Created in COMSOL Multiphysics 6.2

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 Planck's 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.

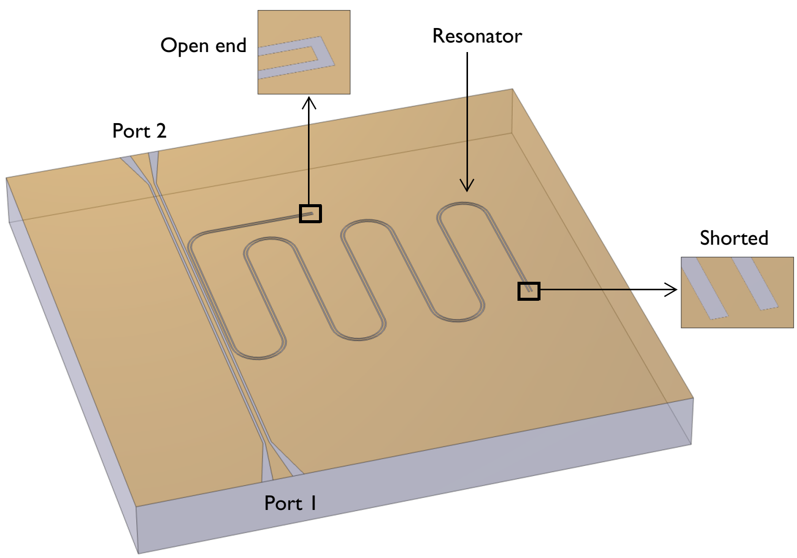


Figure 1: A CPW resonator coupled to a CPW transmission line. The air domains are removed for a better view.

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.

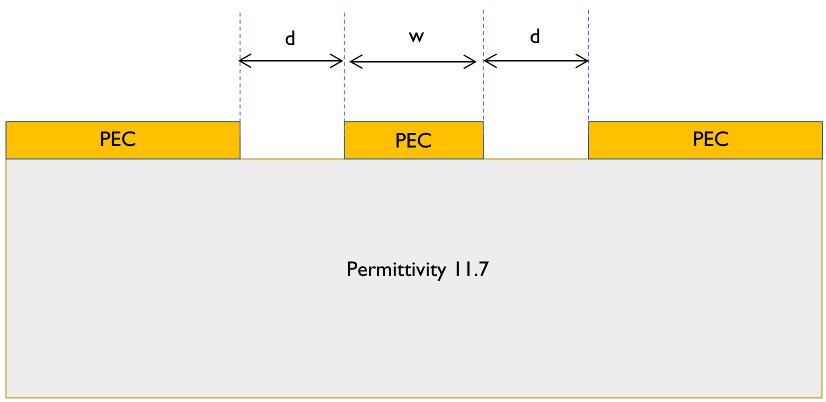


Figure 2: Schematic of the CPW cross section where w/d=7/4 and the characteristic impedance is 50 Ω.

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.

To further reduce the model size, the field distribution is only stored on the CPW filter surface. 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. Since we are mainly interested in the field distribution on the CPW filter surface, we choose to only store field for that surface, and this can be done through the use of geometric selections in the study settings.

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.

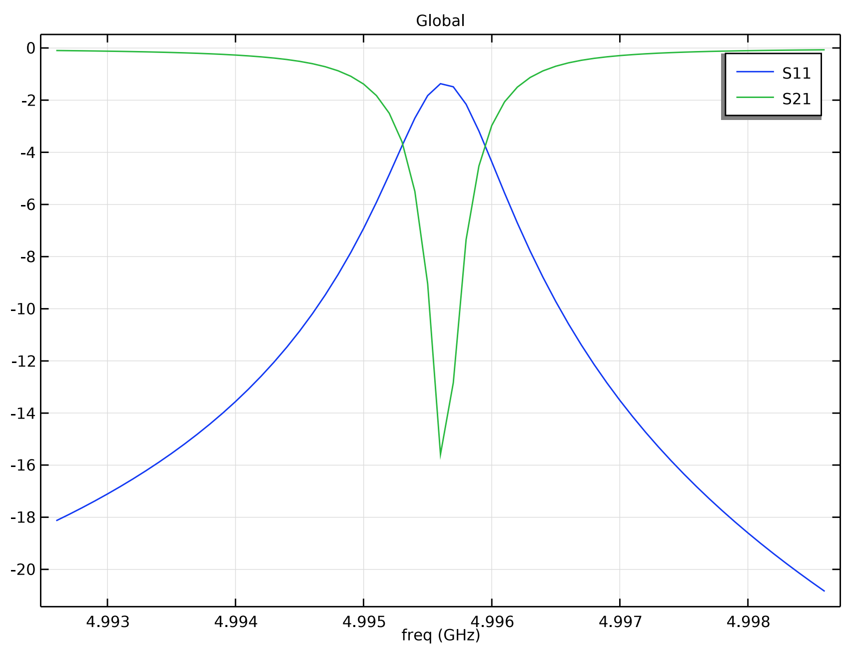


Figure 3: The S-parameters plot demonstrates a very narrow resonance behavior.

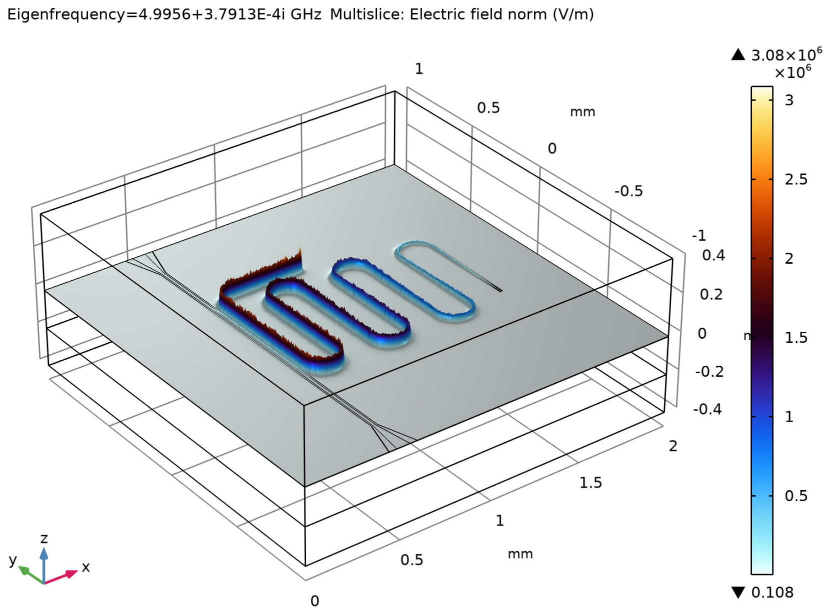


Figure 4: Illustration of standing wave pattern formed within the resonator. The height distribution corresponds to the total electric field. Antinode and node can be observed at the open and short ends.

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

To focus on the model physics and implementation, the import geometry feature is used to build the structure. Because the structure is highly sensitive to the mesh, and the mesh is in turn sensitive to the geometry, it is important to use the **COMSOL kernel** to exactly reproduce the results in this example. The **CAD kernel** could be used as well, however a mesh refinement process should be performed accordingly as the resonance may shift slightly despite identical mesh settings.

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**.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `cpw_resonator.mphbin`.

5 Click **Import**.

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, select **Physics>Advanced Physics Options** in the tree.

3 In the tree, select the check box for the node **Physics>Advanced Physics Options**.

4 Click **OK**.

5 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

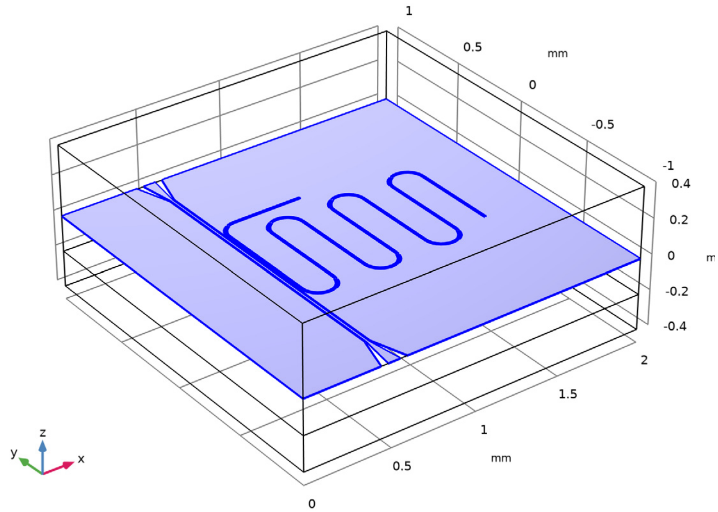
6 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, click to expand the **Port Options** section.

7 From the **Port formulation** list, choose **Constraint-based**.

Perfect Electric Conductor 2

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

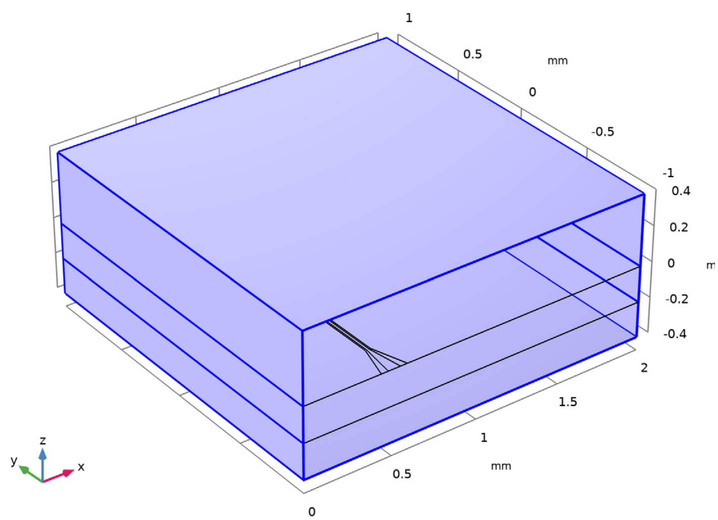
2 Select Boundaries 9, 15, and 17 only.



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 19–21 only.

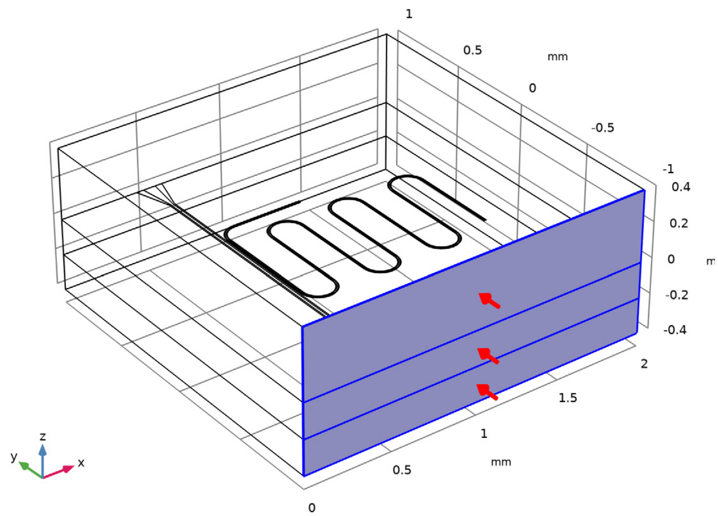


Port 1

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** check box.

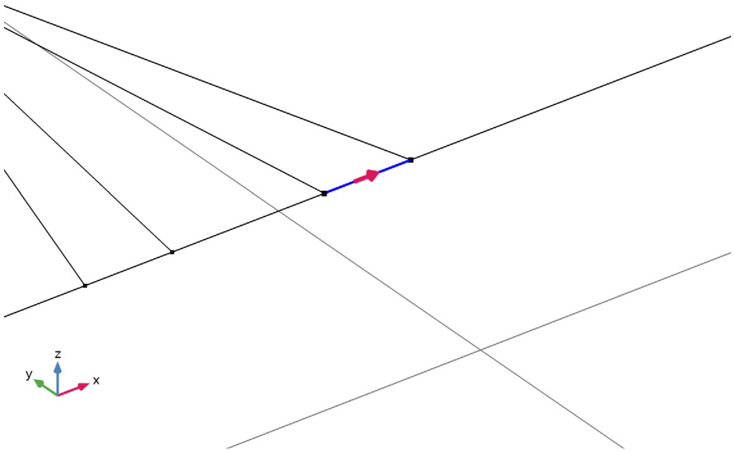
Integration Line for Voltage 1

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**.

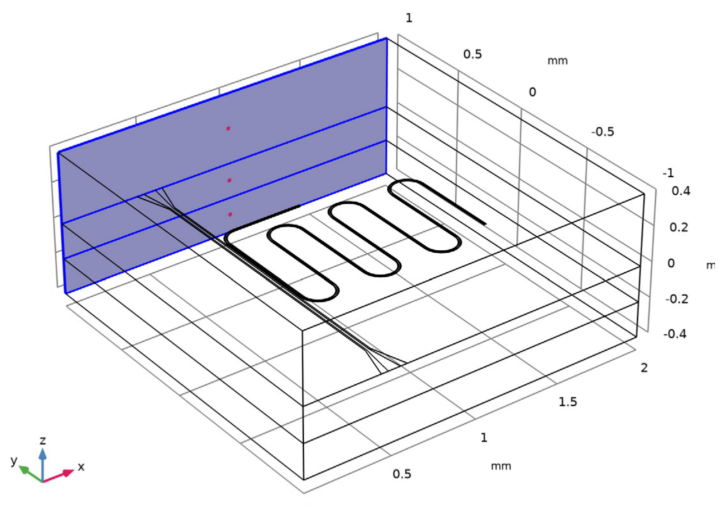
4 Select Edge 47 only.



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** check box.

Integration Line for Voltage 1

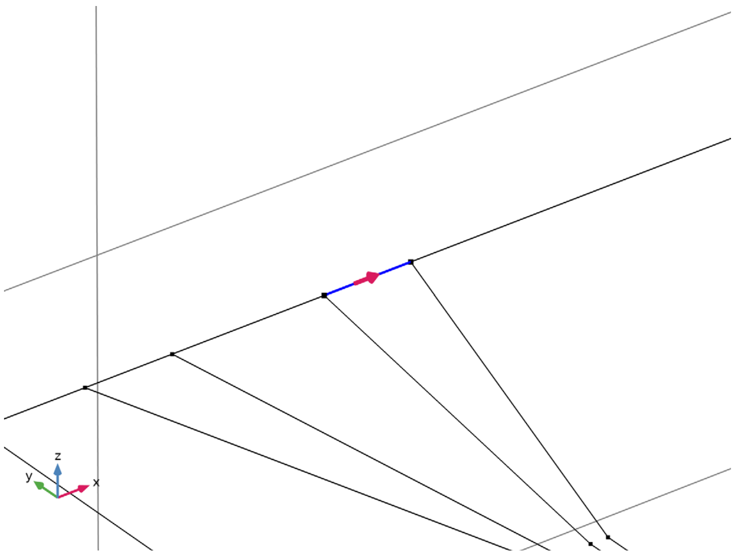
1 In the **Physics** toolbar, click **Attributes** and choose **Integration Line for Voltage**.

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

3 Click **Clear Selection**.

4 Select Edge 48 only.

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



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 tree, select **Built-in>Silicon**.

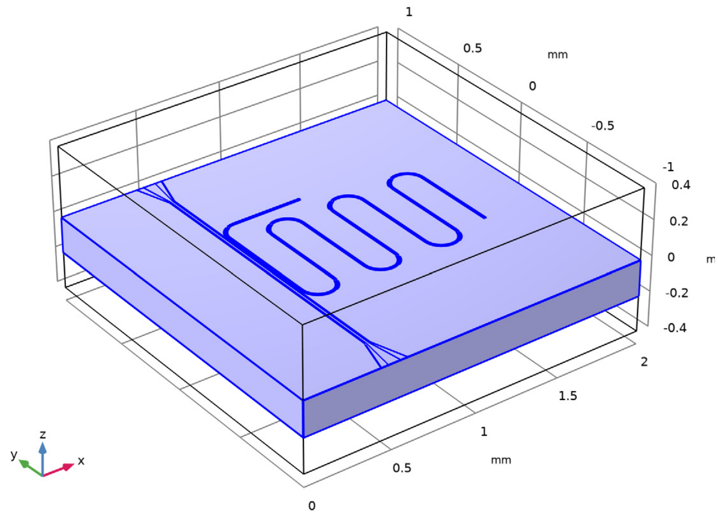
6 Click **Add to Component** in the window toolbar.

7 In the **Home** 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**.

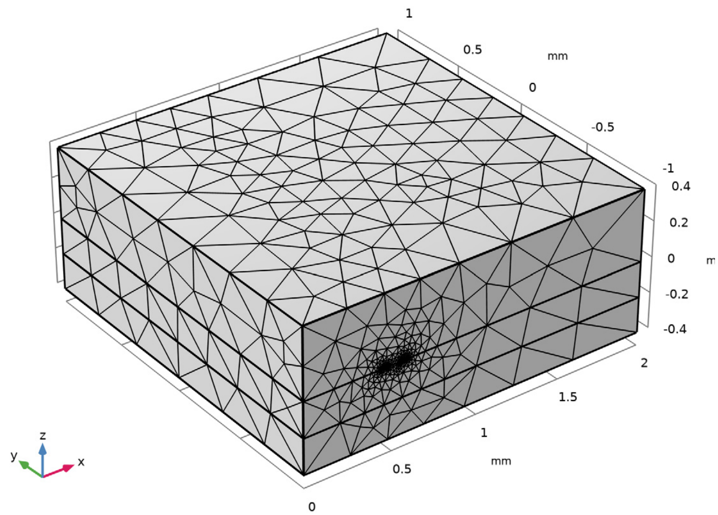
2 In the **Settings** window for **Mesh**, locate the **Electromagnetic Waves, Frequency Domain (emw)** section.

3 Select the **Refine conductive edges** check box.

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

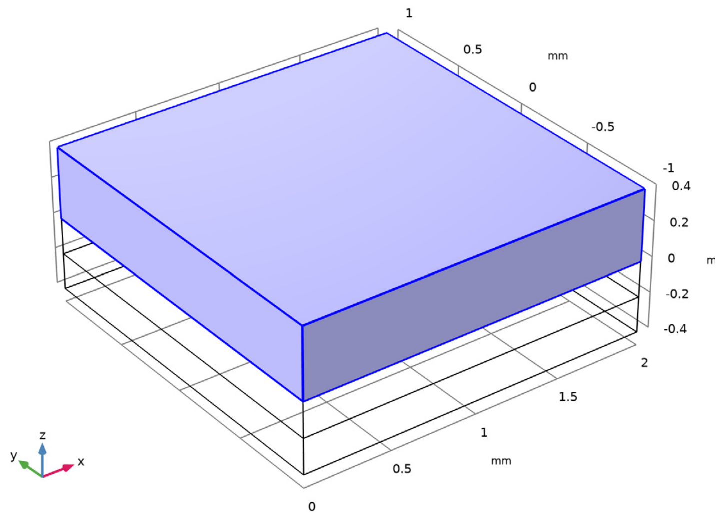
1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

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.

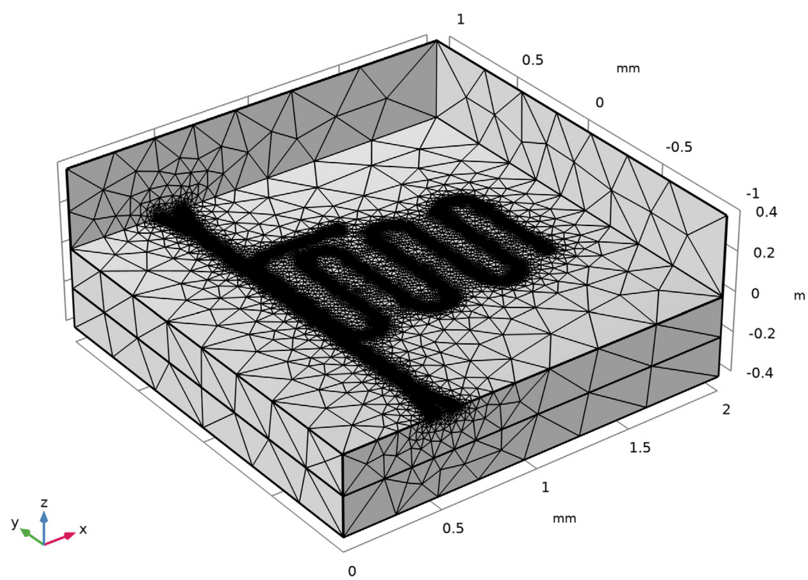
4 From the **Geometric entity level** list, choose **Boundary**.

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

Step 2: Boundary Mode Analysis 1

1. In the **Model Builder** window, click **Step 2: Boundary Mode Analysis 1**.
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 **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** check box. In the associated text field, type `1`.
5. From the **Search method around shift** list, choose **Larger real part**.

By default, COMSOL stores all field values for every spatial location and frequency considered in the model. For a densely meshed problem with a fine frequency sweep, the size of the automatically generated result file could be extremely large. To reduce the file size, we can select a spatial subset of the model the store only the values of interested. For this purpose, we create a geometric selection and to save only the values within that selection.

DEFINITIONS

Explicit 1

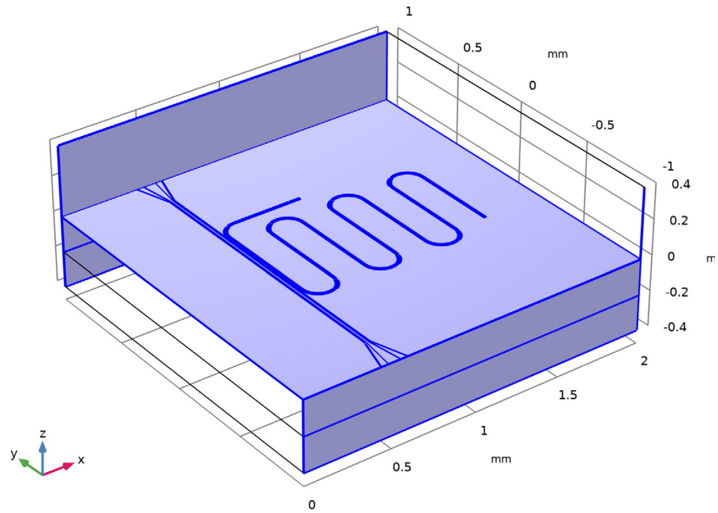
1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `2, 5, 8, 9, 11-18` in the **Selection** text field.

6 Click **OK**.



STUDY 1

Step 3: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 3: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, click to expand the **Store in Output** section.

3 In the table, enter the following settings:

Interface	Output
Electromagnetic Waves, Frequency Domain (emw)	Selection

4 Click to select row number 1 in the table.

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

6 In the **Add** dialog box, select **Explicit 1** in the **Selections** list.

7 Click **OK**.

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, one can take the advantage Vanka presmoother in the settings of Eigenvalue Solver to get a faster convergence and reduce computational time.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Eigenvalue Solver 3**.

3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1.0E-5`.

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

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

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

8 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Eigenvalue Solver 3>Suggested Iterative Solver (emw)>Multigrid 1>Presmoother** and choose **Vanka**.

9 In the **Settings** window for **Vanka**, locate the **Main** section.

10 In the **Number of iterations** text field, type `1`.

11 Under **Variables**, click **Add**.

12 In the **Add** dialog box, select **Electric field (comp1.E)** in the **Variables** list.

13 Click **OK**.

14 In the **Settings** window for **Vanka**, locate the **Main** section.

15 From the **Block solver** list, choose **Direct, stored factorization**.

16 In the **Relaxation factor** text field, type `1`.

17 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Eigenvalue Solver 3>Suggested Iterative Solver (emw)>Multigrid 1>Postsmoother** node, then click **SOR Vector 1**.

18 In the **Settings** window for **SOR Vector**, locate the **Main** section.

19 In the **Number of iterations** text field, type `1`.

20 In the **Relaxation factor** text field, type `0.5`.

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

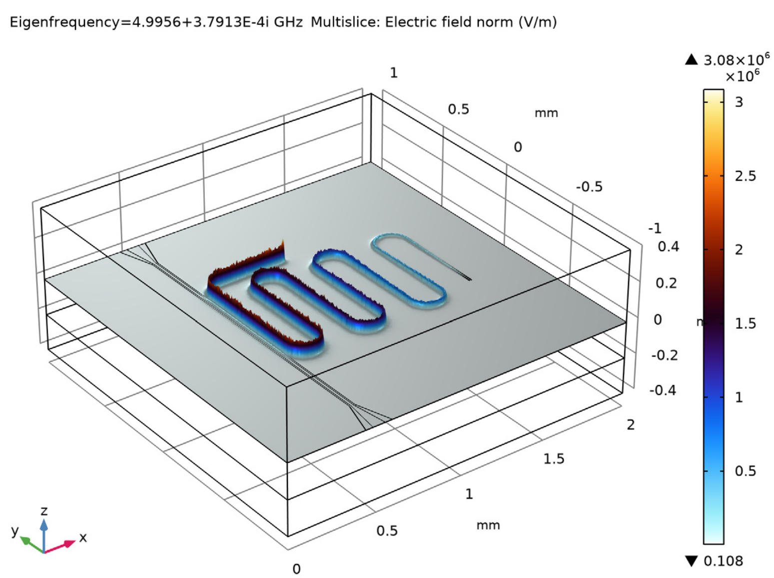
RESULTS

Multislice

1. In the **Model Builder** window, expand the **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**.

Deformation 1

1. Right-click **Multislice** 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**.

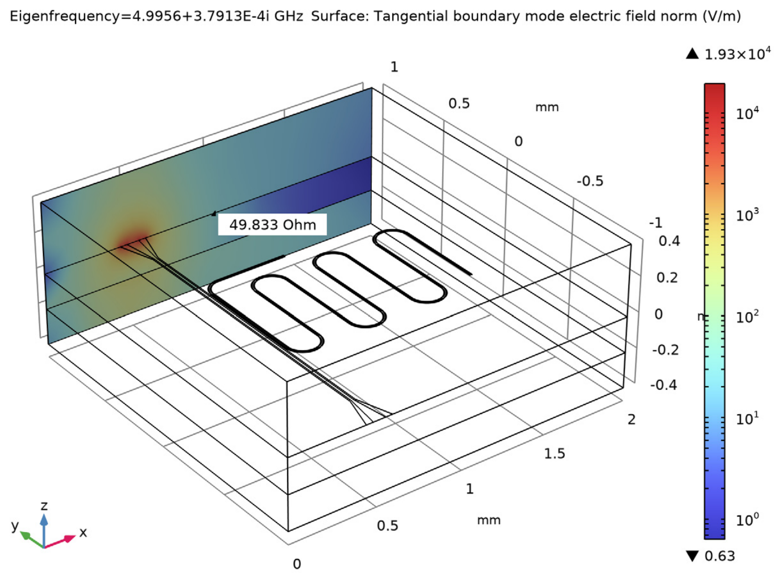


Surface 1

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 **Add Study** 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

1 In the **Model Builder** window, under **Study 1**, Ctrl-click to select **Step 1: Boundary Mode Analysis** and **Step 2: Boundary Mode Analysis 1**.

2 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 **Model Builder** window, click **Study 2**.

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

4 Clear the **Generate default plots** check box.

Step 3: Adaptive Frequency Sweep

1 In the **Study** toolbar, click **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 +-3[MHz] around the resonant frequency with 0.1[MHz] step. For instance, if the computed resonant frequency is 5[GHz], then type `range(5[GHz]-3[MHz],0.1[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)`

DEFINITIONS

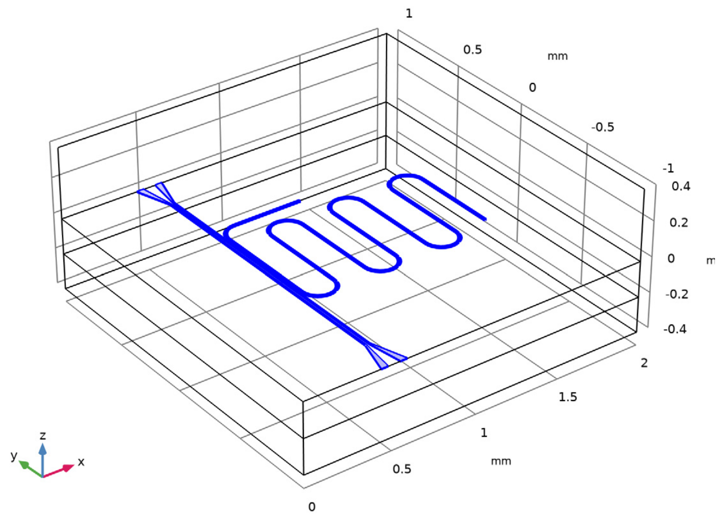
Explicit 2

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 14, 16, and 18 only.



STUDY 2

Step 3: Adaptive Frequency Sweep

1. In the **Model Builder** window, under **Study 2** click **Step 3: Adaptive Frequency Sweep**.
2. In the **Settings** window for **Adaptive Frequency Sweep**, click to expand the **Store in Output** section.
3. In the table, enter the following settings:

Interface	Output
Electromagnetic Waves, Frequency Domain (emw)	Selection

4. Click to select row number 1 in the table.
5. Under **Selections**, click **Add**.
6. In the **Add** dialog box, select **Explicit 2** in the **Selections** list.
7. Click **OK**.
8. In the **Home** toolbar, click **Compute**.

RESULTS

S-parameter

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` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 4 (sol4)**.

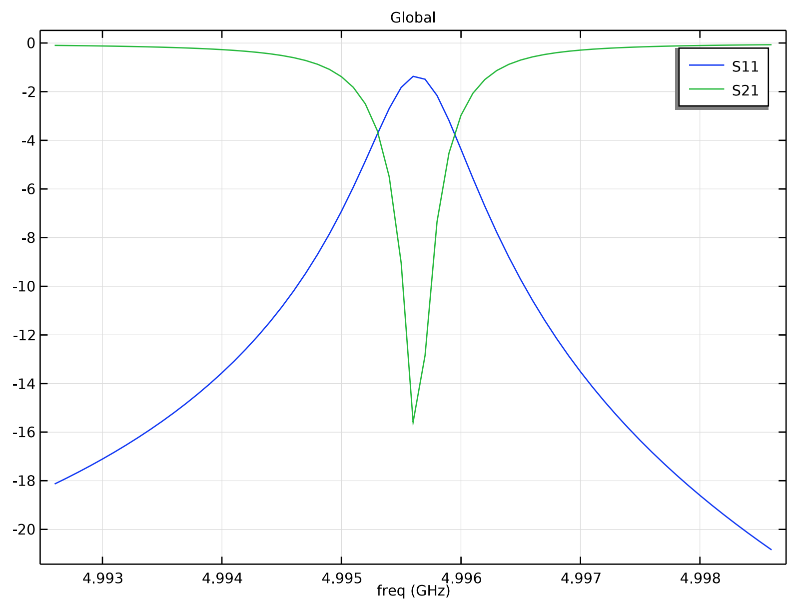
Global 1

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>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>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 **Study 2/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>emw.S11dB - S11**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`emw.S11dB`	`dB`	`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 **Display precision** text field, type `2`.

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

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

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>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 **Display precision** text field, type `3`.

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

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

9 Select the **Show frame** check box.

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