Created in COMSOL Multiphysics 6.2

FEM–BEM Coupling of a Microstrip Patch Antenna

Introduction

This example shows how to couple the finite element method (FEM), analyzing a microstrip patch antenna, to the boundary element method (BEM), for evaluating the field outside the FEM computational domain. The model computes the S-parameter, near-field distribution, and far-field radiation pattern through the FEM and the electric fields outside a given air domain sphere with the BEM.

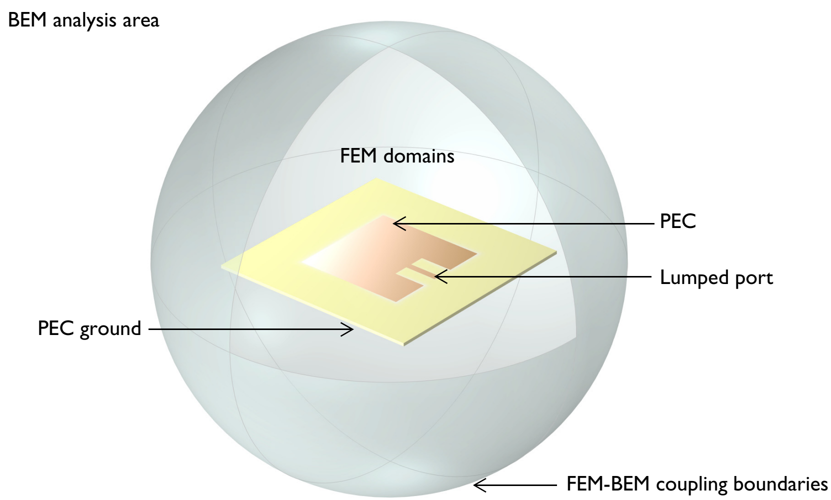


Figure 1: Model setup for the FEM–BEM coupling. One of the FEM–BEM coupling boundaries is removed from the view to show inside the FEM domains.

Model Definition

The FEM domains contain a microstrip patch antenna surrounded by an air domain. The antenna is excited by a uniform type lumped port that bridges the top feed line and bottom ground plane. A 60 mil substrate has a relative dielectric constant of 3.38. Additional details regarding the antenna design and impedance match are given in the Application Library example model Microstrip Patch Antenna. The outside of the FEM domains is set as infinite voids that are analyzed using the Electromagnetic Waves, Boundary Elements Interface. An Electric Field Coupling node under the Multiphysics branch addresses the coupling between FEM and BEM on the outermost boundaries of the FEM domains. For the conventional FEM-only analysis, an absorbing boundary condition such as a Scattering Boundary Condition is assigned on the outermost exterior boundaries to describe an open space for antenna analyses.

Results and Discussion

The computed S-parameter is below –20 dB indicating that the antenna input impedance is matched to the reference impedance of the lumped port (50 Ω). In Figure 2, strong electric fields are observed on the radiating edges.

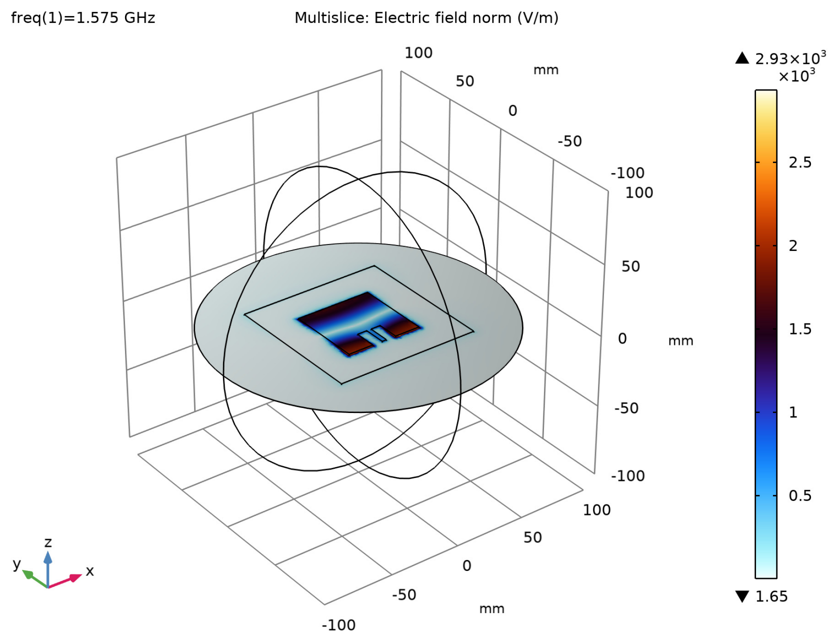


Figure 2: The electric field norm is plotted on the xy-plane inside the FEM domains.

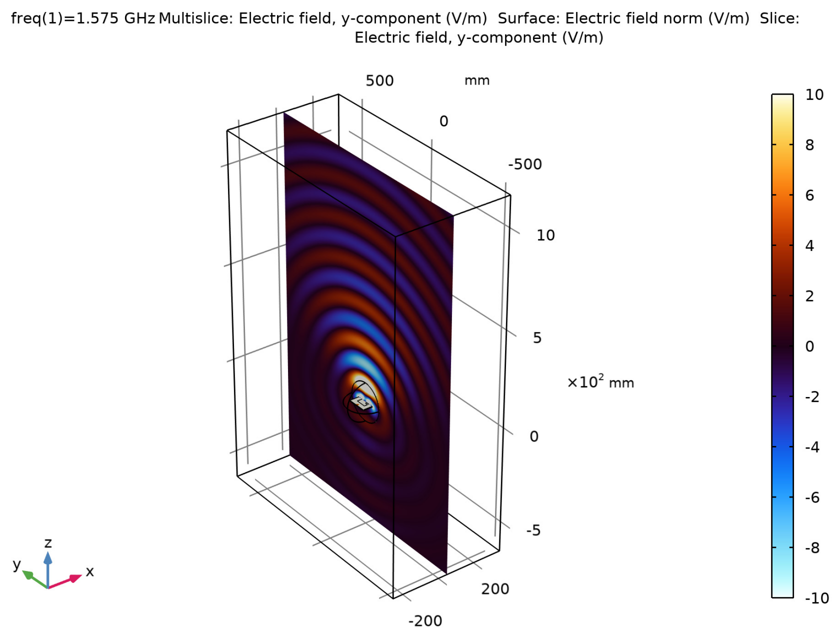


Figure 3: The y-component of the electric field outside the FEM domains are visualized using a Grid 3D dataset that can be configured to any size.

Figure 3 shows the smooth transition of the electric field from the FEM to the BEM surfaces visualizing the dominant polarization of the microstrip patch antenna.

Notes About the COMSOL Implementation

The model uses **Only plot when requested** functionality that is located at the **Results** node in the **Model Builder**. The visualization of BEM results often takes longer than the conventional FEM plots. This option may help saving time since it prevents any plot update until it is explicitly requested by clicking the **Plot** button.

Application Library path: `RF_Module/Antennas/microstrip_patch_antenna_fem_bem`

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, FEM-BEM**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Frequency Domain**.

6 Click **Done**.

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**.

STUDY 1

Step 1: Frequency Domain

Define the study frequency ahead of performing any frequency-dependent operation such as building mesh. The physics-controlled mesh uses the specified frequency value.

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 `1.575[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 In the table, enter the following settings:

Name	Expression	Value	Description
d	60[mil]	0.001524 m	Substrate thickness
w_line	3.2[mm]	0.0032 m	50 ohm line width
w_patch	53[mm]	0.053 m	Patch width
l_patch	52[mm]	0.052 m	Patch length
w_stub	7[mm]	0.007 m	Tuning stub width
l_stub	15.5[mm]	0.0155 m	Tuning stub length
w_sub	100[mm]	0.1 m	Substrate width
l_sub	100[mm]	0.1 m	Substrate length

Here `mil` refers to the unit milliinch, that is 1 mil = 0.0254 mm.

GEOMETRY 1

First, create the substrate block.

Substrate

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

2 In the **Settings** window for **Block**, type `Substrate` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_sub`.

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

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

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

7 Click **Build Selected**.

Add the patch antenna.

Patch

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

2 In the **Settings** window for **Block**, type `Patch` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_patch`.

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

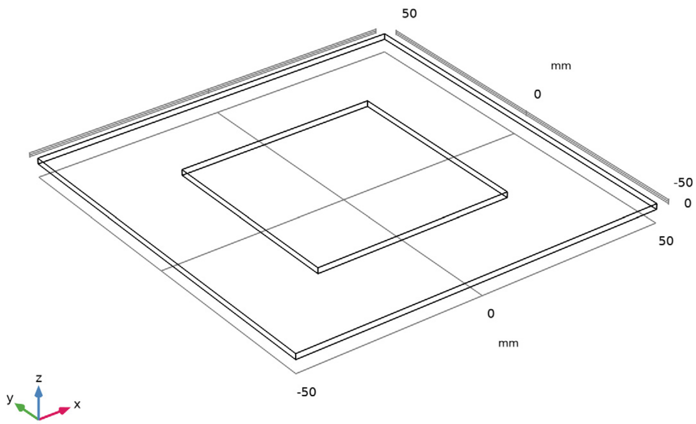
5 In the **Height** text field, type `d`.

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

7 Click **Build Selected**.

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

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



Create the impedance matching parts and a 50Ω feed line.

Stub

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

2 In the **Settings** window for **Block**, type `Stub` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `w_stub`.

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

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

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

7 In the **x** text field, type `w_stub/2+w_line/2`.

8 In the **y** text field, type `l_stub/2-l_patch/2`.

Copy 1 (copy1)

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

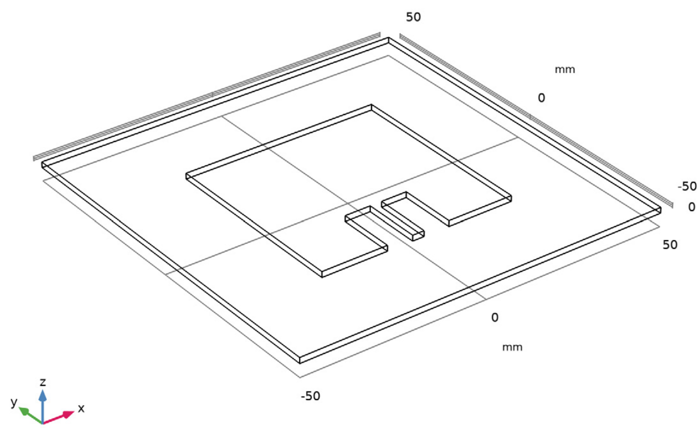
2 Select the object **blk3** only.

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

4 In the **x** text field, type `-w_stub-w_line`.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the object **blk2** only.
3 In the **Settings** window for **Difference**, locate the **Difference** section.
4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.
5 Select the objects **blk3** and **copy1** only.
6 Click **Build Selected**.



Add a sphere for the surrounding air.

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 `l_sub`.
4 Click **Build All Objects**.
5 Click the **Zoom Extents** 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

Air (mat1)

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

2 From the **Selection** list, choose **All domains and voids**.

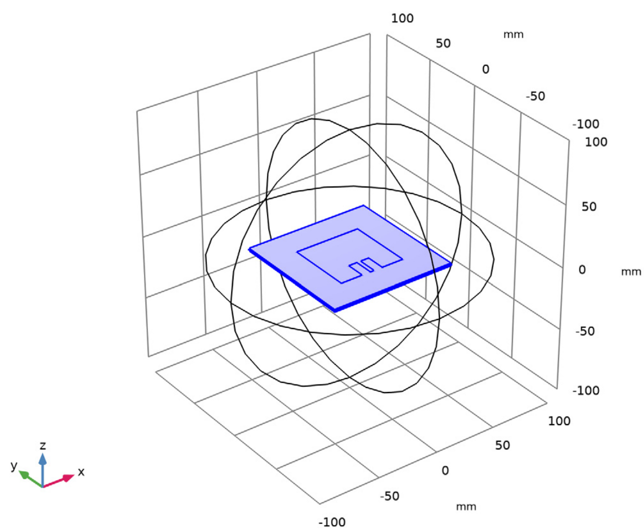
Include voids for the BEM analysis.

Substrate

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

2 In the **Settings** window for **Material**, type `Substrate` in the **Label** text field.

3 Select Domains 2 and 3 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	3.38	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

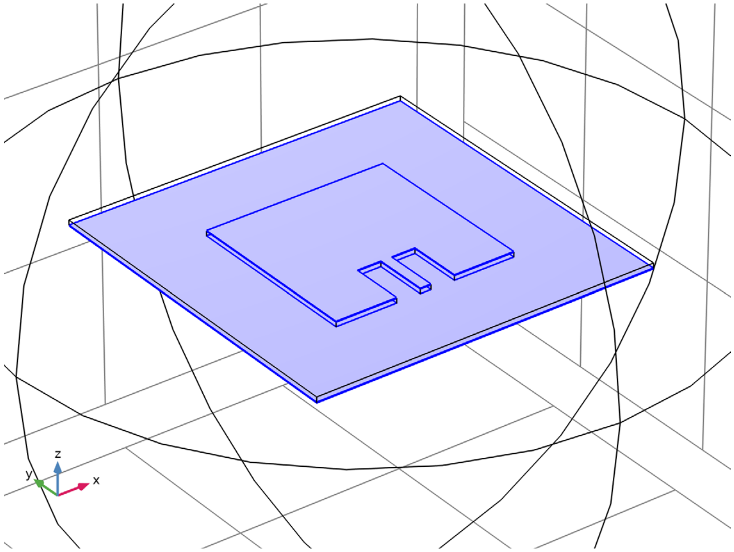
ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

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 Click the **Zoom In** button in the **Graphics** toolbar, a couple of times to get a view of the antenna structure.

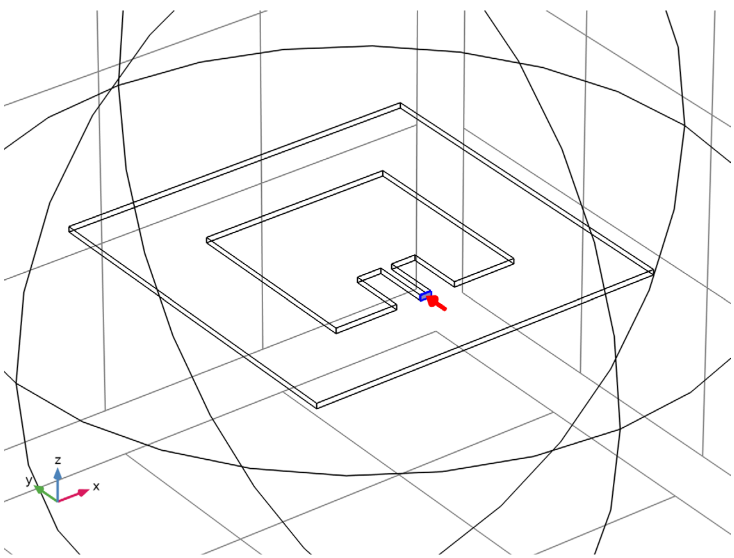
3 Select Boundaries 7, 12, and 13 only.



Lumped Port 1

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

2 Select Boundary 18 only.



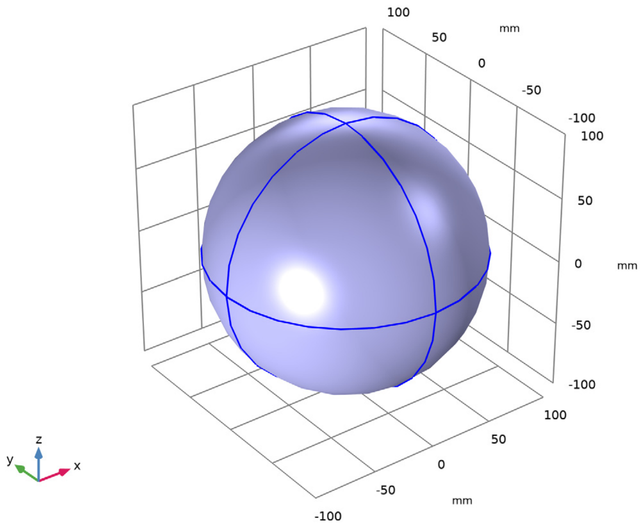
Far-Field Domain 1

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 Click **Clear Selection**.

4 Select Domain 1 only.



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

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

3 Click **Create Selection**.

4 In the **Create Selection** dialog box, type `FEM-BEM coupling boundaries` in the **Selection name** text field.

5 Click **OK**.

ELECTROMAGNETIC WAVES, BOUNDARY ELEMENTS (EMBE)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Boundary Elements (embe)**.

2 In the **Settings** window for **Electromagnetic Waves, Boundary Elements**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All voids**.

MULTIPHYSICS

Electric Field Coupling 1 (elfc1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Electric Field Coupling 1 (elfc1)**.

2 In the **Settings** window for **Electric Field Coupling**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **FEM-BEM coupling boundaries**.

MESH 1

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

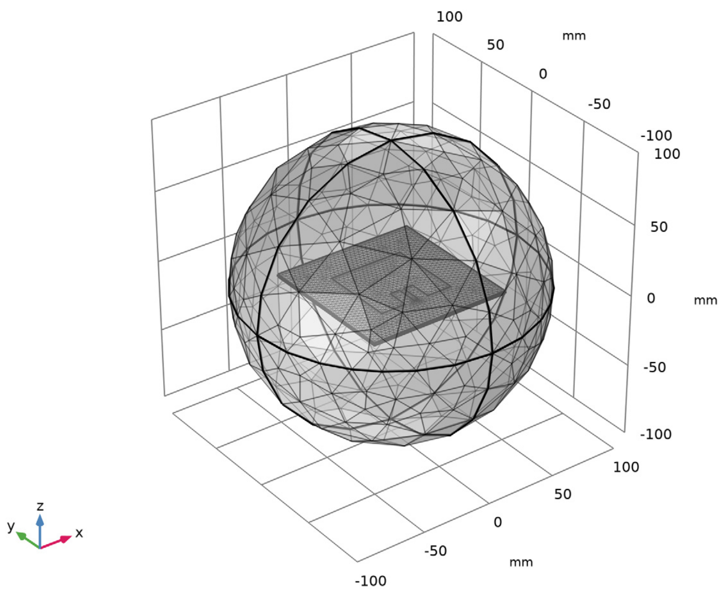
2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **Physics-controlled mesh**.

4 Click **Build All**.

Try to use transparency rendering.

5 Click the **Transparency** button in the **Graphics** toolbar.



6 Click the **Transparency** button in the **Graphics** toolbar.

STUDY 1

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

Check the computed S-parameter value.

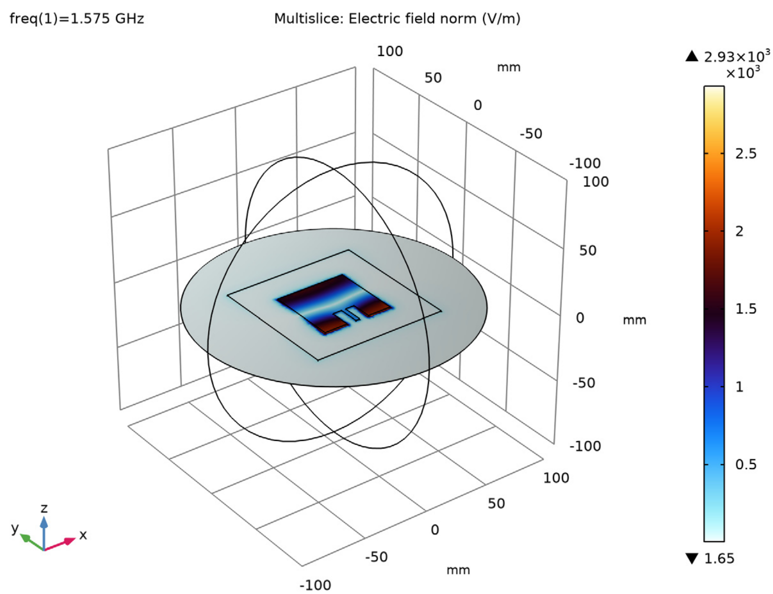
RESULTS

S-parameter (emw)

In the **Model Builder** window, expand the **Results>Derived Values** node.

Multislice

1 In the **Model Builder** window, expand the **Results>Electric Field (emw)** node, then click **Multislice**.
2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.
3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.
4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.
5 Locate the **Coloring and Style** section. Click **Change Color Table**.
6 In the **Color Table** dialog box, select **Thermal>ThermalWaveDark** in the tree.
7 Click **OK**.



Adjust the polar plot settings to generate the E-plane radiation pattern.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **Results>2D Far Field (emw)** node, then click **Radiation Pattern 1**.

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

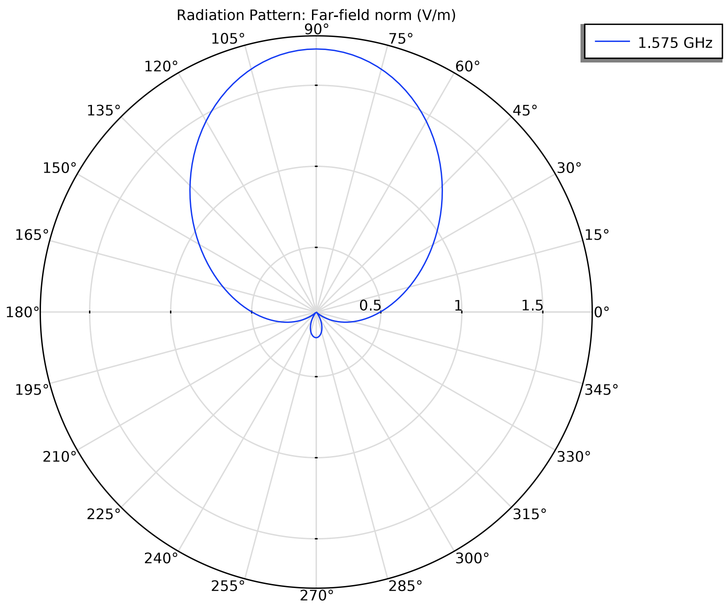
3 Find the **Normal vector** subsection. In the **x** text field, type `1`.

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

5 Find the **Reference direction** subsection. In the **x** text field, type `0`.

6 In the **y** text field, type `1`.

7 In the **2D Far Field (emw)** toolbar, click **Plot**.



Multislice 1

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

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

3 In the **Expression** text field, type `embe.Ey`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>ThermalWaveDark** in the tree.

6 Click **OK**.

The BEM solution is visualized using the **Grid 3D 1** dataset. Resize and enhance the resolution.

Grid 3D 1

1. In the **Model Builder** window, expand the **Results>Datasets** node, then click **Grid 3D 1**.
2. In the **Settings** window for **Grid 3D**, locate the **Parameter Bounds** section.
3. Find the **Second parameter** subsection. In the **Minimum** text field, type `-600`.
4. In the **Maximum** text field, type `600`.
5. Find the **Third parameter** subsection. In the **Minimum** text field, type `-600`.
6. In the **Maximum** text field, type `1200`.
7. Click to expand the **Grid** section. In the **x resolution** text field, type `2`.
8. In the **y resolution** text field, type `600`.
9. In the **z resolution** text field, type `1200`.

   The visualization of BEM results often takes longer than typical FEM cases. By checking **Only plot when requested**, the plot is not instantly updated when changing the settings. Finalize the plot settings first and then click the **Plot** button to see the results.
10. In the **Model Builder** window, click **Results**.
11. In the **Settings** window for **Results**, locate the **Update of Results** section.
12. Select the **Only plot when requested** check box.

Multislice 1

1. In the **Model Builder** window, under **Results>Electric Field, Domains (embe)** click **Multislice 1**.
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 **z-planes** subsection. In the **Planes** text field, type `0`.
5. Click to expand the **Range** section. Select the **Manual color range** check box.
6. In the **Minimum** text field, type `-10`.
7. In the **Maximum** text field, type `10`.

Surface 1

1. In the **Model Builder** window, click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
4. Locate the **Expression** section. In the **Expression** text field, type `emw.normE`.

Selection 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `5-9, 12, 13, 28` in the **Selection** text field.

5 Click **OK**.

Slice 1

1 In the **Model Builder** window, right-click **Electric Field, Domains (embe)** and choose **Slice**.

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

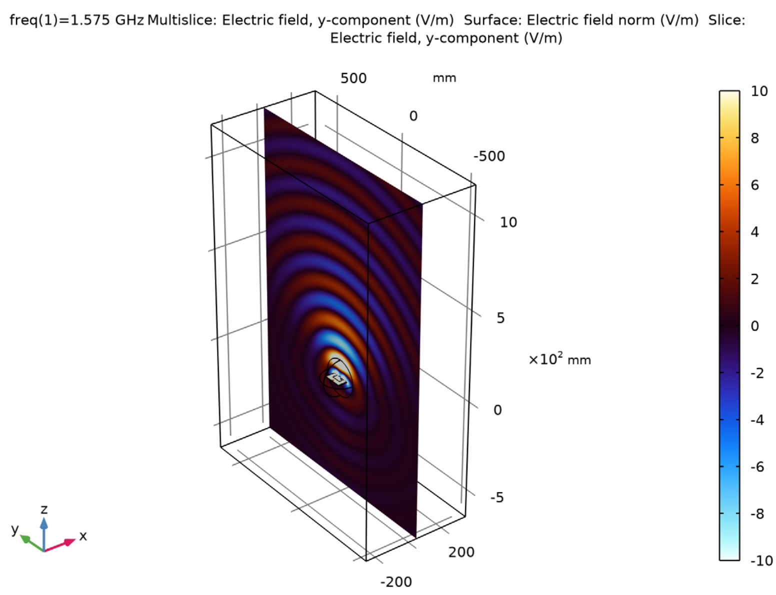
3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Locate the **Expression** section. In the **Expression** text field, type `emw.Ey`.

5 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Multislice 1**.

7 In the **Electric Field, Domains (embe)** toolbar, click **Plot**.



Note that the surface current plot in other plot groups can be physically meaningful when it is visualized on perfect electric conductor boundaries representing metallic surfaces outside the FEM domains.