Model created in COMSOL Multiphysics 6.4

CPW Resonator for Circuit Quantum Electrodynamics

Introduction

Developments in the last decade have led to circuit quantum electrodynamics (cQED) becoming the leading architecture candidate for quantum computation. The cQED architecture for quantum hardware has three main components: superconducting qubits, transmission lines, and transmission line resonators. Superconducting qubits are the artificial meta-atoms that serve as a two-level quantum system, transmission line resonators are high-quality superconducting oscillators that play the role of cavities, and transmission lines route energy through the architecture.

The energy difference between the quantum states of superconducting qubits is given by $E_{01} = hf_{01}$, wherein a two-level quantum system E_{01} is the energy difference between the ground state and the excited state, h is the Planck constant, and f_{01} is the transition frequency between the states. This frequency is typically designed to be in the range of 4–8 GHz so that it is above the thermal energy of dilution refrigerators (~10 mK) but also well below the superconducting gap of any constituent materials, like aluminum (82 GHz). Just like atoms, superconducting quantum qubits interact with microwave photons at quanta levels.

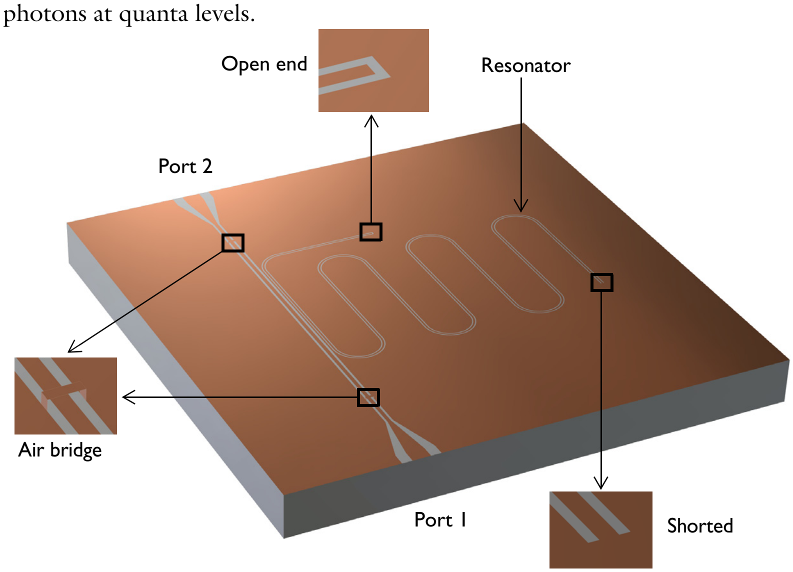


Figure 1: A CPW resonator coupled to a CPW transmission line. The air domains are removed for a better view.

In this model, one of the main components of cQEDs, a transmission line resonator, is demonstrated. This resonator can be built from CPW transmission lines terminated with a combination of open and short ends. These ends create a resonator out of a CPW, with the open and short ends functioning as zero current and zero voltage boundary conditions, respectively. Figure 1 illustrates a quarter-wave resonator, which is formed from a CPW terminated with an open and a shorted end, and shows how the quarter-wave resonator can be coupled to a CPW feeding line.

Model Definition

Figure 2 shows the schematic cross section of the CPW line used for the resonator and the feed line. The impedance of CPW is related to the dielectric constant of the substrate and the ratio between the center conductor and gap widths. The conductive regions are treated as perfect conductors to capture the lossless behavior of the superconducting metal, and these regions are also treated as 2D layer because they are much thinner than any other relevant length scales in the model. The substrate is silicon with a relative permittivity of 11.7.

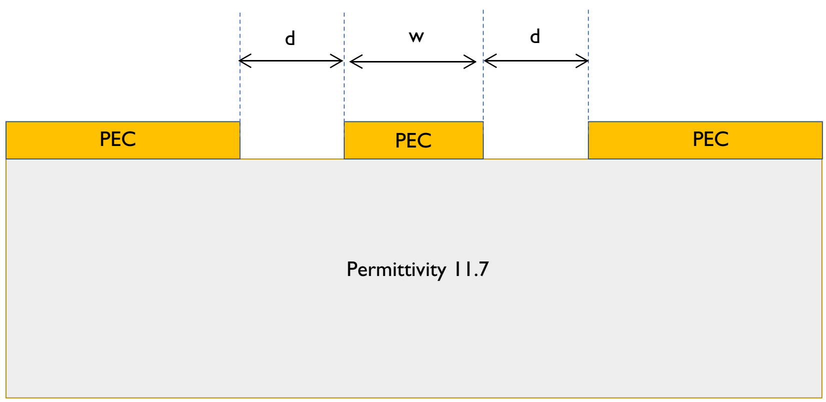


Figure 2: Schematic of the CPW cross section where $w/d=7/4$ and the characteristic impedance is 50 Ω.

Numeric ports are used to excite and terminate the feeding CPW line. Therefore, the boundary conditions on those surfaces are the corresponding mode fields. Scattering boundary conditions are used on the remaining boundaries, although due to the highly confined mode structure of CPW this results in very little loss in the system and an extremely high the quality. To model this system in a computationally efficient manner, an adaptive frequency sweep is used. This is because very fine frequency resolution is required to capture the narrow bandwidth of the resonance, and an adaptive frequency sweep allows evaluation of many frequency points without explicit calculation of each and every one based on asymptotic waveform evaluation (AWE).

To further reduce the model size, no field solution is stored for the adaptive frequency sweep study in this model. By default, COMSOL stores the electromagnetic field values for the entire computational domain for each frequency evaluated, but because of the large number of frequencies accounted for in the adaptive frequency sweep, this can result in a massive file size. Only the S-parameters are of interest for the AWE and they are stored in the global S-parameter variables through the port feature.

Results and Discussion

Figure 3 shows the S-parameters of the system. The behavior of the quarter-wave resonator can be seen in the strong resonant reflectivity in S_{11}dB, whereas there feed line is highly transmissive away from the resonance frequency. This is further illustrated in Figure 4, which shows the standing wave formation in the CPW resonator. There is a large electric field enhancement at the open end and zero field at the shorted end, consistent with the anticipated behavior of the boundary conditions.

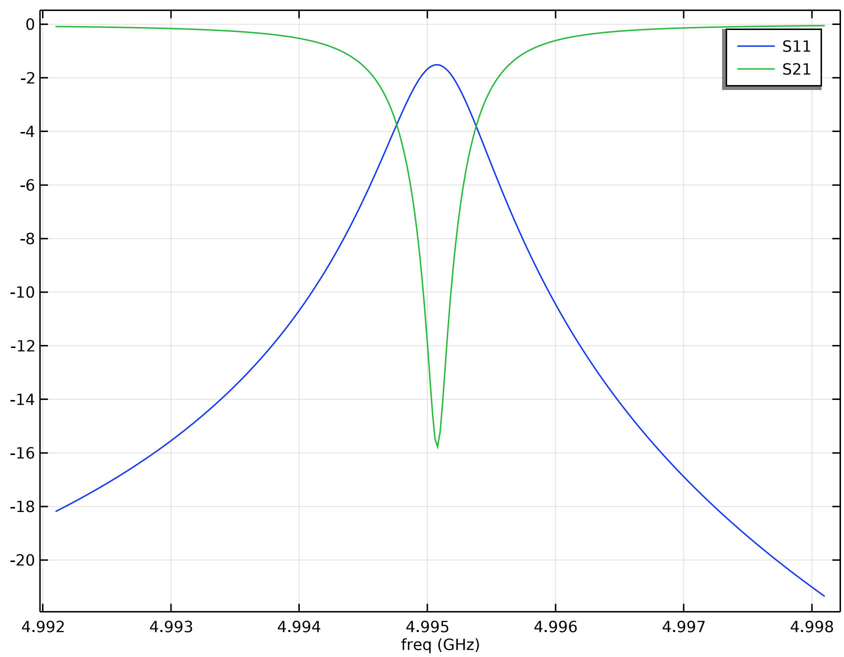


Figure 3: The S-parameters plot demonstrates a very narrow resonance behavior.

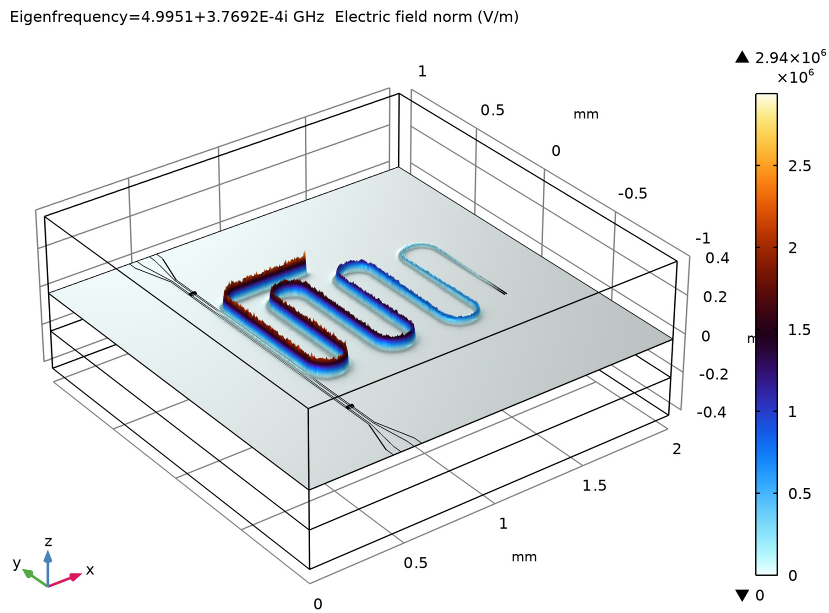


Figure 4: Illustration of standing wave pattern formed within the resonator. The height distribution corresponds to the total electric field. Antinode and node can be observed at the open and short ends.

Notes About the COMSOL Implementation

In this model we conduct two studies. The first is an eigenfrequency simulation to find the resonant frequency of the structure. The second performs a frequency sweep +/- 3 MHz around this point to calculate the S-Parameters for the feed line.

Since the CPW resonator is a very high-quality factor system, it can be a challenging structure to simulate. High-Q systems can be extremely mesh sensitive, and a mesh refinement study is necessary to ensure reliable results. Very fine meshes in turn require more memory, which may require fine tuning of the solver settings for large models. For this demonstration, you can obtain a reasonable mesh using the **Refine conductive edges** feature and a size setting close to the dielectric gap width. In your own modeling, we recommend performing a mesh refinement investigation. Inherent in a mesh refinement study is a tradeoff between computational resources and accuracy, and so an important question to consider is exactly how accurately the resonance frequency needs to be known.

Slight changes to the mesh used here can result in shifts on the order of ~1 MHz, and the model takes ~20 GB of memory to solve. As a result of this large model size, the Eigenfrequency solver settings require slight modifications to converge.

Application Library path: `RF_Module/Filters/cpw_resonator`

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 **Empty Study**.

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**.

Work Plane 1 (wp1)

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

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

Work Plane 1 (wp1) > Plane Geometry

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

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Part Libraries** window, select **RF Module** > **Coplanar Waveguides** > **cpw_straight_onchip** in the tree.

3 Click **Add to Geometry**.

GEOMETRY 1

Work Plane 1 (wp1) > Coplanar Waveguide Trace, Straight, On Chip 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** click **Coplanar Waveguide Trace, Straight, On Chip 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
w_center	14[um]	0.014 mm	Width of center conductor
w_slot	8[um]	0.008 mm	Width of slot
l_cpw	1400[um]	1.4 mm	Length of CPW

4 Locate the **Position and Orientation of Output** section. In the **Rotation angle** text field, type 90.

5 In the **xw-displacement** text field, type `0.515`.

6 Click **Build Selected**.

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** click **Plane Geometry**.

3 In the **Part Libraries** window, select **RF Module** > **Coplanar Waveguides** > **cpw_transition_onchip** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Work Plane 1 (wp1) > Coplanar Waveguide Trace, Transition, On Chip 1 (pi2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** click **Coplanar Waveguide Trace, Transition, On Chip 1 (pi2)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
w_center_n	14[um]	0.014 mm	Width of center conductor, narrow
w_center_w	70[um]	0.07 mm	Width of center conductor, wide
w_slot_n	8[um]	0.008 mm	Width of slot, narrow
w_slot_w	40[um]	0.04 mm	Width of slot, wide
l_cpw_n	50[um]	0.05 mm	Length of CPW, narrow
l_cpw_w	50[um]	0.05 mm	Length of CPW, wide
l_cpw_t	200[um]	0.2 mm	Length of CPW, transition

4 Locate the **Position and Orientation of Output** section. In the **xw-displacement** text field, type `0.515`.

5 In the **yw-displacement** text field, type `0.85`.

6 In the **Rotation angle** text field, type `90`.

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 object **pi2** 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**.

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

Work Plane 1 (wp1) > Coplanar Waveguide Trace, Straight, On Chip 2 (pi3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** right-click **Coplanar Waveguide Trace, Straight, On Chip 1 (pi1)** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
w_center	7[um]	0.007 mm	Width of center conductor
w_slot	4[um]	0.004 mm	Width of slot
l_cpw	396[um]	0.396 mm	Length of CPW
termination2	2	2	0 (through), 1 (short), 2 (open)

4 Locate the **Position and Orientation of Output** section. In the **xw-displacement** text field, type `0.834`.

5 In the **yw-displacement** text field, type `0.4425`.

6 In the **Rotation angle** text field, type `0`.

7 Click **Build Selected**.

Work Plane 1 (wp1) > Coplanar Waveguide Trace, Straight, On Chip 3 (pi4)

1 Right-click **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** > **Coplanar Waveguide Trace, Straight, On Chip 2 (pi3)** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
l_cpw	500[um]	0.5 mm	Length of CPW
termination2	0	0	0 (through), 1 (short), 2 (open)

4 Locate the **Position and Orientation of Output** section. In the **xw-displacement** text field, type `0.5435`.

5 In the **yw-displacement** text field, type `0`.

6 In the **Rotation angle** text field, type `90`.

7 Click **Build Selected**.

Work Plane 1 (wp1) > Array 1 (arr1)

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

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

3 Select the object **pi4** only.

4 In the **Settings** window for **Array**, locate the **Size** section.

5 In the **xw size** text field, type `6`.

6 Locate the **Displacement** section. In the **xw** text field, type `0.185`.

7 Click **Build Selected**.

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

Work Plane 1 (wp1) > Coplanar Waveguide Trace, Straight, On Chip 4 (pi5)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** right-click **Coplanar Waveguide Trace, Straight, On Chip 3 (pi4)** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
l_cpw	100[um]	0.1 mm	Length of CPW

4 Locate the **Position and Orientation of Output** section. In the **yw-displacement** text field, type `0.3`.

5 Click **Build Selected**.

PART LIBRARIES

1 In the **Work Plane** toolbar, click **Part Libraries**.

2 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** click **Plane Geometry**.

3 In the **Part Libraries** window, select **RF Module** > **Coplanar Waveguides** > **cpw_90_round_bend_onchip** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Work Plane 1 (wp1) > Coplanar Waveguide Trace, 90-Degree Round-Bend, On Chip 1 (pi6)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** click **Coplanar Waveguide Trace, 90-Degree Round-Bend, On Chip 1 (pi6)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
w_center	7[um]	0.007 mm	Width of center conductor

Name	Expression	Value	Description
w_slot	4[um]	0.004 mm	Width of slot
r_center	92.5[um]	0.0925 mm	Bend radius of center conductor

4 Locate the **Position and Orientation of Output** section. In the **xw-displacement** text field, type 0.636.

5 In the **yw-displacement** text field, type 0.35.

6 In the **Rotation angle** text field, type 90.

7 Click **Build Selected**.

PART LIBRARIES

1 In the **Work Plane** toolbar, click **Part Libraries**.

2 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** click **Plane Geometry**.

3 In the **Part Libraries** window, select **RF Module** > **Coplanar Waveguides** > **cpw_180_round_bend_onchip** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Work Plane 1 (wp1) > Coplanar Waveguide Trace, 180-Degree Round-Bend, On Chip 1 (pi7)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** click **Coplanar Waveguide Trace, 180-Degree Round-Bend, On Chip 1 (pi7)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
w_center	7[um]	0.007 mm	Width of center conductor
w_slot	4[um]	0.004 mm	Width of slot
r_center	92.5[um]	0.0925 mm	Bend radius of center conductor

4 Locate the **Position and Orientation of Output** section. In the **xw-displacement** text field, type 0.636.

5 In the **yw-displacement** text field, type -0.25.

6 In the **Rotation angle** text field, type -90.

7 Click **Build Selected**.

Work Plane 1 (wp1) > Array 2 (arr2)

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

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

3 Select the object **pi7** only.

4 In the **Settings** window for **Array**, locate the **Size** section.

5 In the **xw size** text field, type `3`.

6 Locate the **Displacement** section. In the **xw** text field, type `0.37`.

7 Click **Build Selected**.

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

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

2 Select the objects **arr2(1,1)**, **arr2(2,1)**, and **arr2(3,1)** 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 Locate the **Center of Rotation** section. In the **xw** text field, type `1.0985`.

7 Click **Build Selected**.

Work Plane 1 (wp1) > Coplanar Waveguide Trace, Straight, On Chip 5 (pi8)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** right-click **Coplanar Waveguide Trace, Straight, On Chip 3 (pi4)** and choose **Duplicate**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
termination1	1	1	0 (through), 1 (short), 2 (open)
termination2	0	0	0 (through), 1 (short), 2 (open)

4 Locate the **Position and Orientation of Output** section. In the **xw-displacement** text field, type `1.6535`.

5 Click **Build Selected**.

Block 1 (blk1)

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

2 In the **Settings** window for **Block**, locate the **Position** section.

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

4 Locate the **Size and Shape** section. In the **Width** text field, type `2`.

5 In the **Depth** text field, type `2`.

6 In the **Height** text field, type `0.8`.

7 Locate the **Position** section. In the **x** text field, type `1`.

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

Layer name	Thickness (mm)
Layer 1	0.2
Layer 2	0.2

9 Click **Build All Objects**.

Block 2 (blk2)

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

2 In the **Settings** window for **Block**, locate the **Position** section.

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

4 In the **Model Builder** window, click **Block 2 (blk2)**.

5 Locate the **Size and Shape** section. In the **Width** text field, type `0.03`.

6 In the **Depth** text field, type `0.01`.

7 In the **Height** text field, type `0.01`.

8 Locate the **Position** section. In the **x** text field, type `0.515`.

9 In the **z** text field, type `0.005`.

10 In the **y** text field, type `-0.595`.

11 Click **Build Selected**.

Mirror 1 (mir1)

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

2 Select the object **blk2** only.

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

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

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

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

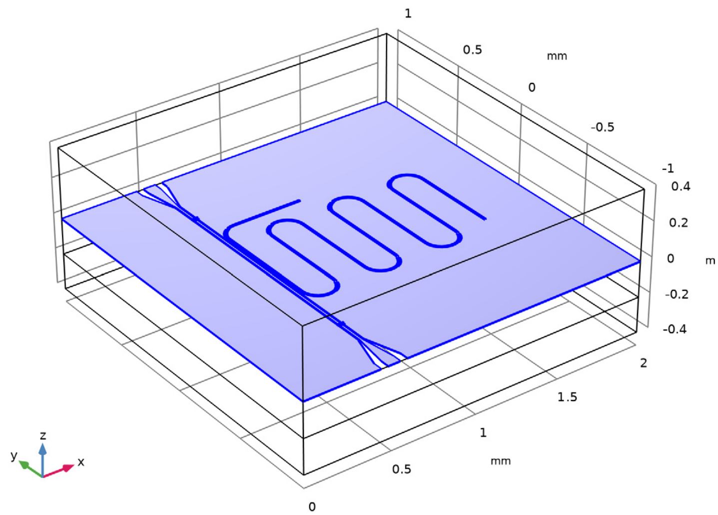
7 Click **Build Selected**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Perfect Electric Conductor 2

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

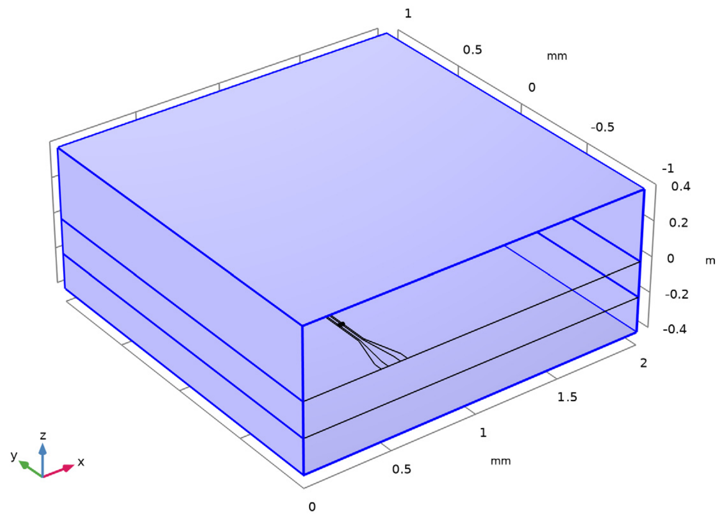
2 Select Boundaries 9, 16–18, 21, 24, 27, 29–31, and 37–39 only.



Scattering Boundary Condition 1

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

2 Select Boundaries 1, 3, 4, 7, 10, and 41–43 only.

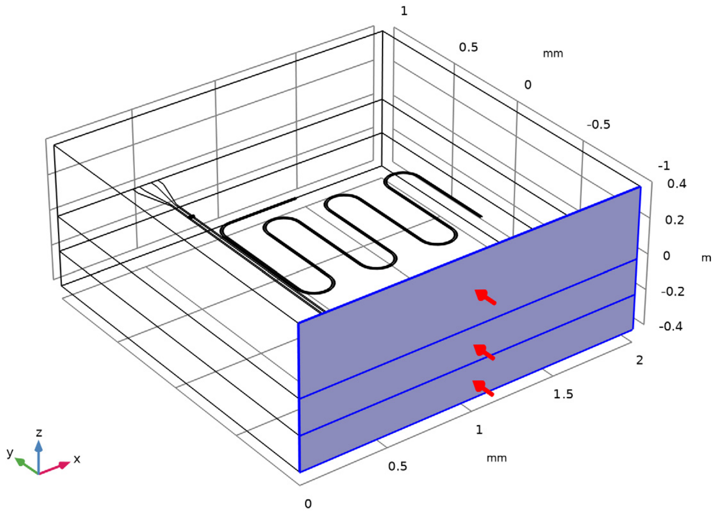


Port 1

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

To excite the CPW we need the mode profile of the electromagnetic fields. Since there is no analytical equation to define these fields, we use numeric ports and calculate them as a preprocessing step using two Boundary Mode Analysis study steps. The field distributions obtained are used for both the eigenfrequency and frequency domain analysis. Since a quasi-TEM wave is propagating on a CPW, use the **Analyze as a TEM field** option and define **Integration Line for Voltage**.

2 Select Boundaries 2, 5, and 8 only.



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

4 From the **Type of port** list, choose **Numeric**.

5 Select the **Analyze as a TEM field** checkbox.

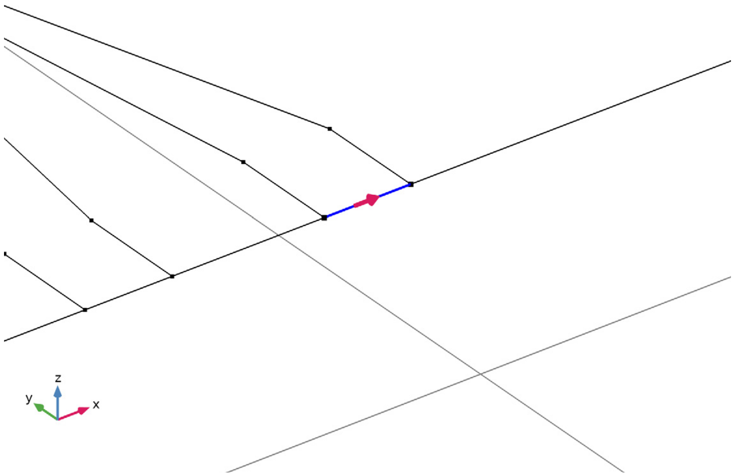
Integration Line for Voltage 1

1 In the **Physics** toolbar, click **Attributes** and choose **Integration Line for Voltage**.

2 In the **Settings** window for **Integration Line for Voltage**, locate the **Edge Selection** section.

3 Click **Clear Selection**.

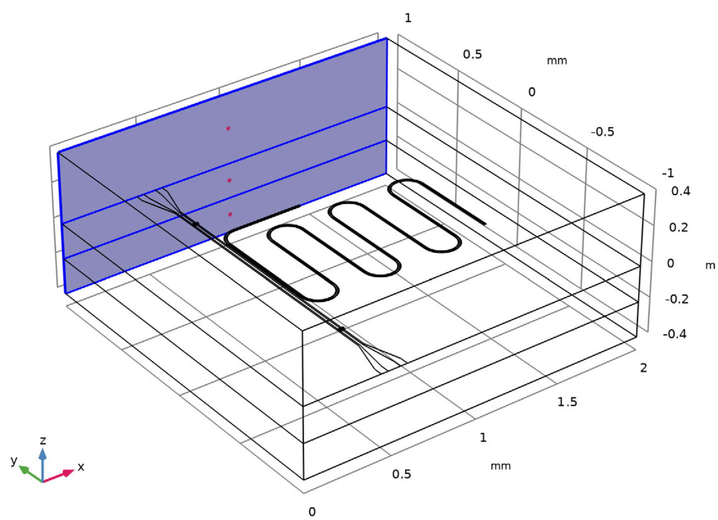
4 Select Edge 104 only.



Port 2

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

2 Select Boundaries 11–13 only.



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

4 From the **Type of port** list, choose **Numeric**.

5 Select the **Analyze as a TEM field** checkbox.

Integration Line for Voltage 1

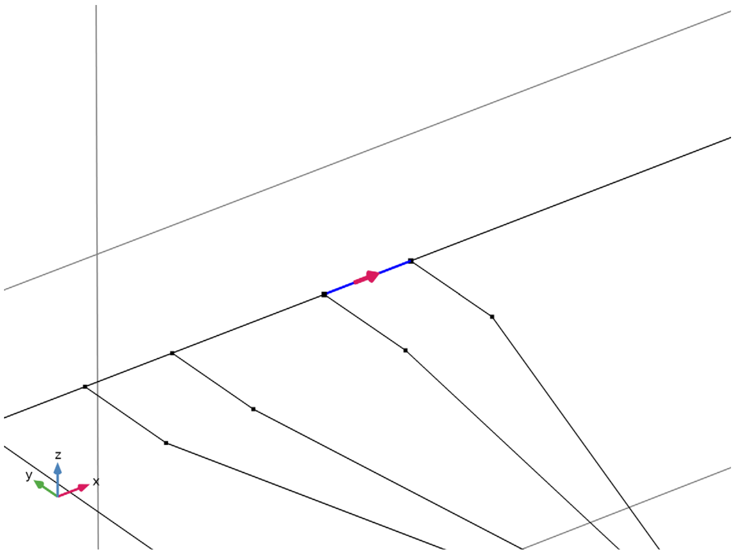
1 In the **Physics** toolbar, click **Attributes** and choose **Integration Line for Voltage**.

2 In the **Settings** window for **Integration Line for Voltage**, locate the **Edge Selection** section.

3 Click **Clear Selection**.

4 Select Edge 106 only.

5 Locate the **Settings** section. Click **Toggle Voltage Drop Direction**.



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.

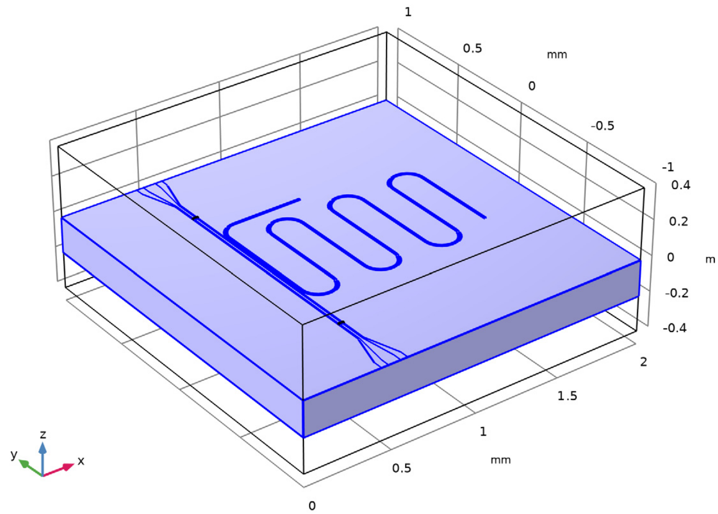
5 In the tree, select **Built-in** > **Silicon**.

6 Click the **Add to Component** button in the window toolbar.

7 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Silicon (mat2)
Select Domain 2 only.



Field is confined in the close vicinity of the CPW gaps. Use **Refine conductive edges** to refine the mesh in the vicinity of CPW gap.

MESH 1

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

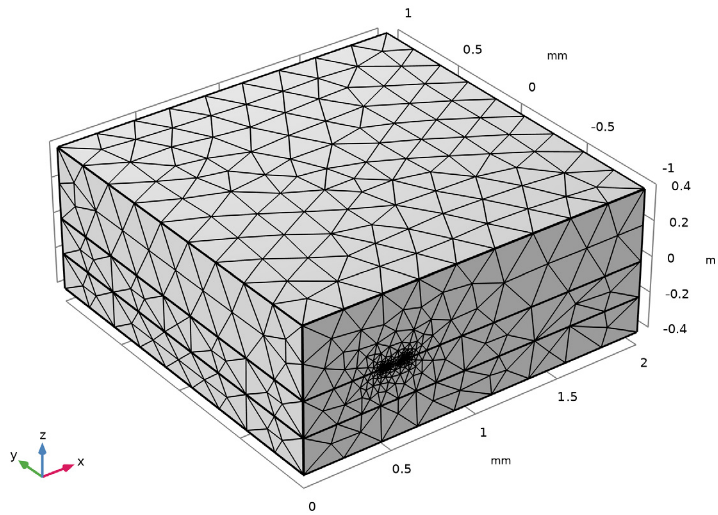
2 In the **Settings** window for **Mesh**, locate the **Electromagnetic Waves, Frequency Domain (emw)** section.

3 Select the **Refine conductive edges** checkbox.

4 From the **Size type** list, choose **User defined**.

5 In the **Size** text field, type `5[um]`.

6 Click **Build All**.



To see the mesh structure on the CPW surface, Use **Hide for Physics**.

DEFINITIONS

Hide for Physics 1

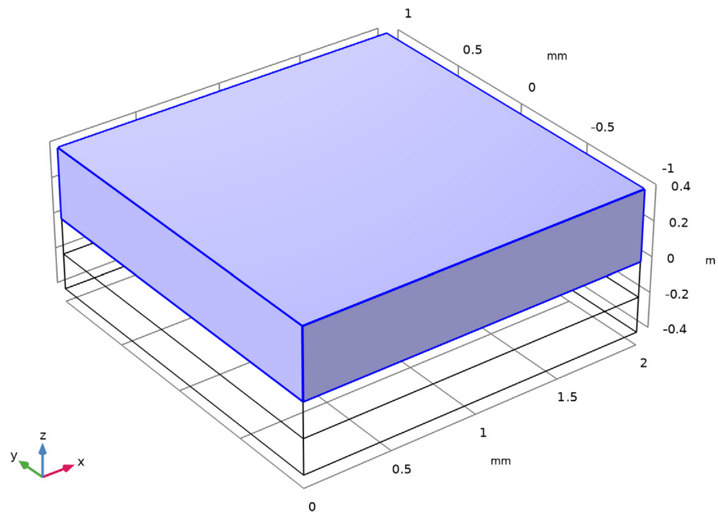
1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

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.

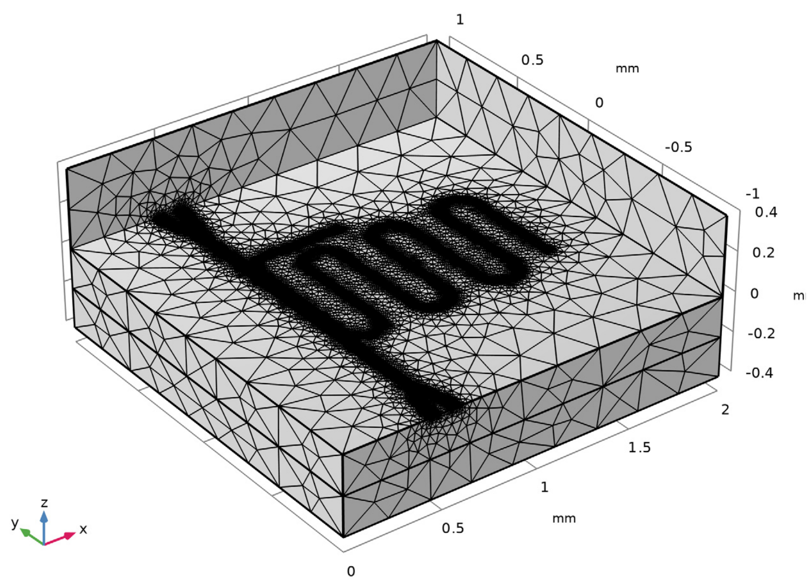
4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 7, 8, and 10 only.



MESH 1

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



Here we perform the **Boundary Mode Analysis** to calculate the mode profiles. The field distributions obtained will be used for both the **Eigenfrequency** and **Adaptive Frequency Sweep**.

STUDY 1

Step 1: Boundary Mode Analysis

1 In the **Study** toolbar, click **More Study Steps** and choose **Other** > **Boundary Mode Analysis**.

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

3 In the **Mode analysis frequency** text field, type `5[GHz]`.

4 In the **Search for modes around shift** text field, type `2.5217`.

Step 2: Boundary Mode Analysis 1

1 Right-click **Step 1: Boundary Mode Analysis** and choose **Duplicate**.

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

3 In the **Port name** text field, type `2`.

Step 3: Eigenfrequency

1 In the **Study** toolbar, click **More Study Steps** and choose **Eigenfrequency** > **Eigenfrequency**.

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

3 In the **Search for eigenfrequencies around shift** text field, type `4.94[GHz]`.

4 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `1`.

5 From the **Search method around shift** list, choose **Larger real part**.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

For this specific example, using the combination of boundary mode analysis and eigenfrequency with a case of high-contrast material properties, such as the combination of air and silicon, one can take advantage of the Vanka presmoother in the settings of Eigenvalue Solver to achieve faster convergence and reduce computational time.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Eigenvalue Solver 3** node.

4 Right-click **Study 1 > Solver Configurations > Solution 1 (sol1) > Eigenvalue Solver 3 > Suggested Iterative Solver (emw) 2** and choose **Enable**.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Multislice 1

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

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. From the **Color table** list, choose **ThermalWaveDark**.

Deformation 1

1 Right-click **Multislice 1** and choose **Deformation**.

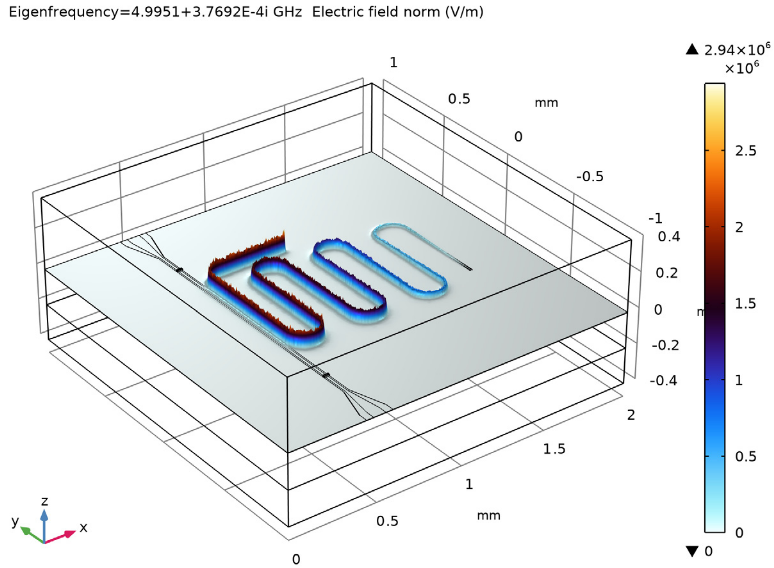
2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **X-component** text field, type `0`.

4 In the **Y-component** text field, type `0`.

5 In the **Z-component** text field, type `emw.normE`.

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

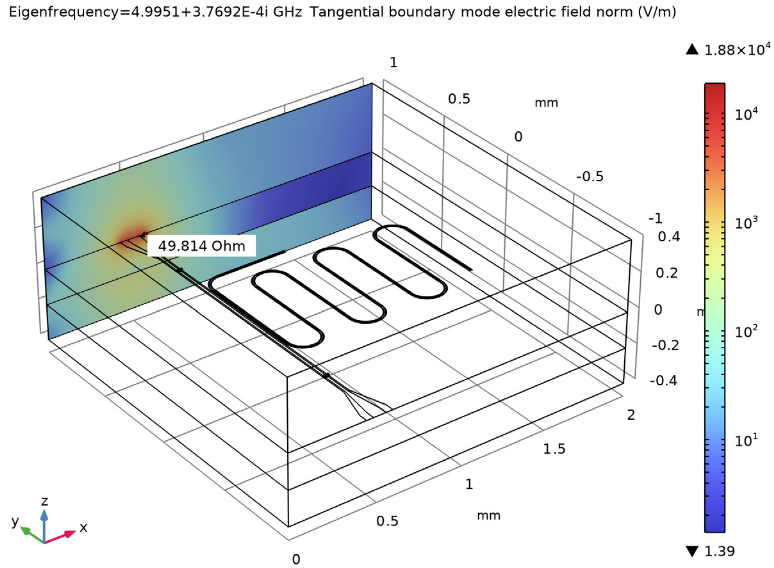


Surface 1

1 In the **Model Builder** window, expand the **Electric Mode Field, Port 2 (emw)** node, then click **Surface 1**.

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

3 From the **Scale** list, choose **Logarithmic**.



ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

4 Click the **Add Study** button in the window toolbar.

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

STUDY 1

Step 1: Boundary Mode Analysis, Step 2: Boundary Mode Analysis 1

Right-click and choose **Copy**.

STUDY 2

1 In the **Model Builder** window, right-click **Study 2** and choose **Paste Multiple Items**.

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

3 Clear the **Generate default plots** checkbox.

Step 3: Adaptive Frequency Sweep

1 In the **Study** toolbar, click **More Study Steps** and choose **Frequency Domain > Adaptive Frequency Sweep**.

Use the evaluated resonant frequency for frequency sweep study settings that can be directly copied from the **Table 1**. Right click on the frequency value in the **Table 1** to access the context menu and choose **Copy Cell to Clipboard** for use later.

In the **Frequencies** input field, sweep +-3MHz around the resonant frequency with 0.02MHz step. For instance, if the computed resonant frequency is 5GHz, then type `range(5[GHz]-3[MHz],0.02[MHz],5[GHz]+3[MHz])`.

Since the CPW resonator has a very sharp resonance, the **Adaptive Frequency Sweep** can be utilized to reduce computational time. A good choice for the **Asymptotic Waveform Evaluation (AWE) expressions** increases the efficiency of adaptive frequency sweep. Magnitude of S11 is a suitable choice for this problem to decrease computational cost.

The physics-controlled mesh uses the highest frequency value in the specified range. Since the model is a high-Q narrow-band device, it would be sensitive to a small mesh change by choosing a different frequency range unless a finer mesh is set with a smaller value of **Relative size to default mesh** in **Refine conductive edges** option in the mesh settings.

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

3 From the **AWE expression type** list, choose **User controlled**.

4 In the table, enter the following settings:

Asymptotic waveform evaluation (AWE) expressions
`abs(comp1.emw.S11)`

5 Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	Output
Electromagnetic Waves, Frequency Domain (emw)	None

Only the S-parameters are of interest for this study. They are stored in the global S-parameter variables through the port features, so there is no need to store the field solution. Since the adaptive frequency sweep runs with a fine frequency resolution, this choice helps reduce the saved file size.

6 In the **Study** toolbar, click **Compute**.

RESULTS

S-parameter

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

2 In the **Settings** window for **1D Plot Group**, type `S-parameter` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Probe Solution 4 (sol4)**.

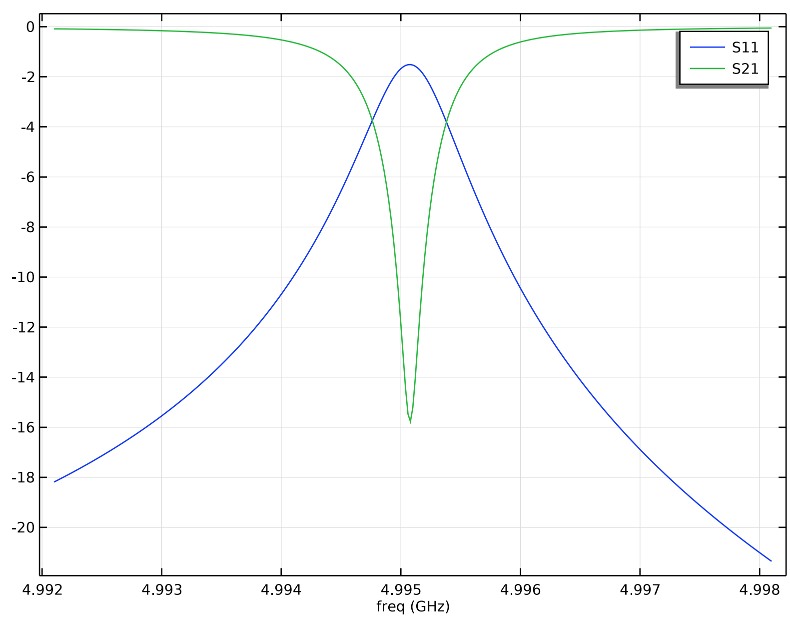
Global 1

1 Right-click **S-parameter** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Ports** > **S-parameter, dB - dB** > **emw.S11dB - S11**.

3 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** > **Ports** > **S-parameter, dB - dB** > **emw.S21dB - S21**.

4 In the **S-parameter** toolbar, click **Plot**.



The following instruction shows how to use the **Graph Marker** subfeature to analyze 1D plots. When plotting S21 of a bandstop filter, the -10dB attenuation bandwidth of the stopband can be computed with a graph marker. Use an additional graph marker on the S11 plot to check the maximum reflection level.

S-parameter with Graph Markers

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

2 In the **Settings** window for **1D Plot Group**, type `S-parameter with Graph Markers` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Probe Solution 4 (sol4)**.

Global 1

1 Right-click **S-parameter with Graph Markers** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Ports** > **S-parameter, dB - dB** > **emw.S11dB - S11**.

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 **Max**.

4 Locate the **Text Format** section. In the **Precision** text field, type `4`.

5 Select the **Show x-coordinate** checkbox.

6 Select the **Include unit** checkbox.

7 In the **S-parameter with Graph Markers** toolbar, click **Plot**.

Global 2

1 In the **Model Builder** window, right-click **S-parameter with Graph Markers** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Ports** > **S-parameter, dB - dB** > **emw.S21dB - S21**.

Graph Marker 1

1 Right-click **Global 2** and choose **Graph Marker**.

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

3 From the **Display mode** list, choose **Bandwidth**.

4 From the **Range type** list, choose **Stopband**.

5 In the **Cutoff value** text field, type -10.

6 Locate the **Text Format** section. In the **Precision** text field, type 3.

7 Select the **Include unit** checkbox.

8 Click to expand the **Coloring and Style** section. From the **Orientation** list, choose **Vertical**.

9 Select the **Show frame** checkbox.

10 In the **S-parameter with Graph Markers** toolbar, click **Plot**.