CPW Resonator for Circuit Quantum Electrodynamics

Introduction

Developments in the last decade have made circuit quantum electrodynamics (cQED) the leading architecture for quantum computation. cQED is the solid-state version of cavity QED, which studies the basic light-matter interactions at the quantum level. This new architecture for quantum hardware has two main components: superconducting qubits and transmission line resonators. Superconducting qubits are the artificial meta-atoms that serve as a two-level quantum system and the transmission-line resonators are high-quality superconducting oscillators that play the role of cavities.

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 Plank's constant, and f_{01} is the operating frequency of quantum qubits. This frequency is typically in the range of 4–8 GHz and also equal to the frequency of the microwave pulse used to coherently excite the qubits. Just like atoms, superconducting quantum qubits interact with microwave photons at quanta levels.

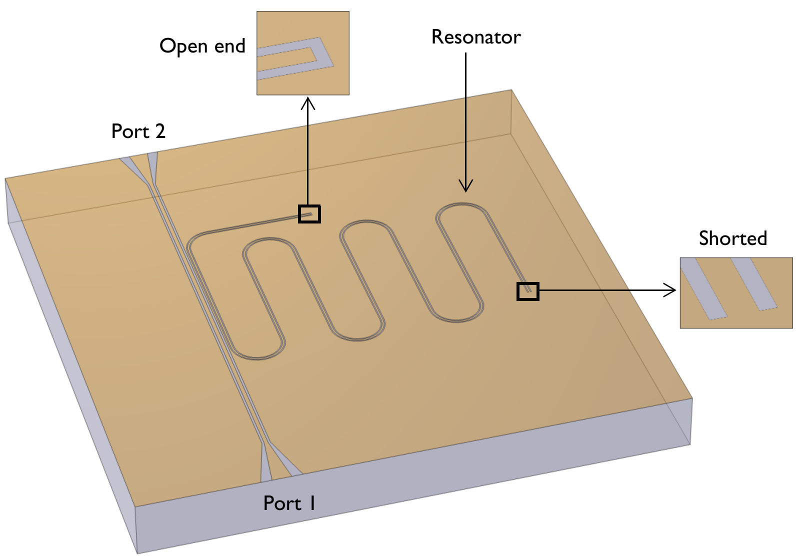


Figure 1: CPW resonator coupled to a CPW transmission line. Air domains are removed for a better view.

In this model, one of the main component of cQEDs, transmission-line resonators, are demonstrated. This resonator can be built from CPW transmission lines terminated with a combination of open and short ends. Open-short ends form nodes or antinodes at the end of the CPW which results in standing wave patterns within the transmission line and the transmission line serves as a resonator. Figure 1 illustrates the CPW resonator terminated with an open and a shorted end, also called a quarter-wave resonator, coupled to a CPW feeding line used in the model.

Model Definition

Figure 2 shows the schematic cross section of a CPW line that is used for the resonator and the feeding line. The impedance of the CPW is basically related to the ratio between center conductor width and gap width and the dielectric constants of the substrate. The conductive regions are simulated as perfect conductors for the sake of simplicity to mimic superconductors. A more realistic temperature-dependent superconductor model can be employed easily. Silicon is used as a substrate with relative permittivity 11.7.

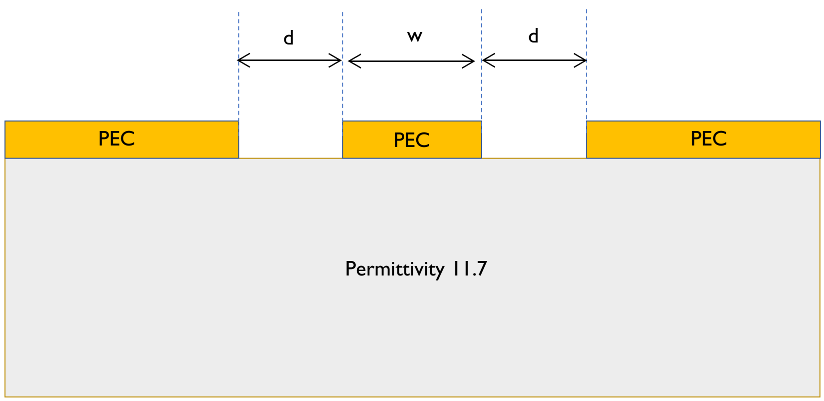


Figure 2: Schematic of the CPW cross section where w/d=7/4 and the characteristic impedance is 50 Ω.

Numeric ports are employed to excite and terminate the feeding CPW line. Therefore the boundary conditions on those surfaces are the corresponding mode fields. Radiation boundary conditions are used on the remaining boundaries. Since the loss is very small, the quality factor of the system is too high, and the resonance is very narrow. To make the simulation computationally efficient adaptive frequency sweep is employed.

In such a high-quality factor component, to capture the resonance behavior, a very fine frequency sweep is required in the vicinity of the resonance. By default, COMSOL stores field values for all the frequencies, within the 3D computational domain. If we proceed with the default settings, generated file which stores field distribution would be quite large. Since we are mainly interested in the field distribution on the CPW filter surface, to reduce the size of the file, we may only choose to store field distribution on that surface. For this purpose, geometric selections are created, and in the study settings, field values for that geometric selection are stored.

Results and Discussion

Figure 3 illustrates S parameters of the system which demonstrate resonance behavior. At the resonant frequency, all the energy is efficiently coupled to the resonator. At off resonances, electromagnetic energy does not interact with the resonator at all.

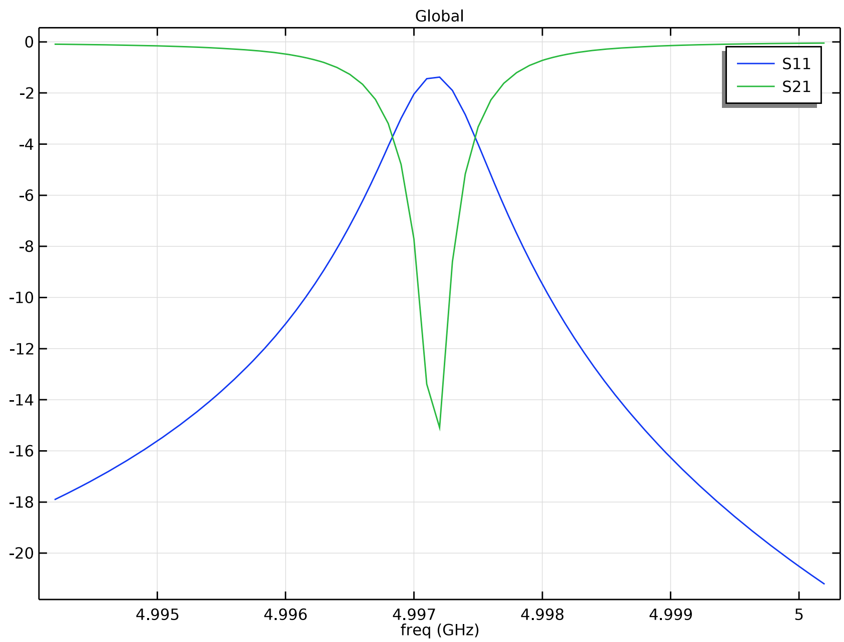


Figure 3: The S-parameters plot demonstrates a very narrow resonance behavior.

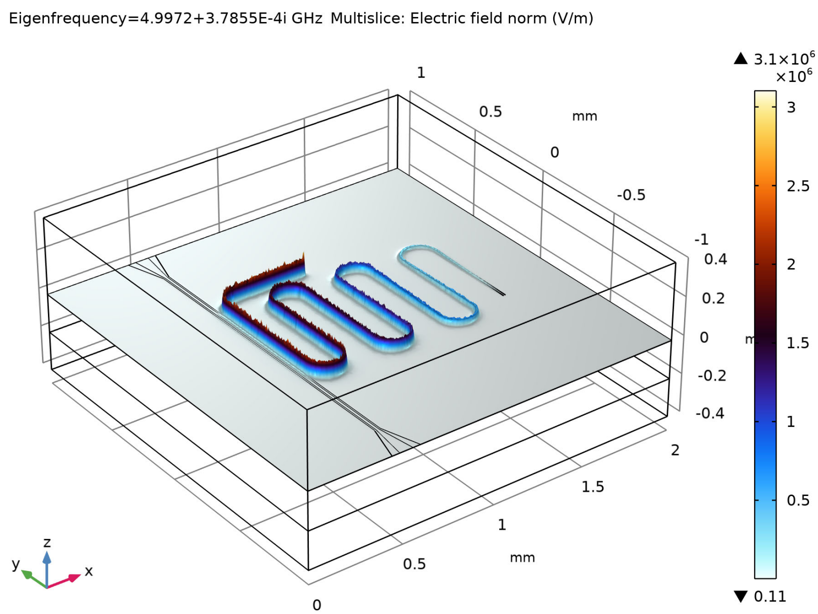


Figure 4: Illustration of standing wave pattern formed within the resonator. The height distribution corresponds to the total electric field. Antinode and node can be observed at the open and short ends.

Figure 4 demonstrates the standing wave formation in the resonator at the resonance frequency. Nodes and antinodes can be observed at the short and open ends of the resonator.

Notes About the COMSOL Implementation

Since the CPW resonator is a very high-quality factor system, it is a challenging structure to simulate. It is highly mesh-sensitive and a mesh-refinement study is necessary to make sure that the results are reliable. To show a simple modeling workflow, this model only focuses on the simulation of a CPW resonator and the mesh-refinement study is not included in the example. It requires ~20 GB of memory with the settings used here. To get accurate results, it is highly recommended to follow all the steps in this document; ignoring some of the settings may result in inaccuracies such as resonance shift. Even though this shift could be very small, since the system is very high Q, it could be challenging to find the resonance.

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**.

For the sake of simplicity, import geometry feature is used since the geometry is complicated and **COMSOL kernel** is used for **Geometry representation**. The CPW resonator contains high-aspect-ratio features, and it's also highly mesh-dependent due to its very narrow bandwidth. It is important to use the **COMSOL kernel** to reproduce the results shown in this example. Otherwise, minor variations in the mesh structure may result in discrepancies. The **CAD kernel** could be used, however, a mesh refinement process should be performed accordingly. In general, the choice of the kernel does not matter. But, this is a very high-quality factor device, and special care should be taken.

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**.

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**.

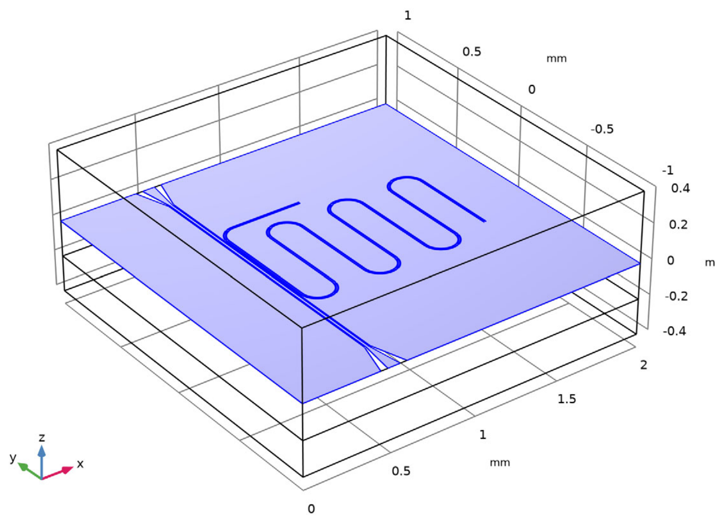
6 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Perfect Electric Conductor 2

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

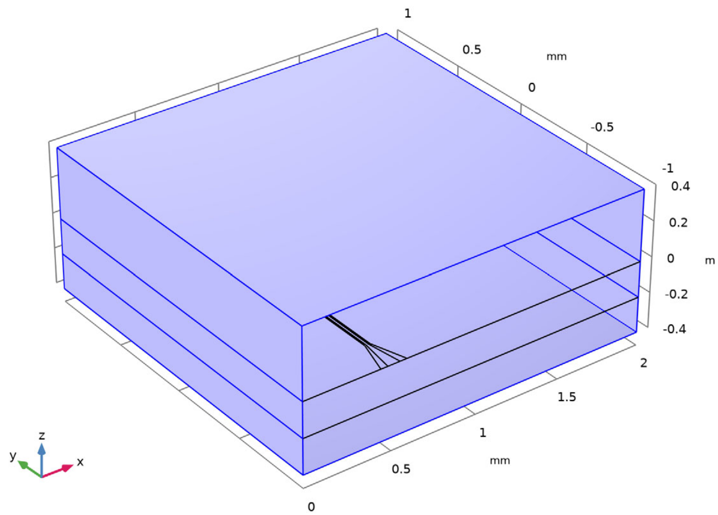
2 Select Boundaries 9, 15, and 17 only.



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.

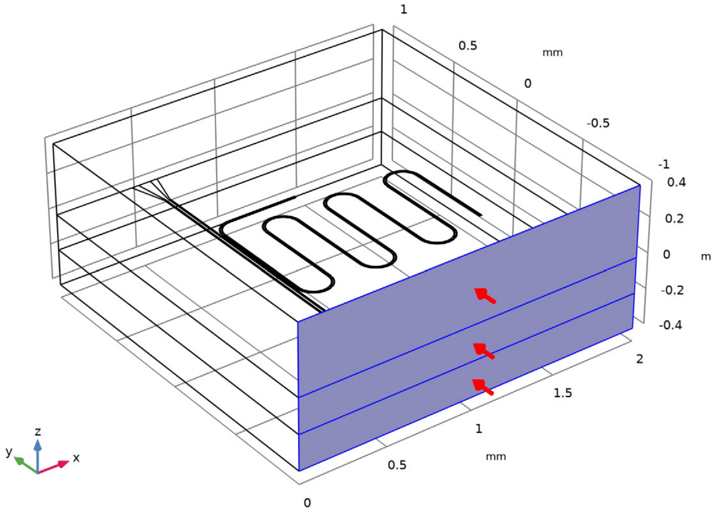


Port 1

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

There is no analytical solution to define mode field of the CPW. Use numeric ports and perform boundary mode analyses. The field distribution obtained from the boundary mode analysis will be used for 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.

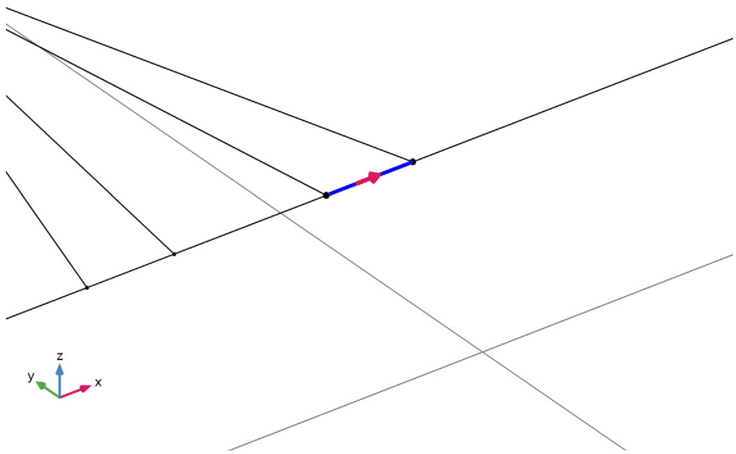
Integration Line for Voltage 1

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**.

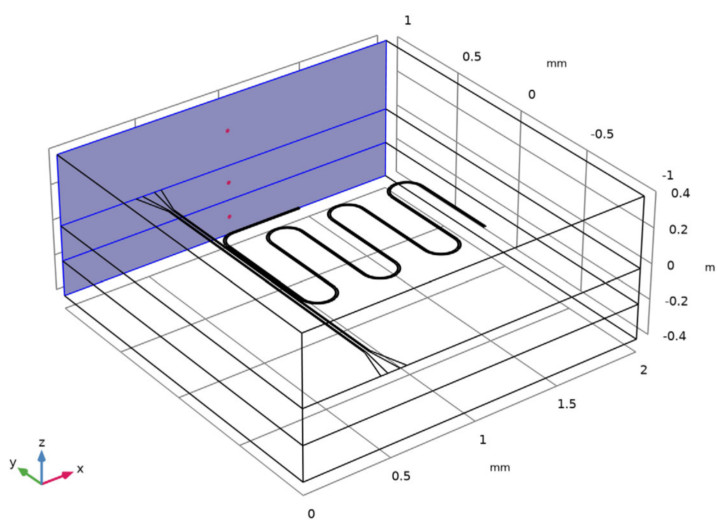
4 Select Edge 47 only.



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

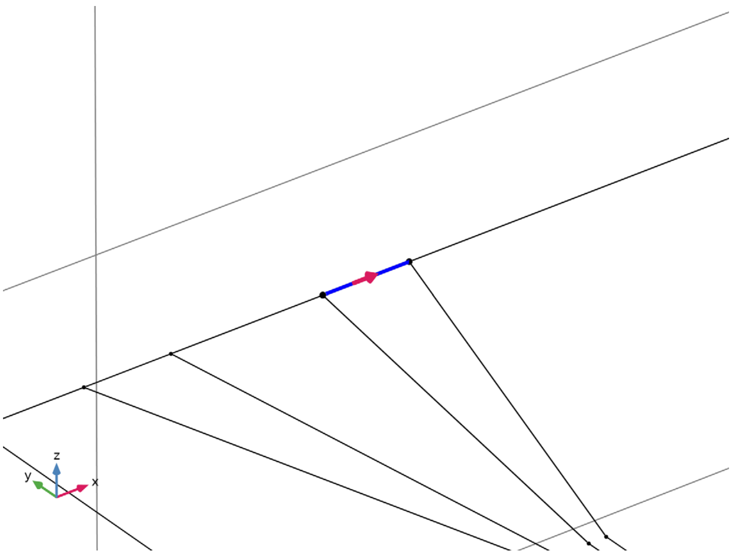
1 In the **Physics** toolbar, click **Attributes** and choose **Integration Line for Voltage**.

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**.



MATERIALS

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-in>Air**.

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

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

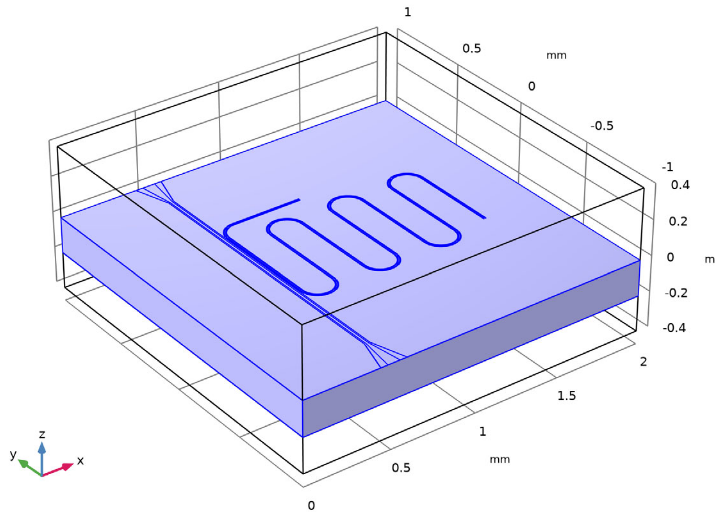
5 Click **Add to Component** in the window toolbar.

6 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**.

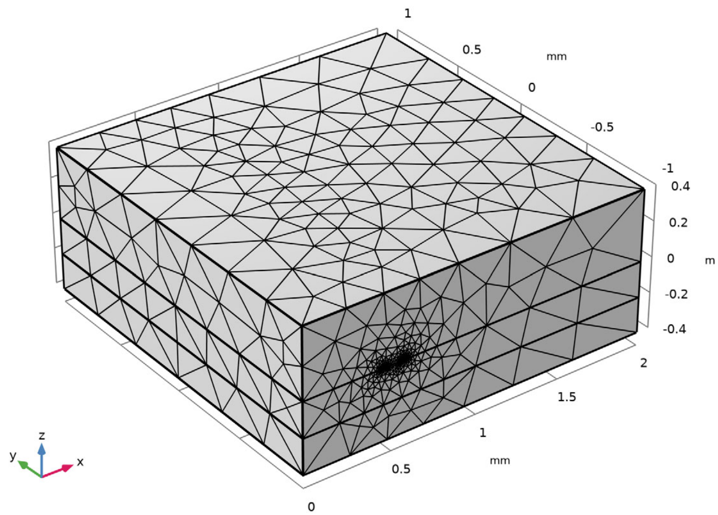
2 In the **Settings** window for **Mesh**, locate the **Electromagnetic Waves, Frequency Domain (emw)** section.

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**.



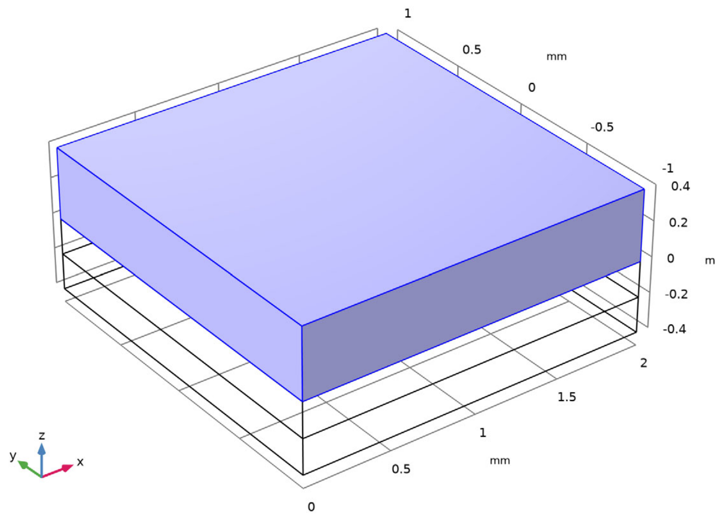
To see the mesh structure on the CPW surface, Use **Hide for Physics**.

DEFINITIONS

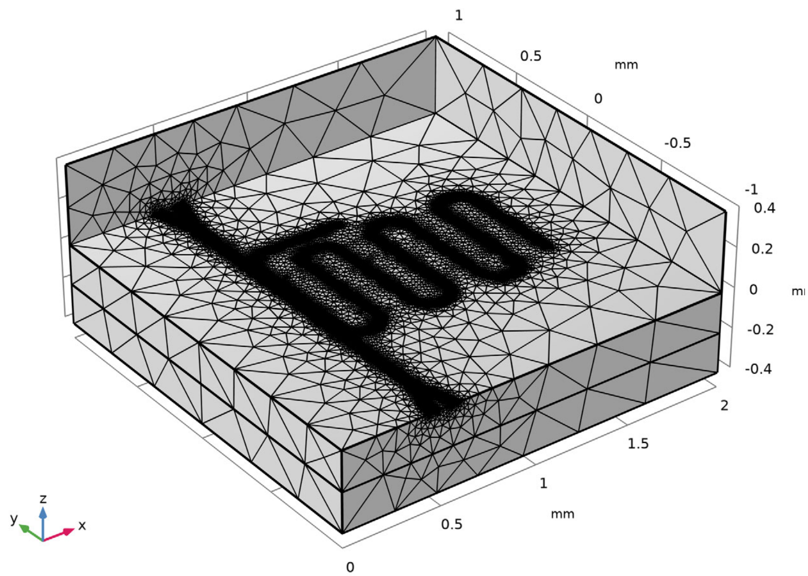
Hide for Physics 1

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



As mentioned in the port section, there is no analytical solution to define mode field of the CPW. Perform **Boundary Mode Analysis**. The field distribution obtained from the boundary mode analysis will be used for the **Eigenfrequency** and **Adaptive Frequency Sweep**.

STUDY 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** text field, type `2.5217`.

Step 2: Boundary Mode Analysis 1

1 Right-click **Study 1>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`.

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** text field, type `4.94[GHz]`.

4 Select the **Desired number of eigenfrequencies** check box.

5 In the associated text field, type `1`.

6 From the **Eigenfrequency search method around shift** list, choose **Larger real part**.

By default, COMSOL stores the field values within the computational domain, for each frequency in the study step. 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 omit the field values that we are not interested. For this purpose, a geometric selection could be generated and field values only within the geometric selection could be saved.

DEFINITIONS

Explicit 1

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

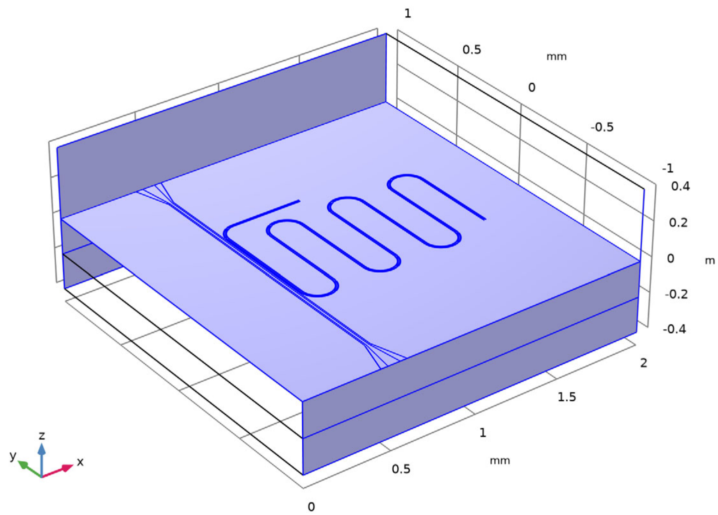
2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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 **Values of Dependent Variables** section.

3 Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

4 Under **Selections**, click **Add**.

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

6 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. The usage is limited in this model.

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

Deformation 1

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

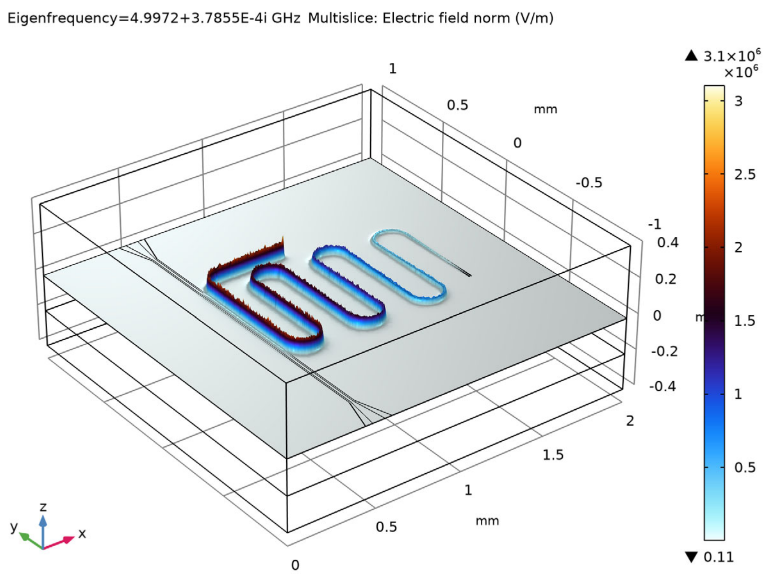
2 In the **Settings** window for **Deformation**, locate the **Expression** section.

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**.

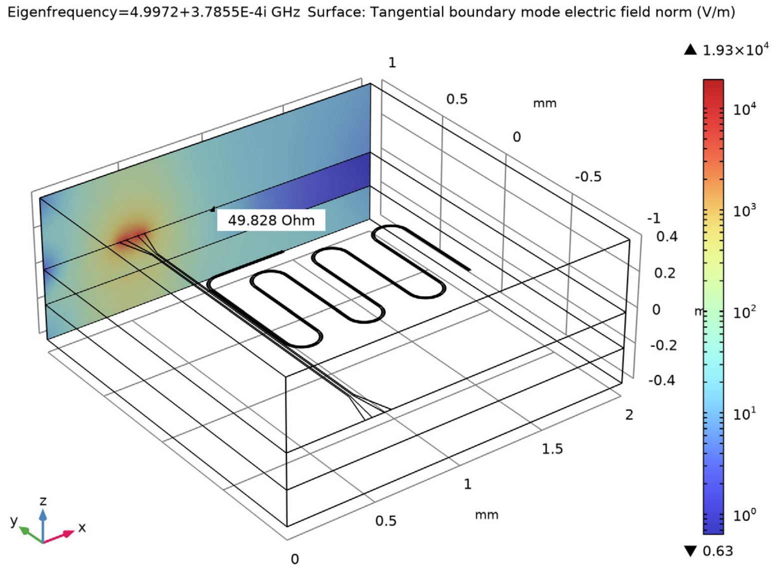


Surface 1

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

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

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

Step 1: Boundary Mode Analysis

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** check box.

Adaptive Frequency Sweep

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

   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.

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

3 In the **Frequencies** text field, type `range(4.9972[GHz]-3[MHz],0.1[MHz],4.9972[GHz]+3[MHz])`.

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

5 In the table, enter the following settings:

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

DEFINITIONS

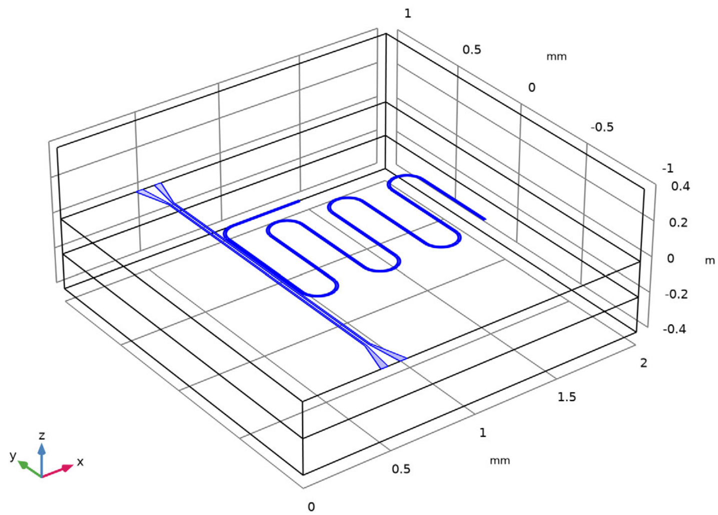
Explicit 2

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**, locate the **Values of Dependent Variables** section.

3 Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

4 Under **Selections**, click **Add**.

5 In the **Add** dialog box, select **Explicit 2** in the **Selections** list.

6 Click **OK**.

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

RESULTS

S-Parameters

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-Parameters` 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-Parameters** 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-Parameters** toolbar, click **Plot**.