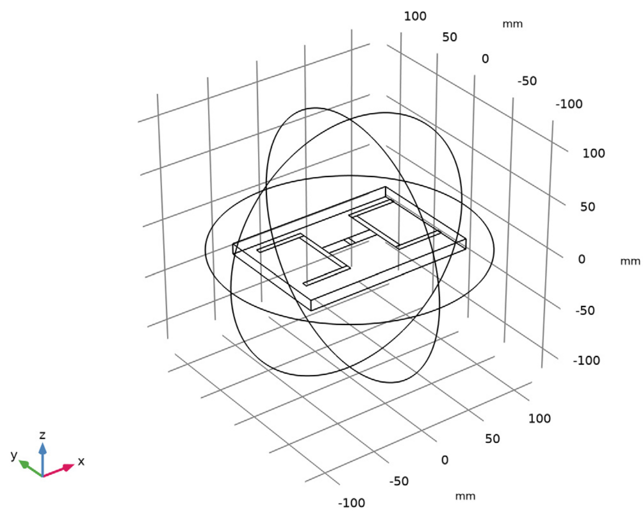
2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type 120.

4 Click **Build All Objects**.

Choose wireframe rendering to get a better view of the interior parts.

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



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.

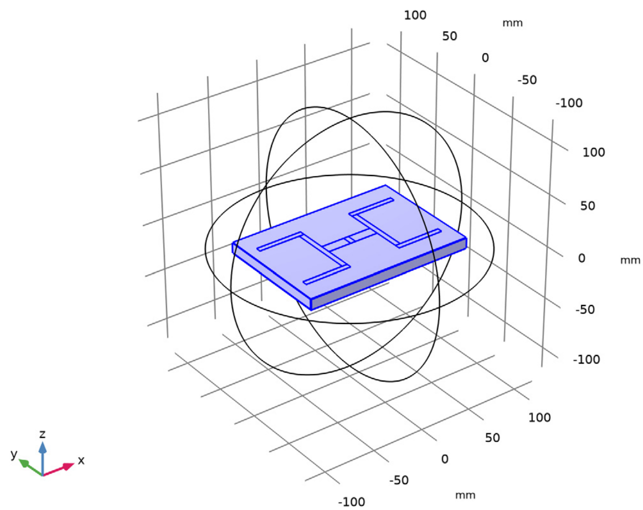
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material 2 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Select Domain 2 only.



3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

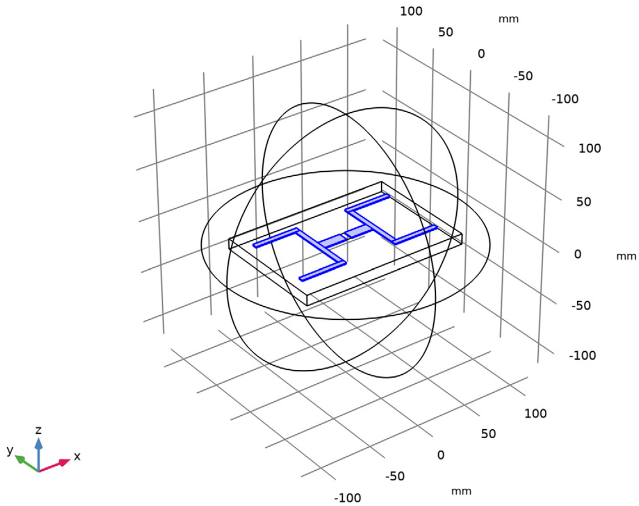
Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1.05	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Perfect Electric Conductor 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Transient (temw)** and choose **Perfect Electric Conductor**.

2 Select Boundaries 10–13 and 19–22 only.

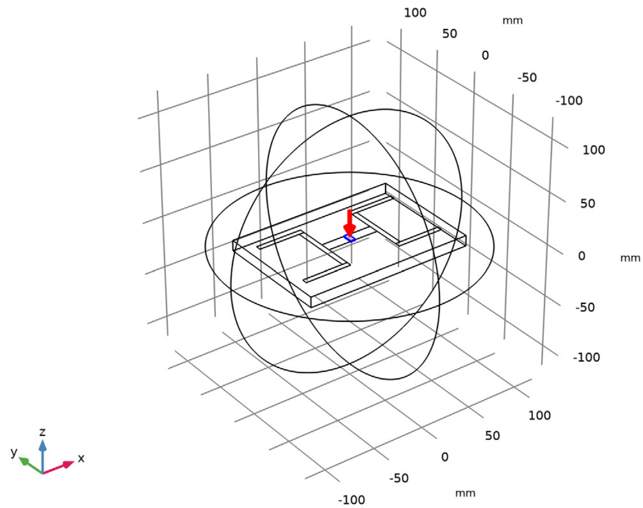


All metallic parts are set to PEC.

Lumped Port 1

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

2 Select Boundary 14 only.



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

4 Select the **Calculate S-parameter** check box.

5 In the f_0 text field, type `f0`.

The center frequency of the modulated Gaussian pulse is f0 that is defined in Parameters node. For the first port, wave excitation is **on** by default.

Far-Field Domain 1

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

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

3 In the list, select **2**.

4 Click **Remove from Selection**.

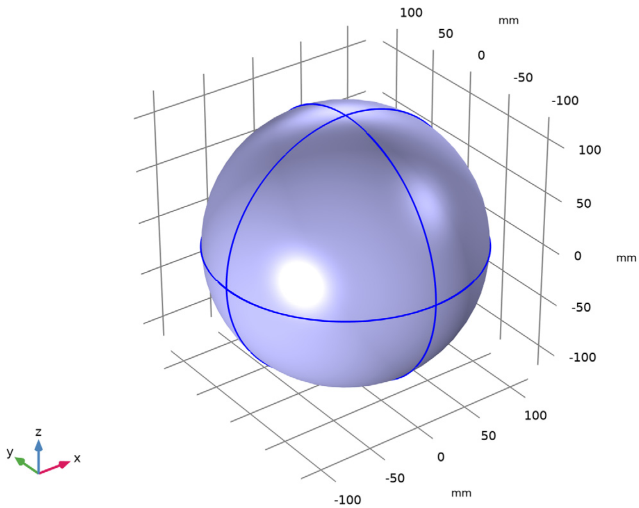
5 Select Domain 1 only.

The far-field domain includes only the air region.

Scattering Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

2 Select Boundaries 1–4 and 15–18 only.



The exterior boundaries is used to absorb the outgoing wave.

MESH 1

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

Hide some boundaries from the view. This helps to see the interior parts when reviewing the mesh.

2 Click the **Click and Hide** button in the **Graphics** toolbar.

3 Select Boundary 2 only.

4 Select Boundary 16 only.

5 Click the **Click and Hide** button in the **Graphics** toolbar.

6 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

7 From the list, choose **User-controlled mesh**.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1** and choose **Size**.

2 Right-click **Size 1** and choose **Move Up**.

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

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

5 Select Boundary 14 only.

Use **Finer** on the lumped port boundary.

6 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

Size

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

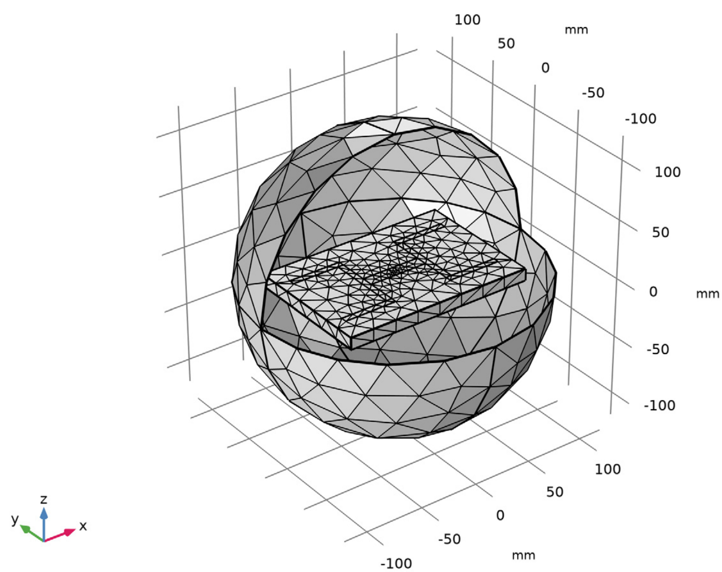
2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `c_const/f0/5`.

The maximum mesh size is 0.2 wavelengths at the center frequency.

5 Click **Build All**.



ELECTROMAGNETIC WAVES, TRANSIENT (TEMW)

Lumped Port 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Transient (temw)** click **Lumped Port 1**.

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

3 Click **Create Selection**.

The added selection will be used to define where to store the results.

4 In the **Create Selection** dialog box, click **OK**.

Far-Field Calculation 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Electromagnetic Waves, Transient (temw)>Far-Field Domain 1** node, then click **Far-Field Calculation 1**.

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

3 Click **Create Selection**.

The added selection will be used to define where to store and visualize the results for the far-field radiation pattern.

4 In the **Create Selection** dialog box, click **OK**.

DEFINITIONS

Explicit 3

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

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

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

4 Select Boundaries 8, 10–13, and 19–22 only.

The model evaluates the solution on these boundaries in time and frequency domain.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,1/(4*f0),Tend)`. The Sampling rate `4*f0` satisfies the Nyquist condition for the time to frequency fast Fourier transform (FFT) where its bandwidth is 2*f0 excluding negative frequencies.

4 Click to expand the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

5 Under **Selections**, click **Add**.

6 In the **Add** dialog box, in the **Selections** list, choose **Explicit 1**, **Explicit 2**, and **Explicit 3**.

7 Click **OK**.

By choosing only the boundaries of interest for **Store fields in output** settings, it is possible to reduce the size of a model file a lot. It is necessary to include the port boundaries to calculate S-parameters and far-field calculation boundaries for visualizing the radiation pattern.

Step 2: Time to Frequency FFT

1 In the **Model Builder** window, click **Step 2: Time to Frequency FFT**.

2 In the **Settings** window for **Time to Frequency FFT**, locate the **Study Settings** section.

3 In the **End time** text field, type `Tend*3`. This makes sure that the FFT end time is longer than the simulation time so zero-padding can be applied during the time to frequency FFT. This will generate a finer frequency resolution in the resulting frequency response.

4 In the **Maximum output frequency** text field, type `2*f0`.

5 Click to expand the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

6 Under **Selections**, click + **Add**.

7 In the **Add** dialog box, in the **Selections** list, choose **Explicit 1**, **Explicit 2**, and **Explicit 3**.

8 Click **OK**.

Step 3: Combine Solutions

1 In the **Model Builder** window, click **Step 3: Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 In the **Excluded if** text field, type `freq<0.1*f0 || freq>2*f0-0.1*f0`. This excludes the first 5% and last 5% of the frequency response after FFT.

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

RESULTS

3D Plot Group 1

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

2 From the **Parameter value (freq (Hz))** list, choose **7.1E8**.

Multislice 1

1 In the **Model Builder** window, expand the **3D Plot Group 1** node.

2 Right-click **Multislice 1** and choose **Disable**.

Surface 1

1 In the **Model Builder** window, right-click **3D Plot Group 1** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Aurora>AuroraAustralis** in the tree.

5 Click **OK**.

Selection 1

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

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

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

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

Strong electric fields are observed on the antenna strip when it is resonant. See Figure 2.

Global 1

1 In the **Model Builder** window, expand the **Results>S-parameter (temw)** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
temw.S11dB	dB	S11

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **GHz**.

Graph Marker 1

1 Right-click **Global 1** and choose **Graph Marker**.

2 In the **Settings** window for **Graph Marker**, locate the **Display** section.

3 From the **Display** list, choose **Min**.

4 From the **Scope** list, choose **Local**.

5 Locate the **Text Format** section. In the **Display precision** text field, type 3.

6 Select the **Show x-coordinate** check box.

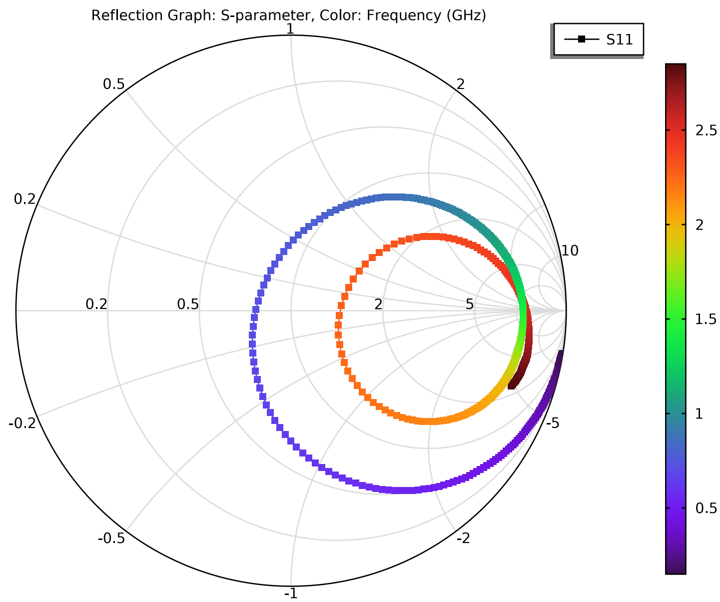
7 Select the **Include unit** check box.

8 Click to expand the **Coloring and Style** section. From the **Anchor point** list, choose **Lower left**.

S-parameter (temw)

Compare the reproduced plot to Figure 3. Two resonances are observed in the simulated frequency range.

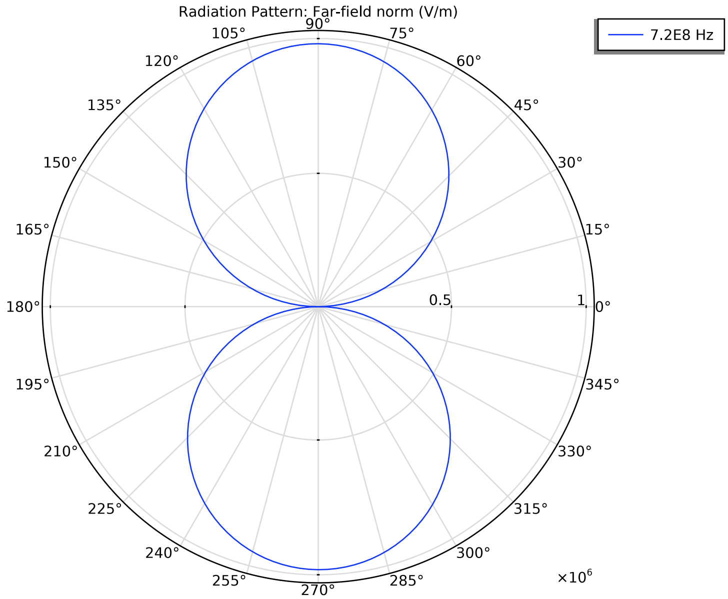
Smith Plot (temw)



2D Far Field (temw)

1. In the **Model Builder** window, click **2D Far Field (temw)**.
2. In the **Settings** window for **Polar Plot Group**, locate the **Data** section.
3. From the **Parameter selection (freq)** list, choose **From list**.
4. In the **Parameter values (freq (Hz))** list, select **7.2E8**.

5 In the **2D Far Field (temw)** toolbar, click **Plot**.



It is the far-field radiation pattern in a polar plot around the first resonance. This E-plane radiation pattern resembles that of a dipole antenna.

3D Far Field, Gain (temw)

1. In the **Model Builder** window, click **3D Far Field, Gain (temw)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Parameter value (freq (Hz))** list, choose **7.2E8**.
4. In the **3D Far Field, Gain (temw)** toolbar, click **Plot**.
5. Click the **Zoom Extents** button in the **Graphics** toolbar.

Radiation Pattern 1

1. In the **Model Builder** window, expand the **3D Far Field, Gain (temw)** node, then click **Radiation Pattern 1**.
2. In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.
3. Find the **Angles** subsection. In the **Number of azimuth angles** text field, type 180.
4. In the **Number of elevation angles** text field, type 180.
5. Locate the **Coloring and Style** section. Click **Change Color Table**.
6. In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Radiation Pattern**, locate the **Coloring and Style** section.

9 From the **Grid** list, choose **Finer**.

10 From the **Color** list, choose **Red**.

11 In the **3D Far Field, Gain (temw)** toolbar, click **Plot**.

TABLE

1 Go to the **Table** window.

Figure 4 (top) shows the far-field radiation pattern at the chosen frequency.

RESULTS

3D Far Field, Gain (temw)

1 In the **Model Builder** window, under **Results** click **3D Far Field, Gain (temw)**.

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

3 From the **Parameter value (freq (Hz))** list, choose **2.265E9**.

4 In the **3D Far Field, Gain (temw)** toolbar, click **Plot**.

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

Figure 4 (bottom) shows the far-field radiation pattern at the chosen frequency.