

Coplanar Waveguide Bandpass Filter

Introduction

Coplanar waveguide (CPW) bandpass filters can be realized using interdigital capacitors (IDCs) and short-circuited stub inductors (SSIs). Such a filter can readily be implemented on a GaAs wafer. The presented model is compact in relation to its resonant frequency and provides a relatively high Q factor compared to capacitively coupled microstrip line model designs.



Figure 1: A coplanar waveguide bandpass filter on a $200 \,\mu m$ GaAs wafer composed of an interdigital capacitor and short-circuited stub inductors.

Model Definition

The structure shown in Figure 1 can be realized by etching a pattern in a thin gold layer on a high dielectric ($\varepsilon_r = 12.9$) GaAs substrate. In this model, the gold layer is treated as an infinitely thin layer of perfectly conducting material. Two lumped ports represent a coplanar waveguide coupling into, and out of, the device. The lumped port applies a voltage difference between the center conductor and the ground planes. This voltage difference is applied through the multielement uniform type of a lumped port that equally divides the signal between two ground planes. The line width and coupling gap on the comb in the interdigital capacitor is 100 microns, which is wide enough to account for

etching tolerances. Series and parasitic SSIs are added to generate a bandpass frequency response.

The model space consists of the GaAs wafer, with the pattern on the surface, and an air box surrounding the entire structure. The air box, in turn, is bounded by a perfect electric conductor boundary representing a die packaging placed far enough from the CPW so as not to introduce any unwanted coupling.

Results and Discussion

The structure is simulated over a range of frequencies from 7.1 GHz to 8.1 GHz. The simulation results show bandpass filter characteristics around 7.65 GHz as presented in Figure 2.

Because the filter is enclosed by the PEC package, it is effectively a grounded coplanar waveguide circuit and there is a parasitic reactance between the circuit and the bottom ground plane. This reactance loading produces wider frequency responses than those generated from the circuit itself. The model without the package as well as the bottom ground plane provides sharper frequency responses.

It is generally recommended to add air bridges around SSIs when structures of this type are used for very high frequencies in order to suppress any potential radiation over the slots.



Figure 2: The frequency response of the coplanar waveguide filter shows bandpass characteristics around 7.65GHz.

Application Library path: RF_Module/Filters/cpw_bandpass_filter

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click 🙆 Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Radio Frequency>Electromagnetic Waves, Frequency Domain (emw).
- 3 Click Add.

- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces> Frequency Domain, Modal.
- 6 Click **M** Done.

STUDY I

Step 1: Eigenfrequency

- I In the Model Builder window, under Study I click Step I: Eigenfrequency.
- 2 In the Settings window for Eigenfrequency, locate the Study Settings section.
- **3** In the **Search for eigenfrequencies around** text field, type **7.1**[GHz].

Step 2: Frequency Domain, Modal

- I In the Model Builder window, click Step 2: Frequency Domain, Modal.
- 2 In the Settings window for Frequency Domain, Modal, locate the Study Settings section.
- 3 Click Range.
- 4 In the Range dialog box, type 7.1[GHz] in the Start text field.
- 5 In the Step text field, type 2[MHz].
- 6 In the Stop text field, type 8.1[GHz].
- 7 Click Replace.

GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose µm.

Import I (imp1)

- I 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_bandpass_filter.mphbin.
- 5 Click া Import.

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



ADD MATERIAL

- I In the Home toolbar, click 🙀 Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-in>Air.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

MATERIALS

GaAs

- I In the Model Builder window, under Component I (comp1) right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type GaAs in the Label text field.

3 Select Domain 2 only.



4 Locate the **Material Contents** section. In the table, enter the following settings:

| Property | Variable | Value | Unit | Property group |
|-------------------------|---|-------|------|-------------------|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | 12.9 | I | Basic |
| Relative permeability | mur_iso ; murii = mur_iso, murij = 0 | 1 | I | Basic |
| Electrical conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | 0 | S/m | Basic |

DEFINITIONS

In the Model Builder window, expand the Component I (compl)>Definitions node.

View I

Before setting up the physics, hide the air domain to get a better view and easier access to the CPW circuit geometry.

Hide for Physics 1

- I In the Model Builder window, expand the Component I (compl)>Definitions>View I node.
- 2 Right-click View I and choose Hide for Physics.
- **3** Select Domain 1 only.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Perfect Electric Conductor 2

- I In the Model Builder window, under Component I (compl) right-click Electromagnetic Waves, Frequency Domain (emw) and choose the boundary condition Perfect Electric Conductor.
- **2** Select Boundary 9 only.



Excite and terminate each end of the CPW line using multielement uniform lumped ports.

Lumped Port I

I In the Physics toolbar, click 🔚 Boundaries and choose Lumped Port.

2 Select Boundaries 10 and 11 only.



- 3 In the Settings window for Lumped Port, locate the Lumped Port Properties section.
- **4** From the **Type of lumped port** list, choose **Multielement uniform**.

For the first port, wave excitation is **on** by default.

- 5 Locate the Boundary Selection section. Click 🖣 Create Selection.
- 6 In the Create Selection dialog box, Create a set of selections for use in the study settings.
- 7 type Lumped Port 1 in the Selection name text field.
- 8 Click OK.

Uniform Element I

- I In the Model Builder window, expand the Lumped Port I node, then click Uniform Element I.
- 2 In the Settings window for Uniform Element, locate the Boundary Selection section.
- 3 Click Clear Selection.

4 Select Boundary 11 only.



 ${\bf 5}~$ Locate the Uniform Element Properties section. Specify the ${\bf a}_h$ vector as

| 0 | x |
|---|---|
| 1 | у |
| 0 | z |

Uniform Element 2

I In the Model Builder window, click Uniform Element 2.

2 Select Boundary 10 only.



- **3** In the **Settings** window for **Uniform Element**, locate the **Uniform Element Properties** section.
- **4** Specify the $\mathbf{a}_{\mathbf{h}}$ vector as



Lumped Port 2

- I In the Physics toolbar, click 🔚 Boundaries and choose Lumped Port.
- 2 Select Boundaries 27 and 28 only.
- 3 In the Settings window for Lumped Port, locate the Lumped Port Properties section.
- 4 From the Type of lumped port list, choose Multielement uniform.
- 5 Locate the Boundary Selection section. Click 🐚 Create Selection.
- 6 In the Create Selection dialog box, type Lumped Port 2 in the Selection name text field.
- 7 Click OK.

Uniform Element I

I In the Model Builder window, expand the Lumped Port 2 node, then click Uniform Element I.

- 2 In the Settings window for Uniform Element, locate the Boundary Selection section.
- 3 Click Clear Selection.
- 4 Select Boundary 28 only.
- **5** Locate the **Uniform Element Properties** section. Specify the \mathbf{a}_h vector as

| 0 | x |
|---|---|
| 1 | у |
| 0 | z |

Uniform Element 2

- I In the Model Builder window, click Uniform Element 2.
- 2 Select Boundary 27 only.
- **3** In the **Settings** window for **Uniform Element**, locate the **Uniform Element Properties** section.
- **4** Specify the **a**_h vector as

| 0 | x |
|-----|---|
| - 1 | у |
| 0 | z |

MESH I

I In the Model Builder window, under Component I (comp1) right-click Mesh I and choose Build All.

The default frequency and material-dependent physics-controlled mesh is not fine enough to accurately evaluate the reactance of the comb structures in the IDCs. Refine the mesh settings manually.

- 2 In the Settings window for Mesh, locate the Sequence Type section.
- **3** From the list, choose **User-controlled mesh**.

Size I

- I In the Model Builder window, right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Boundary**.
- 4 Click Paste Selection.
- 5 In the Paste Selection dialog box, type 13-26 in the Selection text field.

6 Click OK.



- 7 In the Settings window for Size, locate the Element Size section.
- 8 Click the **Custom** button.
- 9 Locate the Element Size Parameters section.
- 10 Select the Maximum element size check box. In the associated text field, type 80.

II Click 📗 Build All.



12 Click the **Comextents** button in the **Graphics** toolbar.

STUDY I

Step 2: Frequency Domain, Modal

- I In the Model Builder window, under Study I click Step 2: Frequency Domain, Modal.
- **2** In the **Settings** window for **Frequency Domain**, **Modal**, 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, in the Selections list, choose Lumped Port I and Lumped Port 2.
- 6 Click OK.
- 7 In the **Home** toolbar, click **= Compute**.

RESULTS

Electric Field (emw)

Since the results of the **Frequency Domain Modal** analysis are stored only on the lumped port boundaries, this default E-field norm plot does not provide useful information. Switch the dataset to the solution of the **Eigenfrequency** analysis.

- I In the Settings window for 3D Plot Group, locate the Data section.
- 2 From the Dataset list, choose Study I/Solution Store I (sol2).

Multislice

- I 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 Find the Z-planes subsection. From the Entry method list, choose Coordinates.
- 6 In the **Coordinates** text field, type 0.
- 7 In the Electric Field (emw) toolbar, click 🗿 Plot.

Eigenfrequency=7.5106+0.070947i GHz Multislice: Electric field norm (V/m)





I In the Model Builder window, under Results click S-parameter (emw).

2 In the Settings window for ID Plot Group, locate the Legend section.

3 From the **Position** list, choose **Lower right**.

The S-parameter plot is shown in Figure 2.

Smith Plot (emw)

