

Coupled-Line Bandpass Filter

Introduction

A narrow-band bandpass filter can be realized using coupled microstrip lines. Each microstrip line piece is approximately a half wavelength long at the bandpass frequency. The bandpass performance of such a filter is much better than that of a single-section coupled-line filter.



Figure 1: Layout of a coupled-line bandpass filter. It is composed of five sections of microstrip lines.

Model Definition

The form of the coupled microstrip line filter being modeled is shown in Figure 1. The layout was designed based upon Ref. 1 to have a center frequency at 3.6 GHz, and is composed of five sections of microstrip lines. The objective of this design is to have better out-of-band rejection compared to a design with fewer cascading strips. The length of each line section is tuned such that the impedance of the filter is 50 Ω . The ports are assumed to be coupled to a 50 Ω microstrip line that is not modeled.

The microstrip lines are each modeled as perfect electric conductor (PEC) surfaces on a 0.020 inch thick dielectric substrate, with another PEC surface below that acts as the ground plane. The entire modeling domain is bounded by PEC boundaries that represent the packaging.

Two rectangular surfaces are used to model the ports. These small rectangular surfaces that bridge between two PEC surfaces represent a connection to a 50 Ω transmission line. The device is excited at one port and S_{11} and S_{21} are monitored around the bandpass frequency. The model is shown in Figure 2.

Due to the thin gap between the microstrip lines, some care must be taken during the meshing. As a general rule of thumb, at least five elements per wavelength in each material are sufficient, and the elements should have an aspect ratio of approximately unity. However, this design also has a small gap between two of the microstrip lines. This leads to very small elements in this gap region when using the default mesh settings, and the solution time is quite long. An alternative approach is to modify the mesh settings such that the elements in the gap region have a higher aspect ratio. Despite this, the solution agrees well with a finer mesh, and uses less memory.



Figure 2: The model of the coupled-line bandpass filter. The color range is scaled to emphasize the resonance on the microstrip lines.

Results and Discussion

Figure 3 shows S_{11} and S_{21} around the bandpass frequency. Excellent out of band rejection is observed. The coupling gaps can further be adjusted for frequency response optimization.



Figure 3: Frequency response of the coupled-line bandpass filter. The bandpass performance of the filter is much better than that of a single-section coupled-line filter.

Reference

1. D.M. Pozar, Microwave Engineering, John Wiley & Sons, 1998

Application Library path: RF_Module/Filters/coupled_line_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 General Studies>Frequency Domain.
- 6 Click **M** Done.

STUDY I

Step 1: Frequency Domain

- I In the Model Builder window, under Study I click Step I: Frequency Domain.
- 2 In the Settings window for Frequency Domain, locate the Study Settings section.
- 3 Click Range.
- 4 In the Range dialog box, type 3[GHz] in the Start text field.
- 5 In the Step text field, type 50[MHz].
- 6 In the **Stop** text field, type 4.2[GHz].
- 7 Click Replace.

GLOBAL DEFINITIONS

Define some parameters that are useful when setting up the mesh and the study.

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 2 In the Settings window for Parameters, locate the Parameters section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
thickness	20[mil]	5.08E-4 m	Substrate thickness
w_line	1.13[mm]	0.00113 m	Line width
l_line	25[mm]	0.025 m	Line length
gap	0.7[mm]	7E-4 m	Gap between lines

Here mil refers to the unit milliinch, that is 1 mil = 0.0254 mm.

GEOMETRY I

First, create a block for the substrate.

Substrate

- I In the **Geometry** toolbar, click 🗍 **Block**.
- 2 In the Settings window for Block, type Substrate in the Label text field.
- 3 Locate the Size and Shape section. In the Width text field, type 1_line/2.
- 4 In the **Depth** text field, type 1_line*2.2.
- 5 In the **Height** text field, type thickness.
- 6 Locate the Position section. From the Base list, choose Center.

Then, add a block for the metal package enclosing the substrate.

Enclosure

- I In the **Geometry** toolbar, click 🗍 **Block**.
- 2 In the Settings window for Block, type Enclosure in the Label text field.
- 3 Locate the Size and Shape section. In the Width text field, type 1_line/2.
- 4 In the **Depth** text field, type 1_line*2.4.
- 5 In the **Height** text field, type 1_line/5.
- 6 Locate the Position section. From the Base list, choose Center.
- 7 In the z text field, type l_line/10-thickness/2.
- 8 Click **H** Build All Objects.

9 Click the 🖂 Wireframe Rendering button in the Graphics toolbar.



Add a work plane on the substrate to create the microstrip lines.

- Work Plane I (wp1)
- I In the Geometry toolbar, click Work Plane.
- 2 In the Settings window for Work Plane, locate the Plane Definition section.
- 3 In the z-coordinate text field, type thickness/2.
- 4 Click 📥 Show Work Plane.

Work Plane I (wp1)>Plane Geometry

Create a rectangle for the microstrip line connected to Lumped port 1.

Work Plane I (wp1)>Rectangle I (r1)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 Click the **Zoom Extents** button in the **Graphics** toolbar.
- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- 4 In the Width text field, type w_line.
- 5 In the **Height** text field, type 1_line*0.6.
- 6 Locate the **Position** section. From the **Base** list, choose **Center**.
- 7 In the **xw** text field, type -w_line*2-gap*1.07.
- 8 In the **yw** text field, type -1_line*0.8.

Work Plane I (wpI)>Plane Geometry

Then, add a rectangle next to the first microstrip line. There is a very thin gap between two microstrip lines.

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

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type w_line.
- 4 In the Height text field, type 1_line.
- 5 Locate the Position section. From the Base list, choose Center.
- 6 In the xw text field, type -w_line-gap.
- 7 In the **yw** text field, type -1_line/2.

Work Plane I (wpI)>Plane Geometry

Generate another pair of microstrip lines with a thin gap at the other side of the substrate by rotating the above two rectangles 180 degrees. Keep the input objects.

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

- I In the Work Plane toolbar, click 💭 Transforms and choose Rotate.
- 2 Click in the Graphics window and then press Ctrl+A to select both objects.



- 3 In the Settings window for Rotate, locate the Input section.
- 4 Select the Keep input objects check box.

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

Work Plane I (wp1)>Plane Geometry

Finish the layout on the substrate by adding the last piece of the microstrip lines on the center of the substrate.

Work Plane 1 (wp1)>Rectangle 3 (r3)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type w_line.
- 4 In the **Height** text field, type 1_line.
- 5 Locate the Position section. From the Base list, choose Center.



6 Click 틤 Build Selected.

Add a work plane to define boundaries for the lumped ports.

Work Plane 2 (wp2)

- I In the Model Builder window, right-click Geometry I and choose Work Plane.
- 2 In the Settings window for Work Plane, locate the Plane Definition section.
- 3 From the Plane type list, choose Face parallel.

4 On the object **blk1**, select Boundary 3 only.



Work Plane 2 (wp2)>Plane Geometry In the Model Builder window, click Plane Geometry.

Create a rectangle for the first lumped port.

Work Plane 2 (wp2)>Rectangle 1 (r1)

- I In the Work Plane toolbar, click 📃 Rectangle.
- **2** Click the 4 **Zoom Extents** button in the **Graphics** toolbar.
- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- 4 In the Width text field, type thickness.
- **5** In the **Height** text field, type w_line.
- 6 Locate the Position section. From the Base list, choose Center.
- 7 In the **yw** text field, type w_line*2+gap*1.07.

Then, rotate the above rectangle to create the second lumped port.

Rotate I (rotI)

- I In the Model Builder window, right-click Geometry I and choose Transforms>Rotate.
- 2 Select the object wp2 only.
- 3 In the Settings window for Rotate, locate the Input section.
- 4 Select the Keep input objects check box.

- **5** Locate the **Rotation** section. In the **Angle** text field, type 180.
- 6 Click 🟢 Build All Objects.



DEFINITIONS

Create a set of selections for use before setting up the physics.

Substrate

- I In the **Definitions** toolbar, click **heat Explicit**.
- 2 In the Settings window for Explicit, type Substrate in the Label text field.

3 Select Domain 2 only.



Add a selection for the microstrip line traces.

Microstrip line

- I In the **Definitions** toolbar, click **here Explicit**.
- 2 In the Settings window for Explicit, type Microstrip line in the Label text field.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.

4 Select Boundaries 13 and 15–18 only.



View I

Hide three boundaries to get a better view of the interior parts when setting up the physics.

Hide for Physics 1

- I In the Model Builder window, right-click View I and choose Hide for Physics.
- 2 In the Settings window for Hide for Physics, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.

4 Select Boundaries 1, 2, and 4 only.



To see the objects in selections, make sure that **Show Objects in Selection** is turned on. The button is located at the upper-left corner in the **Hide Geometric Entities** settings window.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. The default boundary condition is **Perfect Electric Conductor**, which applies to all exterior boundaries. Also, assign a perfect electric conductor condition on the interior boundary of the microstrip lines. Use the selection for the microstrip line boundaries that you defined earlier.

Perfect Electric Conductor 2

- In the Model Builder window, under Component I (comp1) right-click
 Electromagnetic Waves, Frequency Domain (emw) and choose the boundary condition
 Perfect Electric Conductor.
- **2** In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
- **3** From the Selection list, choose Microstrip line.

Lumped Port I

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

2 Select Boundary 12 only.

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

Lumped Port 2

- I In the Physics toolbar, click 📄 Boundaries and choose Lumped Port.
- **2** Select Boundary 19 only.

MATERIALS

Next, assign material properties on the model. Begin by specifying air for all domains.

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

Override the substrate with a dielectric material of $\varepsilon_r = 3.38$.

Substrate

- I In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type Substrate in the Label text field.
- **3** Locate the Geometric Entity Selection section. From the Selection list, choose Substrate.
- 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	3.38	I	Basic

Property	Variable	Value	Unit	Property group
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

MESH I

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



STUDY I

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

RESULTS

Electric Field (emw)

The default plot shows the distribution of the norm of the electric field. Show that only on the *xy*-plane.

I In the Settings window for 3D Plot Group, locate the Data section.

2 From the Parameter value (freq (GHz)) list, choose 3.55.

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



The microstrip line in the middle of the substrate shows the resonance at the selected frequency.

S-parameter (emw)



Smith Plot (emw)



Analyze the same model with a Frequency Domain Modal method. When a device presents resonances, the **Frequency Domain Modal** method combined with an **Eigenfrequency** analysis provides a faster solution time.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Lumped Port I

- In the Model Builder window, under Component I (compl)>Electromagnetic Waves, Frequency Domain (emw) click Lumped Port I.
- 2 In the Settings window for Lumped Port, locate the Boundary Selection section.
- 3 Click here are a create Selection.
- 4 In the Create Selection dialog box, Create a set of selections for use in the study settings.
- 5 type Lumped port 1 in the Selection name text field.
- 6 Click OK.

Lumped Port 2

- I In the Model Builder window, click Lumped Port 2.
- 2 In the Settings window for Lumped Port, locate the Boundary Selection section.
- 3 Click http://www.create Selection.
- 4 In the Create Selection dialog box, type Lumped port 2 in the Selection name text field.
- 5 Click OK.

ADD STUDY

- I In the Home toolbar, click \sim_{1}^{2} 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

Preset Studies for Selected Physics Interfaces>Frequency Domain, Modal.

- 4 Click Add Study in the window toolbar.
- 5 In the Home toolbar, click 2 Add Study to close the Add Study window.

STUDY 2

Step 1: Eigenfrequency

- I In the Settings window for Eigenfrequency, locate the Study Settings section.
- 2 In the Search for eigenfrequencies around text field, type 3[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 In the Frequencies text field, type range(3[GHz],50[MHz]/50,4.2[GHz]).

With a 50 times finer frequency step, the solutions will increase the model file size tremendously when it is saved. When only S-parameters are of interest, a common theme in most passive RF and microwave device designs, it is not necessary to store all of the field solutions. By selecting the **Store fields in output** check box in the **Values of Dependent Variables** section, we can control the part of the model on which the computed solution is saved. We only add the selection containing these boundaries where the S-parameters are calculated. The lumped port size is typically very small compared to the entire modeling domain, and the saved file size with the finer frequency step is more or less that of the regular discrete frequency sweep model when only the solutions on the port boundaries are stored.

- **4** Click to expand the **Values of Dependent Variables** section. Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.
- **5** Under **Selections**, click + **Add**.
- 6 In the Add dialog box, in the Selections list, choose Lumped port I and Lumped port 2.
- 7 Click OK.
- 8 In the Home toolbar, click **=** Compute.

RESULTS

Electric Field (emw) 1

Since the results are stored only on the lumped port boundaries, this default E-field norm plot does not provide useful information.

I Right-click Results>Electric Field (emw) I and choose Delete.

Generate all S-parameters from each analysis on the same plot and compare them to each other.

S-parameter (emw) 1

- I In the Model Builder window, under Results click S-parameter (emw) I.
- 2 In the Settings window for ID Plot Group, click to expand the Title section.
- **3** From the **Title type** list, choose **Manual**.
- **4** In the **Title** text area, type S-parameter Comparison between Frequency Domain Modal and Discrete Sweep.

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

Global 2

- I In the Model Builder window, expand the S-parameter (emw) I node.
- 2 Right-click Results>S-parameter (emw) I>Global I and choose Duplicate.
- 3 In the Settings window for Global, locate the Data section.
- 4 From the Dataset list, choose Study I/Solution I (soll).
- 5 Click to expand the Coloring and Style section. Find the Line style subsection. From the Line list, choose Dotted.
- 6 Find the Line markers subsection. From the Marker list, choose Cycle.

Global I

- I In the Model Builder window, click Global I.
- 2 In the Settings window for Global, locate the y-Axis Data section.
- **3** In the table, enter the following settings:

Expression	Unit	Description
emw.S11dB	1	S11, FD Modal
emw.S21dB	1	S21, FD Modal

4 In the S-parameter (emw) I toolbar, click **I** Plot.

Compare the resulting plots with that shown in Figure 3.

Smith Plot (emw) I



Compare the solution time between two studies.

The following instruction shows how to use the **Graph Marker** subfeature to analyze 1D plots. When plotting S11 of a bandpass filter, poles are of interest and a graph marker captures the local minima. For analyzing the insertion loss such as S21, the -3dB bandwidth of the passband can be computed through an additional graph marker.

S-parameter with Graph Markers

- I In the Home toolbar, click 🚛 Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID 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 2 (sol2).

Global I

- I Right-click S-parameter with Graph Markers and choose Global.
- 2 In the Settings window for Global, click Add Expression in the upper-right corner of the y-Axis Data section. From the menu, choose Component I (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 I

- I Right-click Global I and choose Graph Marker.
- 2 In the Settings window for Graph Marker, locate the Display section.
- 3 From the **Display** list, choose **Min**.
- 4 From the **Scope** list, choose **Local**.
- 5 Locate the Text Format section. In the Display precision text field, type 3.
- 6 Select the Show x-coordinate check box.
- 7 Select the Include unit check box.
- 8 Click to expand the Coloring and Style section. From the Anchor point list, choose Lower left.

Global 2

- I In the Model Builder window, right-click S-parameter with Graph Markers and choose Global.
- 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 I (compl)>
 Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>emw.S21dB S21.

Graph Marker I

- I 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 Locate the Text Format section. In the Display precision text field, type 3.
- **5** Select the **Include unit** check box.

6 Locate the Coloring and Style section. Select the Show frame check box.



Touchstone I

- I In the **Results** toolbar, click **Data** and choose **Touchstone**.
- 2 In the Settings window for Touchstone, locate the Output section.
- 3 In the Filename text field, type coupled_line_filter_touchstone.s2p.
- 4 From the Parameter format list, choose Magnitude in dB and angle.
- 5 Click 📑 Export.

Warning I

Review the warning. Since a port sweep is not performed, the solution is incomplete to generate a full S-parameter matrix. The Touchstone export assumes the simulation model is a reciprocal network and creates a symmetric matrix.