Model created in COMSOL Multiphysics 6.4

Mixed-Mode S-Parameters Analysis

Introduction

Mixed-mode S-parameters describe the responses of a circuit with balanced ports excited and terminated by two types of modes: common and differential modes. They are calculated using a full S-parameter matrix of a four-port network that is composed of four single ended lines. This example analyzes two adjacent microstrip lines and computes the mixed-mode S-parameters.

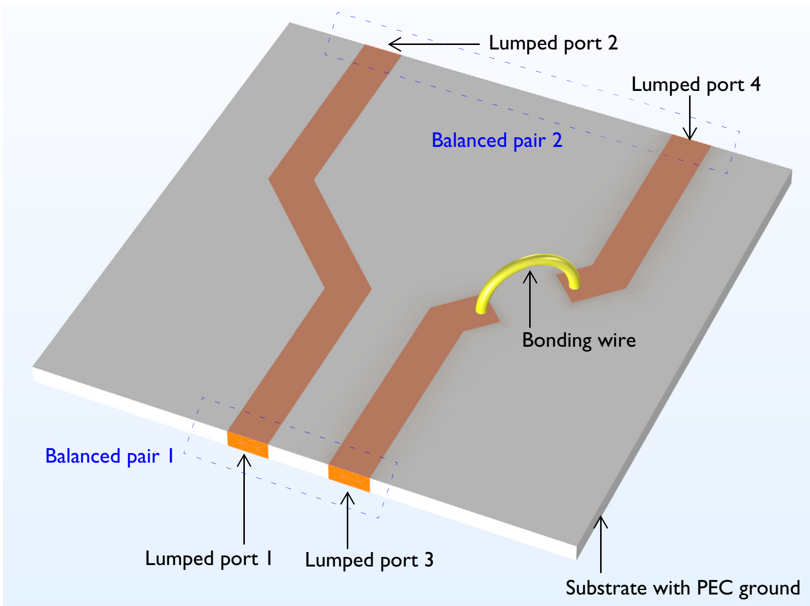


Figure 1: Microstrip line circuit board modeled with lumped ports for mixed-mode S-parameters calculation. The surrounding air domain is not included for visualization purposes.

Model Definition

The model consists of a pair of microstrip line adjacent to each other on a 20 mil substrate with a dielectric constant of ε_r = 3.38. One line has a discontinuity that is connected with a bonding wire. The bonding wire geometry is built with a half of torus that has a wire radius of 0.15 mm. All metallic parts, including the microstrip lines, the bottom ground plane, and the bonding wire surface, are set to perfect electric conductor (PEC), due to the negligible loss from the finite conductivity. The circuit is surrounded by an air domain.

The exterior surfaces of the air domain are finished by a scattering boundary condition – an absorbing boundary to describe an open radiating space.

On the physics interface settings, activate port sweep. With port sweep activated, a parametric sweep over the port name is added in the study. Thereby a full four-by-four S-parameter matrix is obtained. The S-parameter matrix is required to process the mixed-mode S-parameters. The balanced ports out of a four-port network are configured by a global feature called mixed-mode S-parameters where the balanced port 1 is composed of port 1 and 3, and the balanced port 2 is defined with port 2 and 4.

Figure 2: Balanced port configuration of a four port network in the mixed-mode S-parameter feature.

Based on the balanced port settings, the mixed-mode S-parameters are defined as

$$S_{\text{mixed-mode}} = \frac{1}{2}\begin{pmatrix} S_{dd11} & S_{dd12} & S_{dc11} & S_{dc12} \\ S_{dd21} & S_{dd22} & S_{dc21} & S_{dc22} \\ S_{cd11} & S_{cd12} & S_{cc11} & S_{cc12} \\ S_{cd21} & S_{cd21} & S_{cc11} & S_{cc22} \end{pmatrix}$$

$$S_{dd11} = \frac{1}{2}(S_{AA} - S_{AB} - S_{BA} + S_{BB}),\ \ S_{dd12} = \frac{1}{2}(S_{AC} - S_{AD} - S_{BC} + S_{BD})$$

$$S_{dd21} = \frac{1}{2}(S_{CA} - S_{CB} - S_{DA} + S_{DB}),\ \ S_{dd22} = \frac{1}{2}(S_{CC} - S_{CD} - S_{DC} + S_{DD})$$

$$S_{dc11} = \frac{1}{2}(S_{AA} + S_{AB} - S_{BA} - S_{BB}),\ \ S_{dc12} = \frac{1}{2}(S_{AC} + S_{AD} - S_{BC} - S_{BD})$$

$$S_{dc21} = \frac{1}{2}(S_{CA} + S_{CB} - S_{DA} - S_{DB}),\ \ S_{dc22} = \frac{1}{2}(S_{CC} + S_{CD} - S_{DC} - S_{DD})$$

$$S_{cd11} = \frac{1}{2}(S_{AA} - S_{AB} + S_{BA} - S_{BB}),\ \ S_{cd12} = \frac{1}{2}(S_{AC} - S_{AD} + S_{BC} - S_{BD})$$

$$S_{cd21} = \frac{1}{2}(S_{CA} - S_{CB} + S_{DA} - S_{DB}),\ \ S_{cd22} = \frac{1}{2}(S_{CC} - S_{CD} + S_{DC} - S_{DD})$$

$$S_{cc11} = \frac{1}{2}(S_{AA} + S_{AB} + S_{BA} + S_{BB}),\ S_{cc12} = \frac{1}{2}(S_{AC} + S_{AD} + S_{BC} + S_{BD})$$

$$S_{cc21} = \frac{1}{2}(S_{CA} + S_{CB} + S_{DA} + S_{DB}),\ S_{cc22} = \frac{1}{2}(S_{CC} + S_{CD} + S_{DC} + S_{DD})$$

where subscript c and d stand for common mode and differential mode, respectively.

Each subscript of the mixed-mode S-parameters in the notation of S_{mnij} represents

- m: output observation mode
- n: input excitation mode
- i: output observation balanced port
- j: input excitation balanced port

All sixteen mixed-mode S-parameter components are listed in Table 1.

TABLE 1: MIXED-MODE S-PARAMETER SUBSCRIPT DESCRIPTION.

	OUTPUT MODE	INPUT MODE	OUTPUT PORT	INPUT PORT
S_{cc11}	common	common	1	1
S_{cc12}	common	common	1	2
S_{cc21}	common	common	2	1
S_{cc22}	common	common	2	2
S_{cd11}	common	differential	1	1
S_{cd12}	common	differential	1	2
S_{cd21}	common	differential	2	1
S_{cd22}	common	differential	2	2
S_{dc11}	differential	common	1	1
S_{dc12}	differential	common	1	2
S_{dc21}	differential	common	2	1
S_{dc22}	differential	common	2	2
S_{dd11}	differential	differential	1	1
S_{dd12}	differential	differential	1	2
S_{dd21}	differential	differential	2	1
S_{dd22}	differential	differential	2	2

Results and Discussion

Figure 3 shows the plot of the electric field norm when port 4 is excited. In this surface plot, there is no visible coupling effect to the adjacent microstrip line between port 1 and 2.

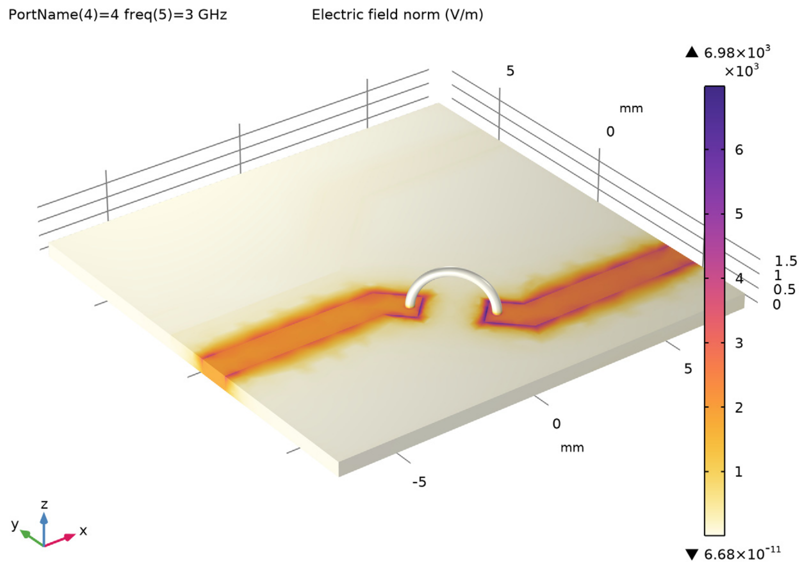


Figure 3: Electric field norm plot when port 4 is excited.

In Figure 4, the mixed-mode S-parameters, S_{cc11}, S_{cd12}, S_{dc21}, and S_{dd22}, are plotted. The level of mode conversion between common and differential modes increases as frequency increases.

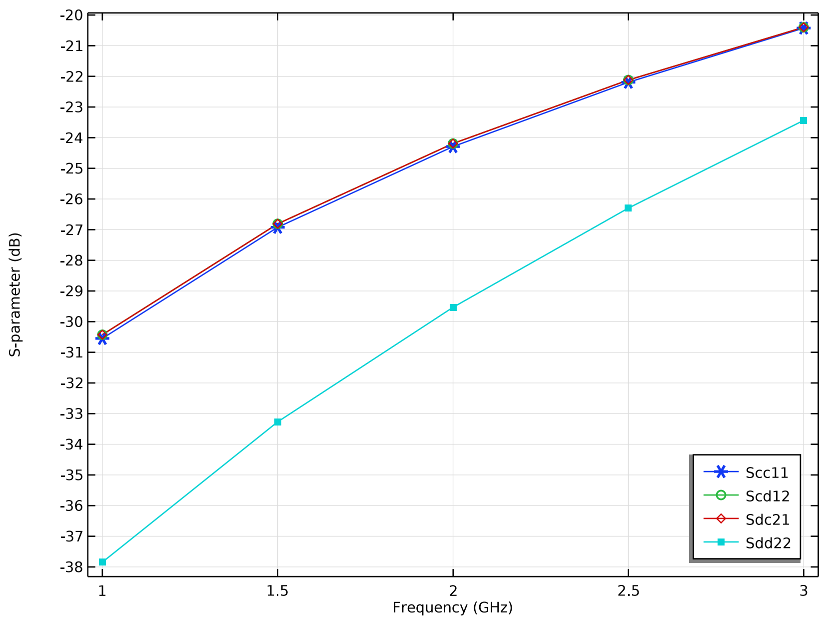


Figure 4: The mixed-mode S-parameters. The cross-mode conversion between common and differential modes can be estimated from the mixed-mode S-parameters.

Application Library path: `RF_Module/EMI_EMC_Applications/microstrip_line_mixed_mode`

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**.

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
sub_l	15[mm]	0.015 m	Substrate length
sub_w	15[mm]	0.015 m	Substrate width
sub_t	20[mil]	5.08E-4 m	Substrate thickness
line_w	1.13[mm]	0.00113 m	Line width

It is convenient to define parameters for frequently used values. Here, `mil` refers to the unit milliinch.

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**.

Add a box for the substrate geometry.

Block 1 (blk1)

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

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

3 In the **Width** text field, type `sub_l`.

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

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

6 Locate the **Position** section. In the **z** text field, type `sub_t/2`.

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

Draw two microstrip line patterns on top of the substrate.

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 In the **z-coordinate** text field, type `sub_t`.

Work Plane 1 (wp1) > Plane Geometry

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

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`.

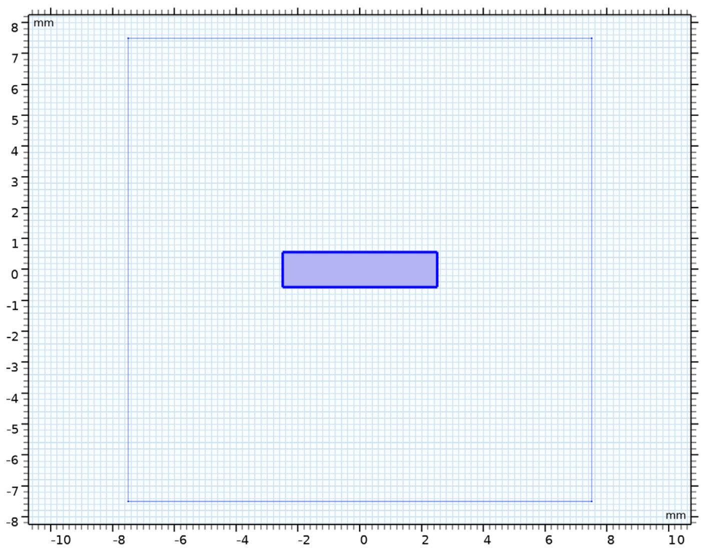
4 In the **Height** text field, type `line_w`.

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

Work Plane 1 (wp1) > Rotate 1 (rot1)

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

2 Select the object **r1** only.



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

4 In the **Angle** text field, type `45`.

5 Click **Build Selected**.

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

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** right-click **Rectangle 1 (r1)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `sub_l/2-5/2/sqrt(2)+line_w/2/sqrt(2)`.

4 Locate the **Position** section. In the **xw** text field, type `-sub_l/2+(sub_l/2-5/2/sqrt(2)+line_w/2/sqrt(2))/2`.

5 In the **yw** text field, type `-5/2/sqrt(2)+(line_w/2-line_w/2/sqrt(2))`.

Work Plane 1 (wp1) > Rotate 2 (rot2)

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

2 Select the object **r2** only.

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

4 Select the **Keep input objects** checkbox.

5 Locate the **Rotation** section. In the **Angle** text field, type `180`.

6 Click **Build Selected**.

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.

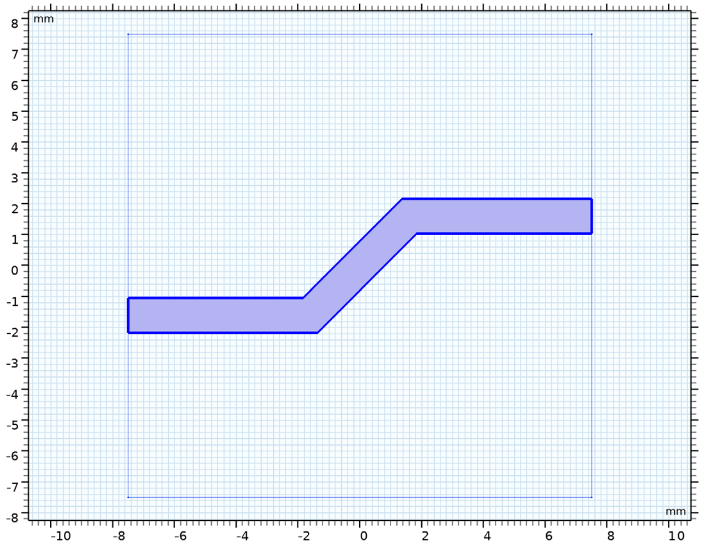
3 In the **Settings** window for **Union**, locate the **Union** section.

4 Clear the **Keep interior boundaries** checkbox.

Work Plane 1 (wp1) > Move 1 (mov1)

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

2 Select the object **uni1** only.



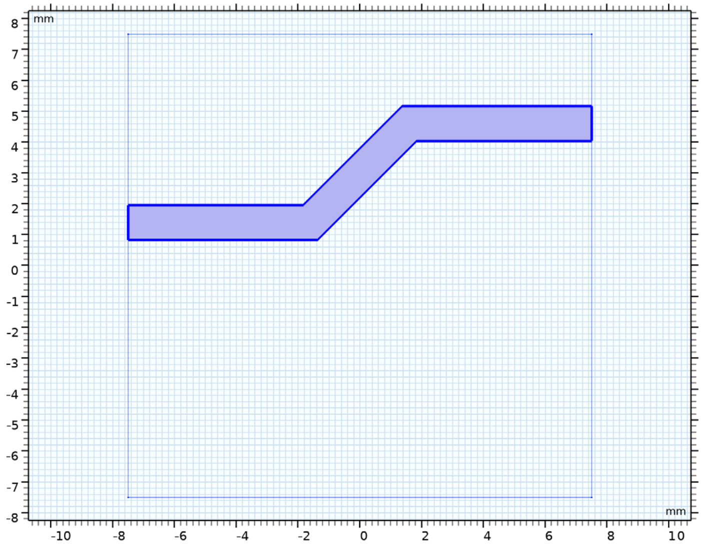
3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **yw** text field, type 3.

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

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

2 Select the object **mov1** only.



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

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

5 In the **yw** text field, type `1`.

6 Click **Build Selected**.

7 Locate the **Input** section. Select the **Keep input objects** checkbox.

8 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 `2`.

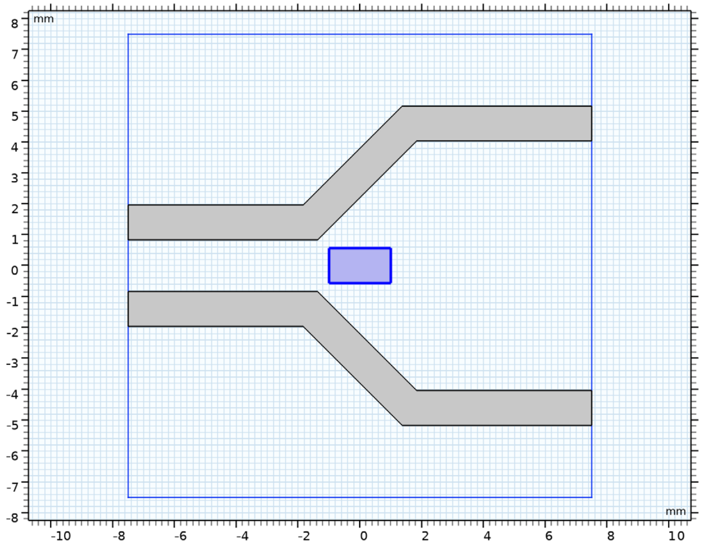
4 In the **Height** text field, type `line_w`.

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

Work Plane 1 (wp1) > Rotate 3 (rot3)

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

2 Select the object **r3** only.



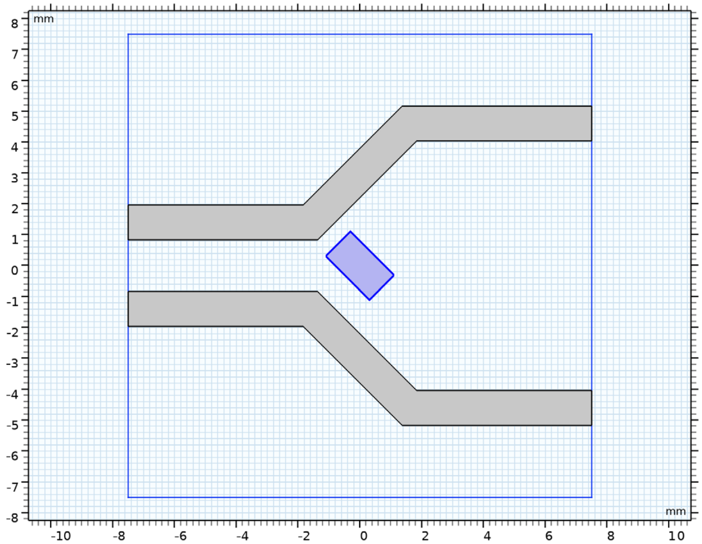
3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `-45`.

Work Plane 1 (wp1) > Move 2 (mov2)

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

2 Select the object **rot3** only.



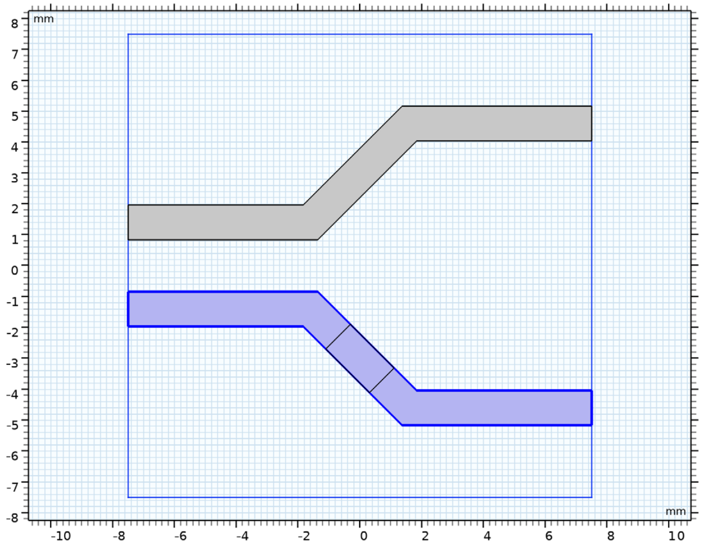
3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **yw** text field, type `-3`.

Work Plane 1 (wp1) > Difference 1 (dif1)

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

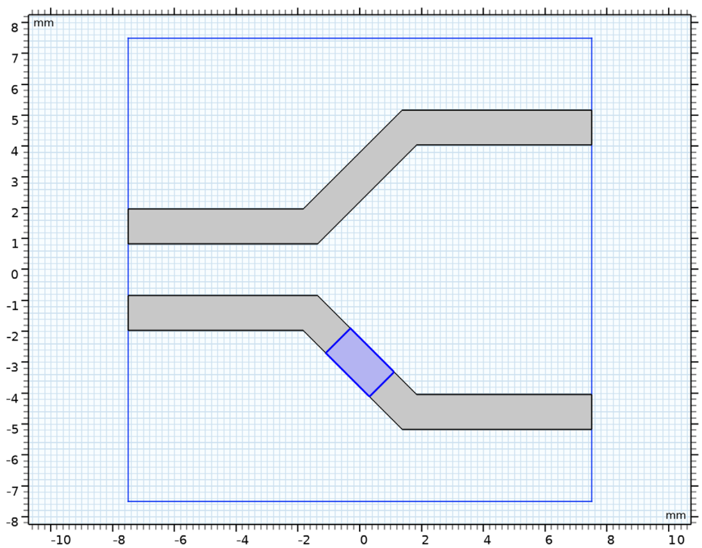
2 Select the object **mir1** only.



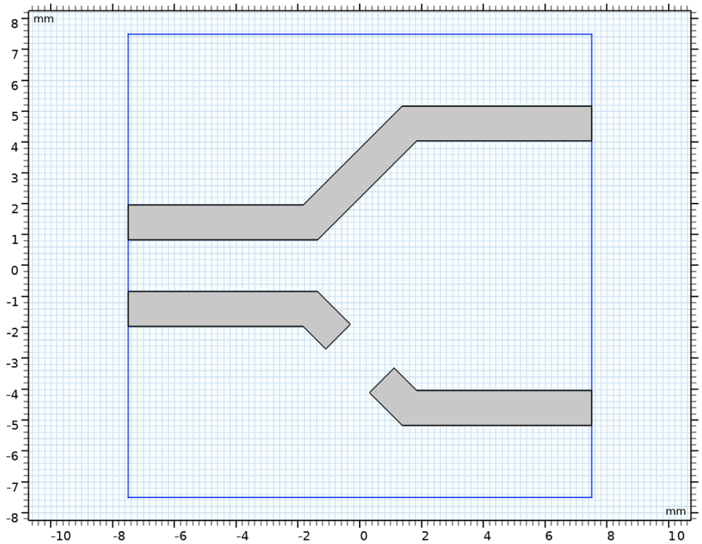
3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **mov2** only.



6 Click **Build Selected**.



By extruding the pattern, the boundaries for the four lumped ports can be created.

Extrude 1 (ext1)

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

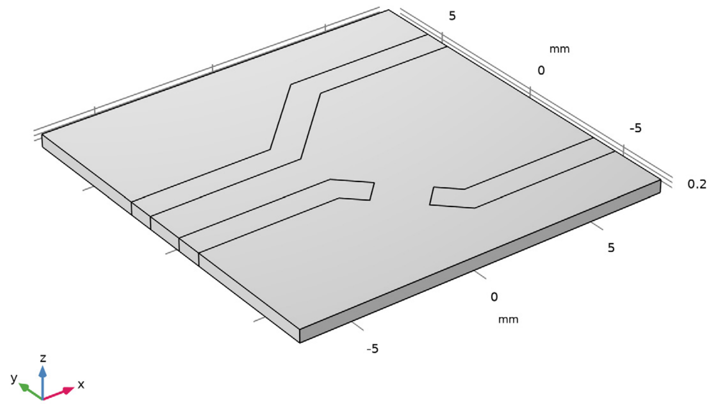
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (mm)
sub_t

4 Select the **Reverse direction** checkbox.

5 Click Build Selected.



Add a structure representing a bonding wire.

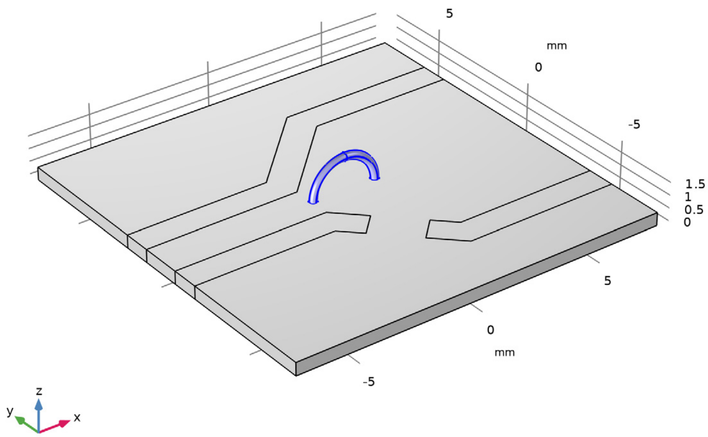
Torus 1 (tor1)

1. In the **Geometry** toolbar, click **Torus**.
2. In the **Settings** window for **Torus**, locate the **Size and Shape** section.
3. In the **Major radius** text field, type `1.3`.
4. In the **Minor radius** text field, type `0.15`.
5. In the **Revolution angle** text field, type `180`.
6. Locate the **Position** section. In the **z** text field, type `sub_t`.
7. Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.
8. Locate the **Rotation Angle** section. In the **Rotation** text field, type `90`.
9. Click **Build Selected**.

Rotate 1 (rot1)

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

2 Select the object **tor1** only.



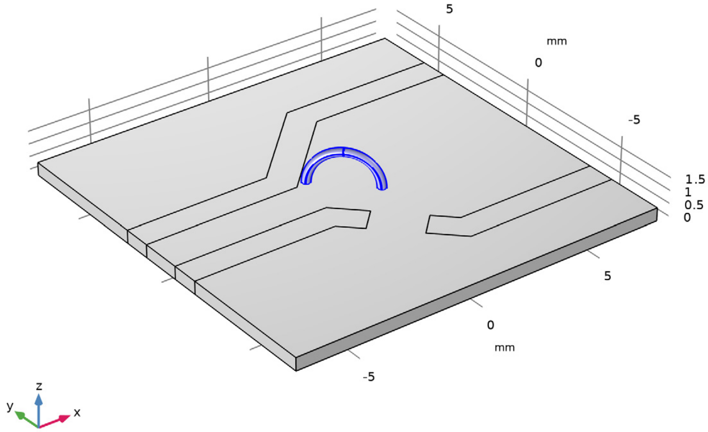
3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `-45`.

Move 1 (mov1)

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

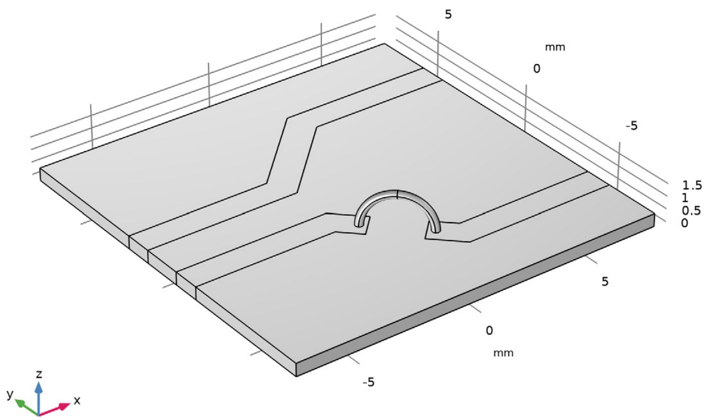
2 Select the object **rot1** only.



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

4 In the **y** text field, type `-3`.

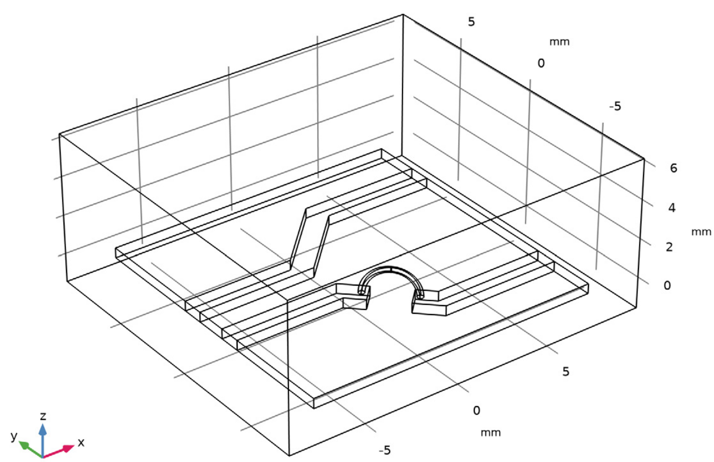
5 Click **Build Selected**.



Add a box for the air domain.

Block 2 (blk2)

1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type `sub_l+4[mm]`.
4 In the **Depth** text field, type `sub_w+2[mm]`.
5 In the **Height** text field, type `sub_t*15`.
6 Locate the **Position** section. In the **x** text field, type `-sub_l/2-2[mm]`.
7 In the **y** text field, type `-sub_w/2-1[mm]`.
8 In the **z** text field, type `-1[mm]`.
9 Click **Build All Objects**.
10 Click the **Wireframe Rendering** button in the **Graphics** toolbar. See the interior using the wireframe rendering.



ADD MATERIAL

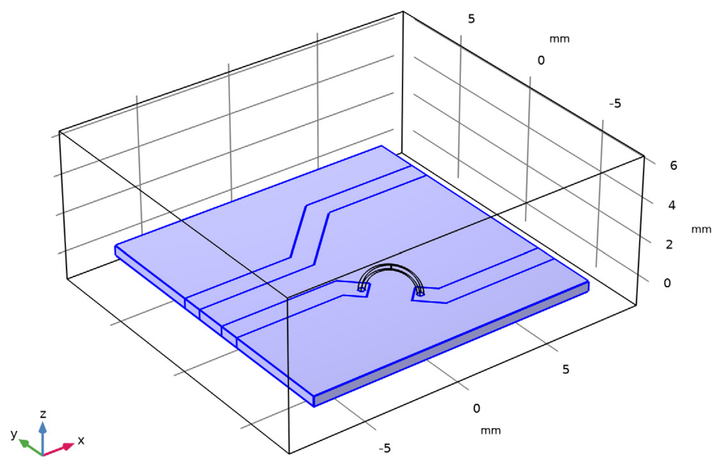
1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in** > **Air**.
4 Click the **Add to Component** button in the window toolbar.
5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material 2 (mat2)

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

2 Select Domains 2–5 and 7 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	3.38	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

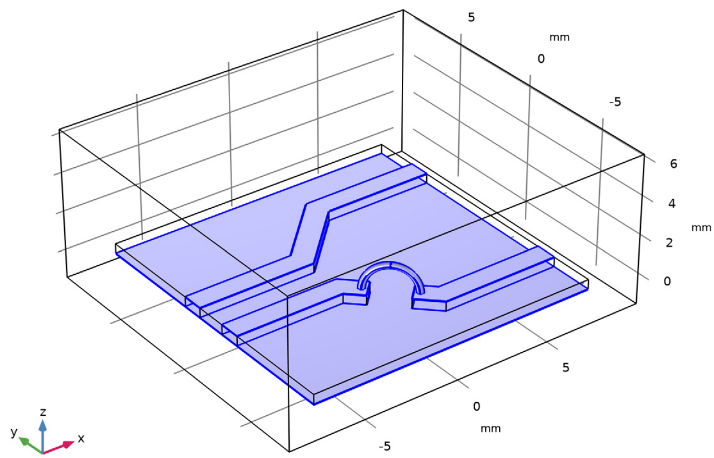
Assign **Perfect Electric Conductor** on all metal boundaries.

Perfect Electric Conductor 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Electric Conductor**.

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

3 Select Boundaries 8, 12, 13, 18, 19, 22, 31–38, 41, and 42 only. These are the boundaries of all metallic parts, including the microstrip lines, the bottom ground plane, and the bonding wire surface.

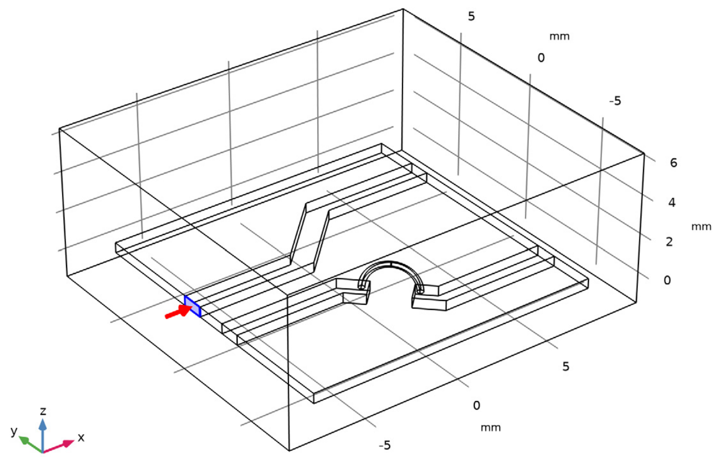


Add **Lumped Ports** on the boundaries between the microstrip lines and the ground plane.

Lumped Port 1

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

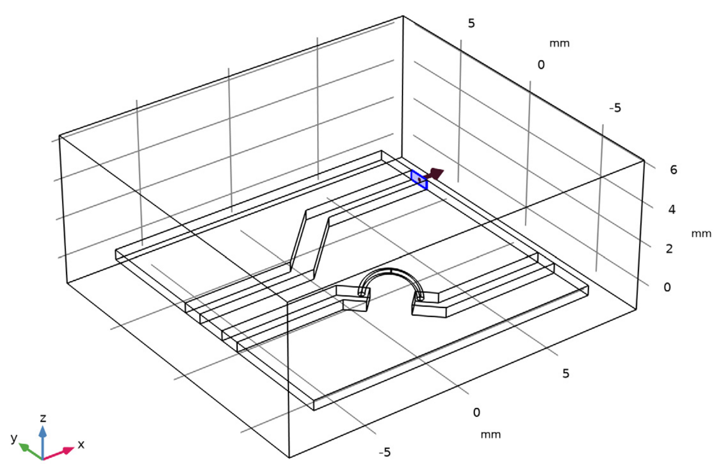
2 Select Boundary 16 only.



Lumped Port 2

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

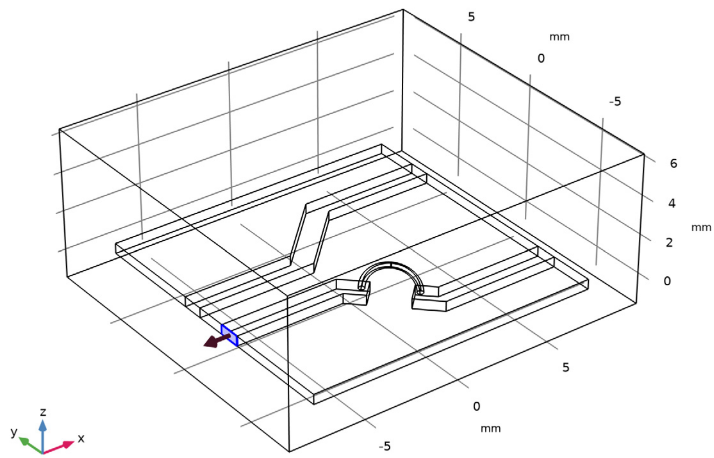
2 Select Boundary 52 only.



Lumped Port 3

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

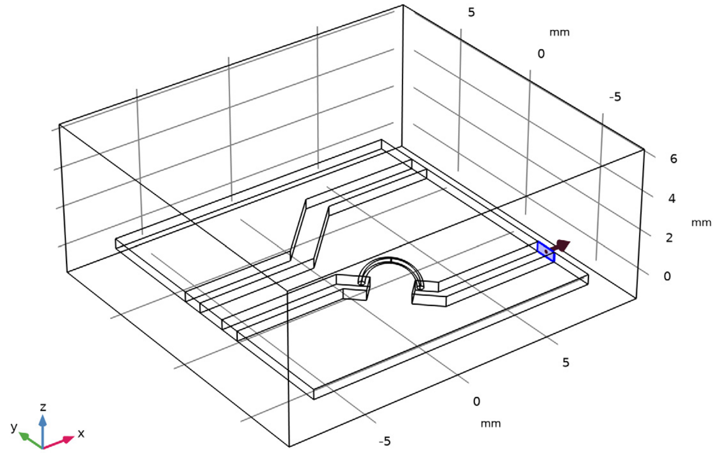
2 Select Boundary 10 only.



Lumped Port 4

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

2 Select Boundary 50 only.



Add **Scattering Boundary Condition** to absorb any radiation from the circuit board.

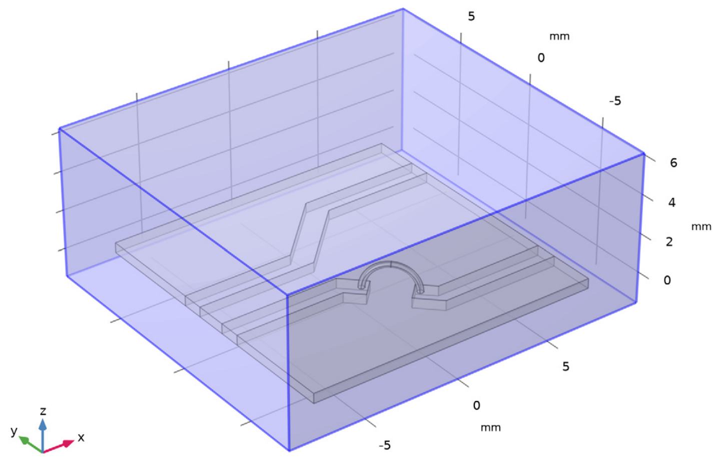
Scattering Boundary Condition 1

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

2 Select Boundaries 1–5 and 54 only.

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

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



The **Mixed-Mode S-parameters** global feature configures balanced ports and generates a four by four mixed mode S-parameter matrix.

Mixed-Mode S-Parameters 1

1 In the **Physics** toolbar, click **Global** and choose **Mixed-Mode S-Parameters**.

2 In the **Settings** window for **Mixed-Mode S-Parameters**, locate the **Balanced Port** section.

3 In the **Port name for port B in balanced pair 1** text field, type 3. **Balanced pair 1** includes port 1 and port 3.

4 In the **Port name for port C in balanced pair 2** text field, type 2. **Balanced pair 2** includes port 2 and port 4.

let us set up the physics properties.

5 In the **Model Builder** window, click **Electromagnetic Waves, Frequency Domain (emw)**.

6 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Port Sweep Settings** section.

7 Select the **Use manual port sweep** checkbox.

8 Click **Configure Sweep Settings**. By clicking the **Configure Sweep Settings** button, all necessary port sweep settings such as sweep parameter and parametric study step will be automatically added. It is necessary to run the parametric sweep with port names to get a full S-parameter matrix and build the mixed mode S-parameters.

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 **Coarse**.

4 Click  **Build All**.

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

Hide some boundaries to get a better view of the interior parts.

DEFINITIONS

Hide for Physics 1

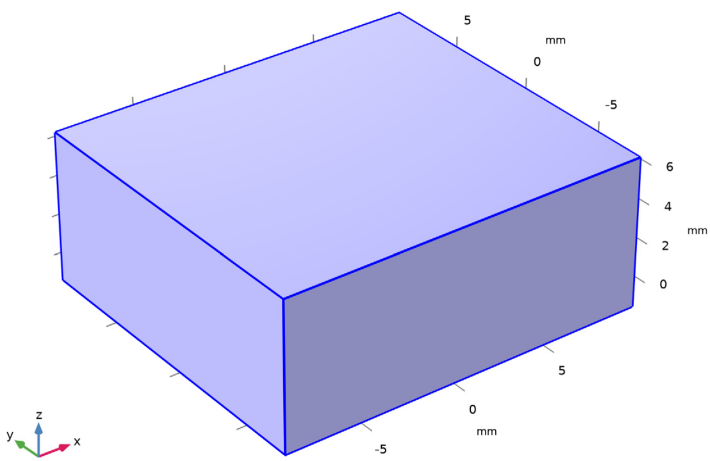
1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **View 1** and choose **Hide for Physics**.

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

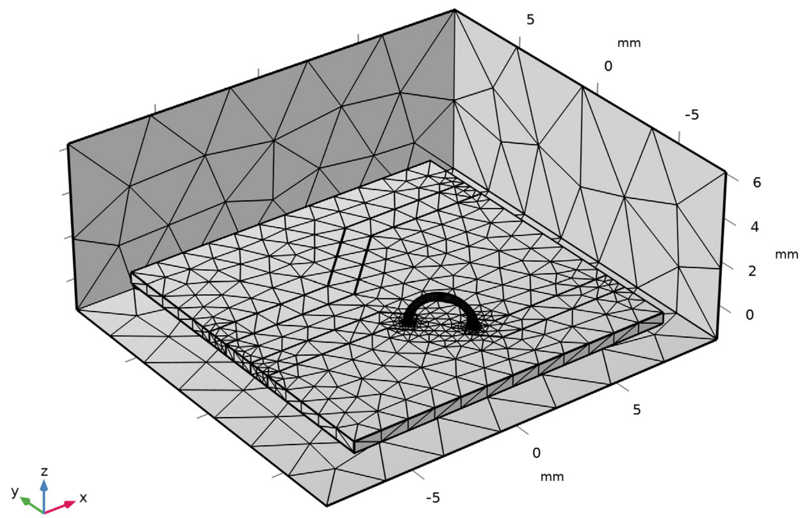
4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 1, 2, and 4 only.



MESH 1

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



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 Click **Range**.
4 In the **Range** dialog, type `1[GHz]` in the **Start** text field.
5 In the **Step** text field, type `0.5[GHz]`.
6 In the **Stop** text field, type `3[GHz]`.
7 Click **Replace**.
8 In the **Study** toolbar, click **Compute**.

Disable the default field plot and add a volume plot.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Results > Electric Field (emw)** node.

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

Volume 1

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

Selection 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Selection**.

2 Select Domains 2–7 only.

These are all domains except for the air domain.

Volume 1

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

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

3 From the **Color table** list, choose **HeatCameraLight**.

4 From the **Color table transformation** list, choose **Reverse**.

Electric Field (emw)

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

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

3 Clear the **Plot dataset edges** checkbox.

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

This reproduces Figure 3, the volume plot of the circuit board when port 4 is excited.

S-Parameter (emw)

1 In the **Model Builder** window, click **S-Parameter (emw)**.

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

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

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 In the **Model Builder** window, expand the **S-Parameter (emw)** node, then click **Global 1**.

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

3 Click **Clear Table**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Mixed-Mode S-Parameters 1** > **S-parameter, dB, common mode to common mode** > **emw.Scc11dB - Scc11**.

5 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Mixed-Mode S-Parameters 1** > **S-parameter, dB, common mode to differential mode** > **emw.Scd12dB - Scd12**.

6 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Mixed-Mode S-Parameters 1** > **S-parameter, dB, differential mode to common mode** > **emw.Sdc21dB - Sdc21**.

7 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Mixed-Mode S-Parameters 1** > **S-parameter, dB, differential mode to differential mode** > **emw.Sdd22dB - Sdd22**.

8 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 In the **S-Parameter (emw)** toolbar, click **Plot**.

Compare to Figure 4, the global 1D plot of the mixed mode S-parameters.

Electric Field, Logarithmic (emw)

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

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

3 Clear the **Plot dataset edges** checkbox.

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