RF/Microwave Circuits I Laboratory #4

Coupled-Line Band-Pass Filter Design

Overview

In this laboratory you will design and simulate a microstrip coupled-line band-pass filter with a center frequency at 3.5 GHz (fc) and a required "stop frequency" (fstop) at 4.0 GHz. The design and topology of the filter are relatively simple, and the primary requirements are that the filter must pass a signal at fc with minimal attenuation, and block a signal at fstop with maximum attenuation.

The laboratory involves multiple phases of filter implementation and analysis. In the first part of the assignment you will determine the filter design parameters and then implement an ideal transmission line version of the topology. Next, you will create a microstrip version of the filter. The microstrip schematic will be 'tuned' (manually or using optimization) to achieve the necessary performance.

The final part of the laboratory assignment requires you to capture the schematic into the layout program, and then perform a Momentum simulation. You will use the 'visualization' tools of ADS to plot the current density on the microstrip.

As with previous laboratories, you will make use of information that is contained in several of the ADS tutorial procedures. In comparison to previous laboratory assignments, less detail is provided for the step-by-step processes as you should be gaining some level of familiarity with the software.

** NOTE ** The examples and plots in the following write-up are based on a substrate with the following properties: H = 0.356 mm, Er = 4.3, TanD = 0.022 (i.e., 14 mil FR4).

You should base your design on the following properties: H = 0.508 mm, Er = 3.6, TanD = 0.005, T=0.043 mm.

The assignment is organized as follows:

♦ Determine a topology that exhibits a center frequency at fc and provides at least 30 dB of rejection at fstop, using a 0.5 dB ripple Chebyshev design with a reference impedance of 50 Ω. The only specification on the filter bandwidth is that it be greater than 2%. In the physical microstrip coupled-line implementation, all gap dimensions must be greater than 50 microns in order for the filter to be manufactured.

♦ Create a schematic for the filter design based on **ideal (coupled) transmission line elements**.

♦ Create a second filter schematic, in which the ideal elements are replaced by microstrip transmission line sections. You will use *Linecalc* to determine the necessary lengths, widths and spacings of each section.

♦ Compare the performance of the two schematics. Subsequently, modify the microstrip filter design to center the pass-band at fc and ensure the attenuation goal at fstop is met.

♦ Capture the microstrip filter schematic to the layout program and perform a simulation using *Momentum*. Upon completion, generate a plot of the electric current density.

I. Determine the Filter Geometry and Create the "Ideal" Filter Schematic

You will start by designing the "ideal" filter and testing your design with "ideal transmission line" elements to ensure that you have followed the process correctly. This is a sound practice to follow in any design, since you are minimizing the amount of "uncertainty" by conducting a sanity check with the ideal elements.

1) Using a **0.5 dB ripple Chebyshev response**, determine a filter design that will meet the following specifications:
   a) Center frequency at fc
   b) Bandwidth greater than or equal to 2%
   c) Minimum of 30 dB rejection at fstop
   d) **NOTES**: The amount of out-of-band attenuation at a given frequency-offset from the center frequency increases as the filter bandwidth decreases (for a given number of sections). Generally speaking, however, the minimum gap spacing on the coupled-line filter sections also will decrease as the filter bandwidth decreases. While at this point you haven't looked at the microstrip coupled-line dimensions, you will eventually need to ensure that no gap spacing is less than 50 microns. Another way to increase the out-of-band attenuation at a given frequency-offset is to increase the number of sections in the filter. I strongly suggest, though, that you not use an even order filter (N=2, 4, 6,…) since the terminating impedance in a Chebyshev design is no longer 50 Ohms.
   SUGGESTION: start with a minimum reasonable number of sections and experiment with different values for the bandwidth (start with a large bandwidth and work your way down), checking to see the amount of attenuation you can achieve at fstop. Increase the number of sections only if you cannot achieve the out-of-band attenuation specification with a reasonable amount of margin (e.g., 5 dB) while staying within the bounds of allowable bandwidths ($> 2\%$).
   e) **Summary**: for this step in the process, you are trying to determine (a) the order of the filter, and (b) the filter bandwidth.
2) Create a schematic representation of the filter using ideal transmission line components (see Figure 1). In what follows, this schematic is referred to as lab4_ideal_1.
   a) Use "ideal coupled transmission line" (CLIN) elements in the schematic. These elements are found in the T-lines Ideal palette.

b) As shown in the figure, you need to enter the even and odd mode impedance for each element, the electrical length, and the center frequency (for the electrical length).
c) Also as shown in the schematic, there will be two "pins" on each element that are not connected to any other part.
d) Attach terminations at each end of the schematic and perform an S-parameter simulation from 2-7 GHz.
e) Once you have verified that your design meets the center frequency and out-of-band attenuation specifications (e.g., see Figure 2), proceed to the microstrip implementation in the next section.
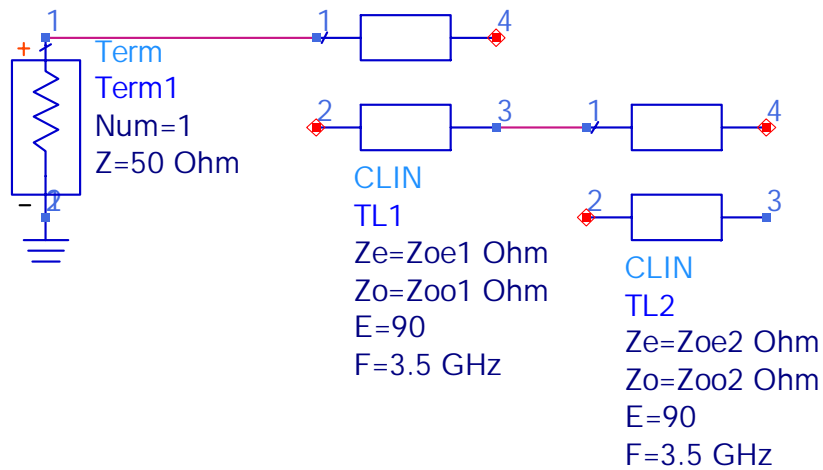
Figure 1. Partial schematic for band-pass filter using ideal coupled transmission line elements.
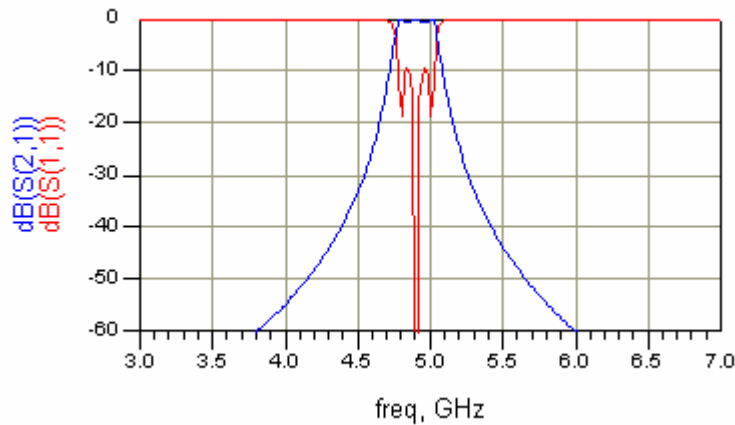
Figure 2. Example results for an ideal band-pass filter, with a center frequency of 4.9 GHz.

II. Create the Microstrip Filter Schematic

1) Use the *Linecalc* program in ADS to determine the correct lengths and widths for microstrip coupled-line sections corresponding to each part of the filter schematic. Also determine the width of a 50 Ω microstrip line. Set the frequency to fc and use the substrate parameters given above (these are also shown in Figure 3). Note that we are specifying the conductivity to be that of copper (5.8e7 S/m).

If any gap dimension for the coupled-line microstrip sections is less than 50 microns, you must return to Part I above and determine a different filter design. Refer to the "notes" under step 1 of Part I.

When you are using the "synthesis" routines in Linecalc to determine the dimensions of the coupled microstrip line sections, you sometimes have to coerce the program to provide "reasonable" dimensions. For a more detailed discussion on this topic, you should refer to ADS Procedure 5 (Using the Linecalc Program in ADS) that is included in the files you should have previously downloaded from the course web-site. Note that this procedure was written primarily for use in the Wireless Circuits and Systems Laboratory, so you should not pay attention to the particular substrate and transmission line parameters that are specified therein.

2) Create a schematic for the filter that is based on microstrip elements, herein referred to as lab4_microstrip_1 (see Figure 3).
    a) In addition to the filter elements, include 5 mm-long sections of 50 Ω microstrip line at each end of the filter.
    b) Attach terminations at each end of the schematic and perform an S-parameter simulation from 2-7 GHz.
    c) Once you are satisfied that your filter meets the design specifications (e.g., see Figure 4), obtain a hardcopy of the schematic and a hardcopy of the simulated S-parameter results (magnitude of S11 and S21 in dB). Set the minimum value on the y-axis to –60 dB.
    d) As seen in Figure 4, the pass-band insertion loss of the microstrip filter will be higher than the ideal design. The return loss in the pass-band should be approximately 9-10 dB or better.
3) At this point you have an ideal filter design and a microstrip version that should meet the design specifications. Obtain a hardcopy of the simulated S-parameter results for the ideal filter. Plot the magnitude of S11 and S21 in dB, setting the minimum value on the y-axis to –60 dB. Also obtain a hardcopy of your ideal filter schematic.

Term
Term1
Num=1
Z=50 Ohm

MLIN
TL1
Subst="MSub1"
W=W50 mm
L=5 mm

MCFIL
CLin1
Subst="MSub1"
W=W1 mm
S=S1 mm
L=L1 mm

MSub

MSUB
MSub1
H=0.508 mm
Er=3.6
Mur=1
Cond=5.8e7
Hu=1.0e+033 mm
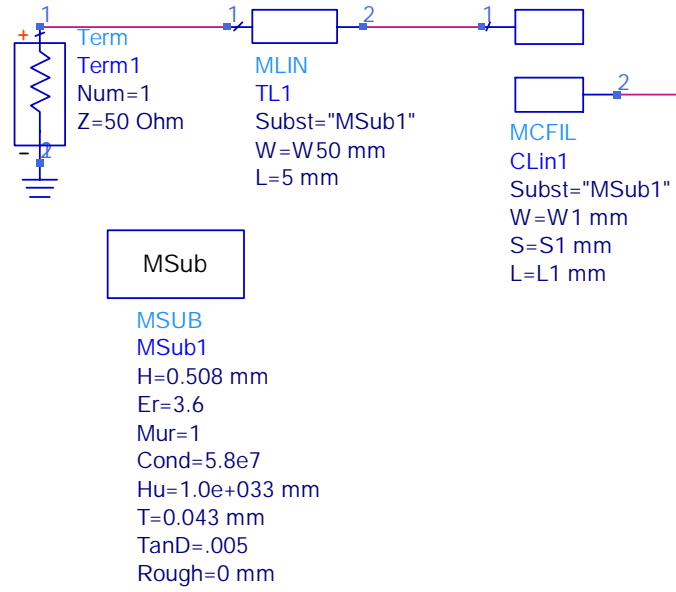T=0.043 mm
TanD=.005
Rough=0 mm

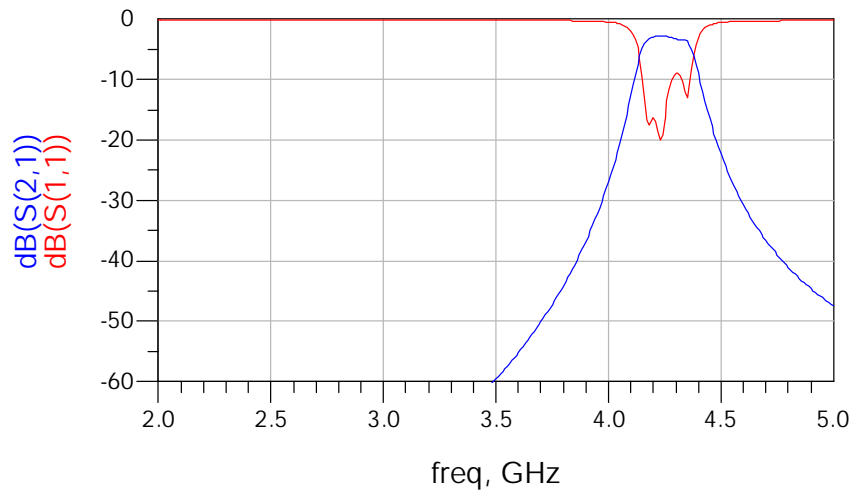*Figure 3. Partial schematic for a band-pass filter using microstrip coupled-line elements.*

Figure 4. Example results for a microstrip coupled-line band-pass filter with a center frequency of 4.25 GHz.

EEL4421_6935SoftwareLaboratory4Fall2007.doc

III. Perform a Momentum Simulation of the Microstrip Filter

1) Use the 'schematic capture' feature to automatically generate a layout of the microstrip filter (e.g. lab4_microstrip_1).
    a) In the schematic window, select Layout -> Generate/Update Layout. (Note: if you perform this step more than once, you should always open the Layout window and Edit -> Delete All to clear the layout before each schematic capture is performed.)
    b) In the Layout window, select and delete the large boxes that represent the "terminations." (If you do not see any large boxes at the termination positions then don't worry about it.) If there are "pins" that remain on the extreme left and/or right sides of the layout (i.e., pins that are not connected to the filter geometry) also select and delete these.
    c) Your layout should resemble that shown in Figure 5.
    d) In the layout window, select Momentum>Substrate>Update from schematic. This step should automatically transfer the MSUB information from the schematic into the Momentum files (if you select Substrate>Create/Modify, the definition should resemble that given in Figure 6 and Figure 7.
    e) Select Momentum>Substrate>Save As… and use the default name that is provided.
    f) Add ports to each end of the layout:
        i) Select Options -> Midpoint Snap
        ii) Zoom in on the left-hand side of the filter.
        iii) Add a "port" to the mid-point of the 50 Ohm line on the left-hand side of the filter. Refer to Figure 8.
        iv) Repeat these steps to add a port on the right-hand side of the filter.
    g) Select Momentum -> Port Editor. Click on each port and make certain they are specified as shown in Figure 9 and click Apply. (Make sure to do this for both ports.)
    h) Select Momentum>Mesh>Setup controls. Enter the parameters shown in Figure 10.
    i) Select Momentum>Mesh>Precompute mesh. The generated mesh should resemble that shown in Figure 11.
    j) Obtain a hardcopy of your layout with the mesh shown.

Figure 5. Partial layout of the microstrip filter created using the schematic capture tool.

2) Perform the Momentum simulation and generate a plot of the calculated S-parameters and of the electric current density.

a) Select Momentum>Simulation>Simulate circuit. Set the parameters as shown in Figure 12 and perform the simulation.

b) When the simulation is complete, a display will automatically open with plots of the S-parameters. If you already had a display open from the previous schematic window simulation, the results will simply be updated with the Momentum results.

When the Momentum simulation was performed, two separate output data sets were created. One was named lab4_microstrip_1.dds (assuming this was the saved name of your layout file) and the second was named lab4_microstrip_1_a.dds. The first dataset contains S-parameter results for the particular frequencies that were used by Momentum during the simulation – this file will actually overwrite the output file from the schematic simulation if the names of the schematic and layout files are the same. However, the more important dataset in this case is the second one (with the _a extension) as this contains S-parameters at more frequency points, created during the "adaptive" sweep.

c) In the display window, create (or modify the) plots to ensure that the data being plotted is from the lab4_microstrip_1_a dataset. Generate a plot from this dataset that includes the magnitude of S11 and S21, with the minimum value on the y-axis set to –60 dB. (If you double-click on the plot, you will be able to select the dataset from the pull-down menu in the middle of the pop-up window.)

d) Sample results are shown in Figure 13.

e) Obtain a hardcopy of the plots.

3) Finally, use the visualization capability of Momentum to generate a plot of the electric current density on the microstrip filter.

a) From the layout window, select Momentum>Post-Process>Visualization.

b) From the visualization window, select Current>Set Port Solution Weights from the menu near the top of the window. Verify that the default settings are as shown in Figure 14. (The port solution weight should be 1 at Port 1 at fc GHz, specifying that the excitation should be applied only at this port and at this frequency.)

c) Now select Current>Plot Currents and use the "shaded" option. You should obtain a plot similar to that shown in Figure 15.

d) Experiment with plotting the current densities as the excitation frequency changes (by modifying the port solution weights). For each change, specify a different "view" from the Plot Currents window, else you will have multiple plots being generated on top of each other. You can also click on the Display Properties button on the left-hand side of the main visualization window, and create an "animation" of the current. Select 2 GHz as one of your plot frequencies.
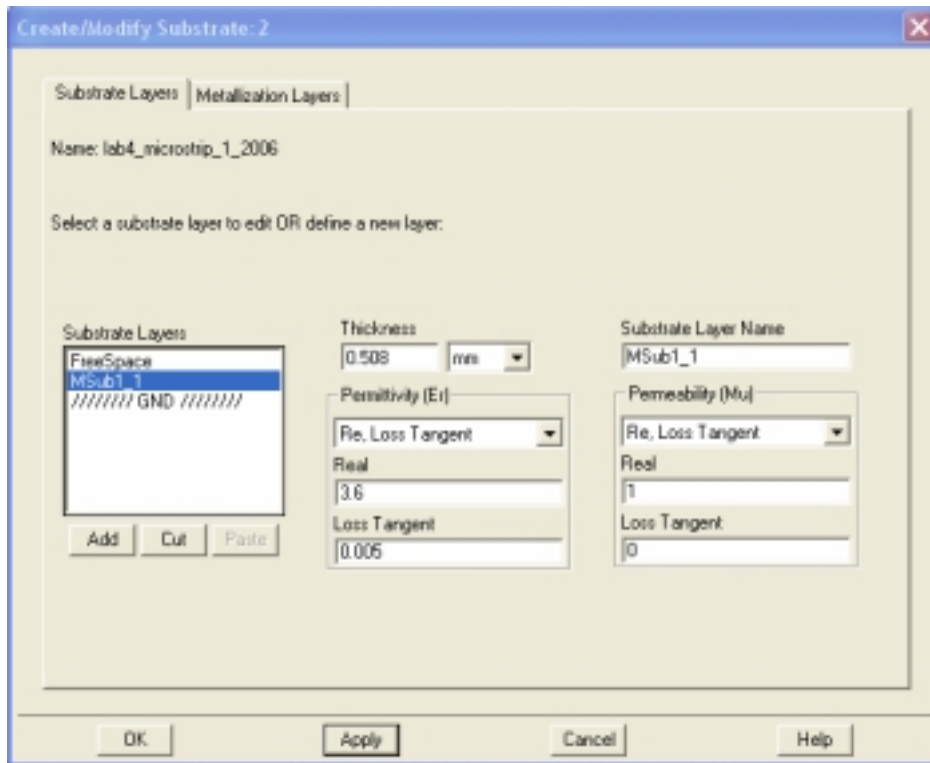
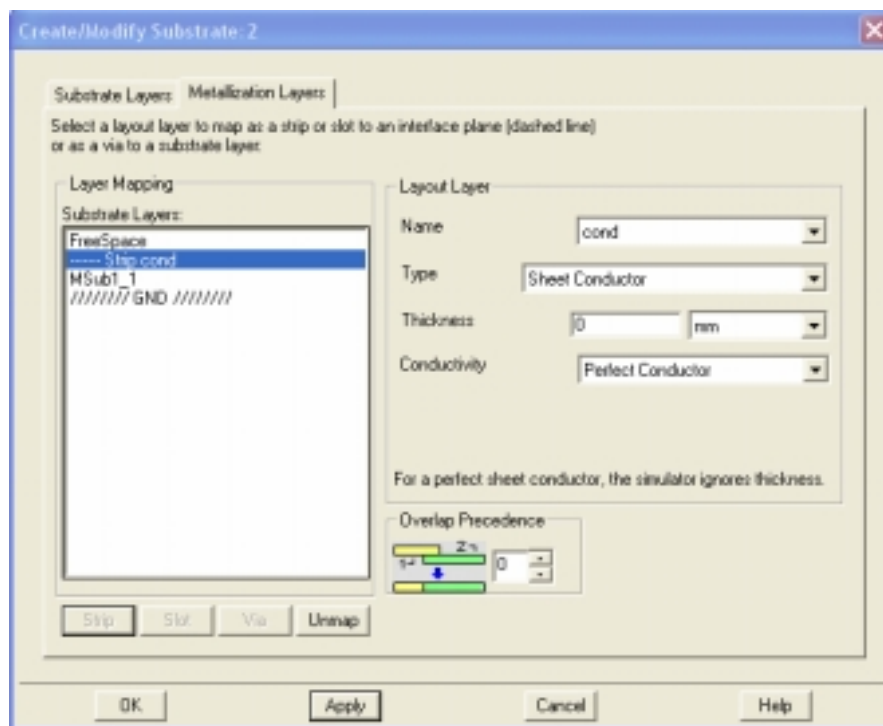*Figure 6. Substrate definition used for the Momentum simulation.*

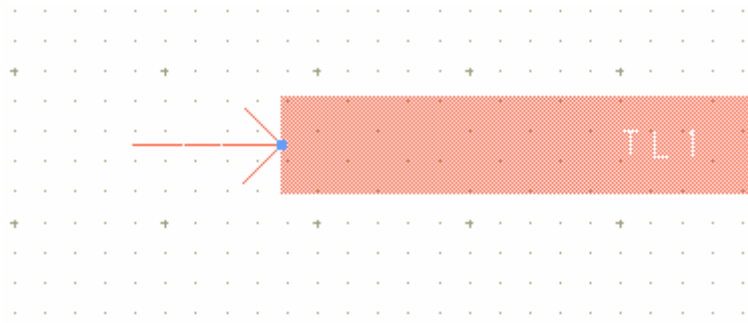*Figure 7. Substrate definition (mapping of metallization layers) used in the Momentum simulation.*
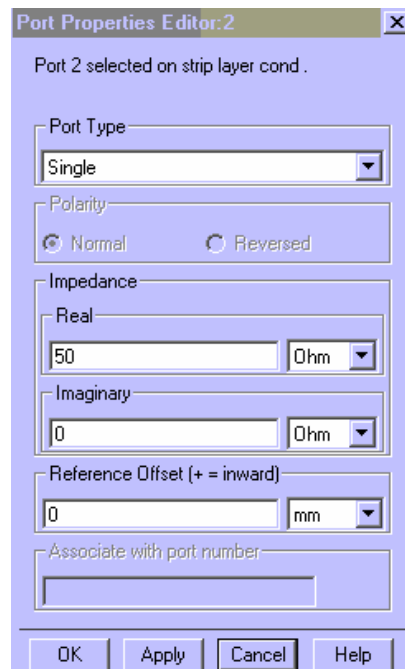
Figure 8. Port added to the left-hand side of the filter.
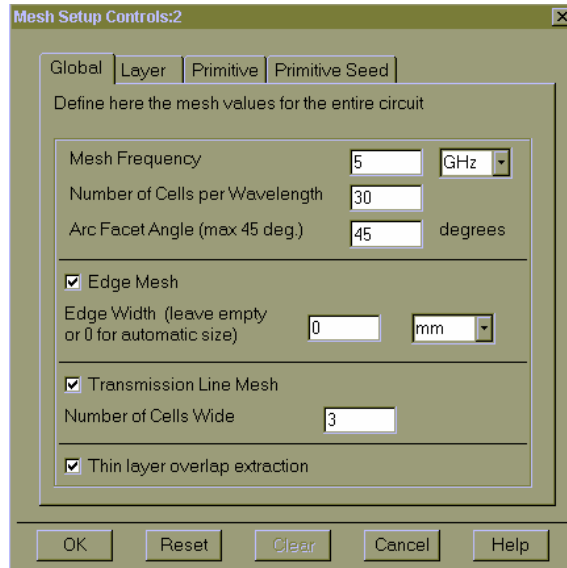

Figure 9. Port property definition.

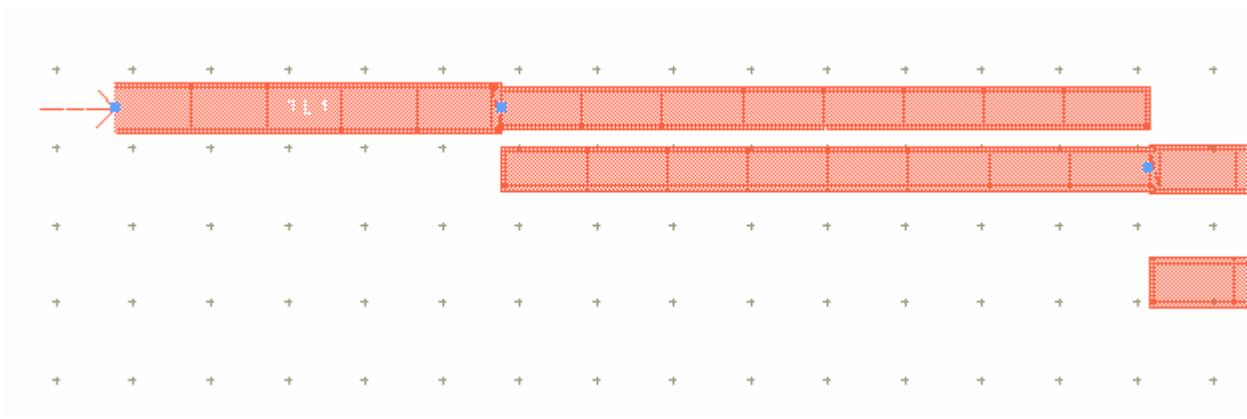*Figure 10. Mesh definition used in the Momentum simulation.*

Figure 11. Layout of microstrip filter showing the Momentum mesh.
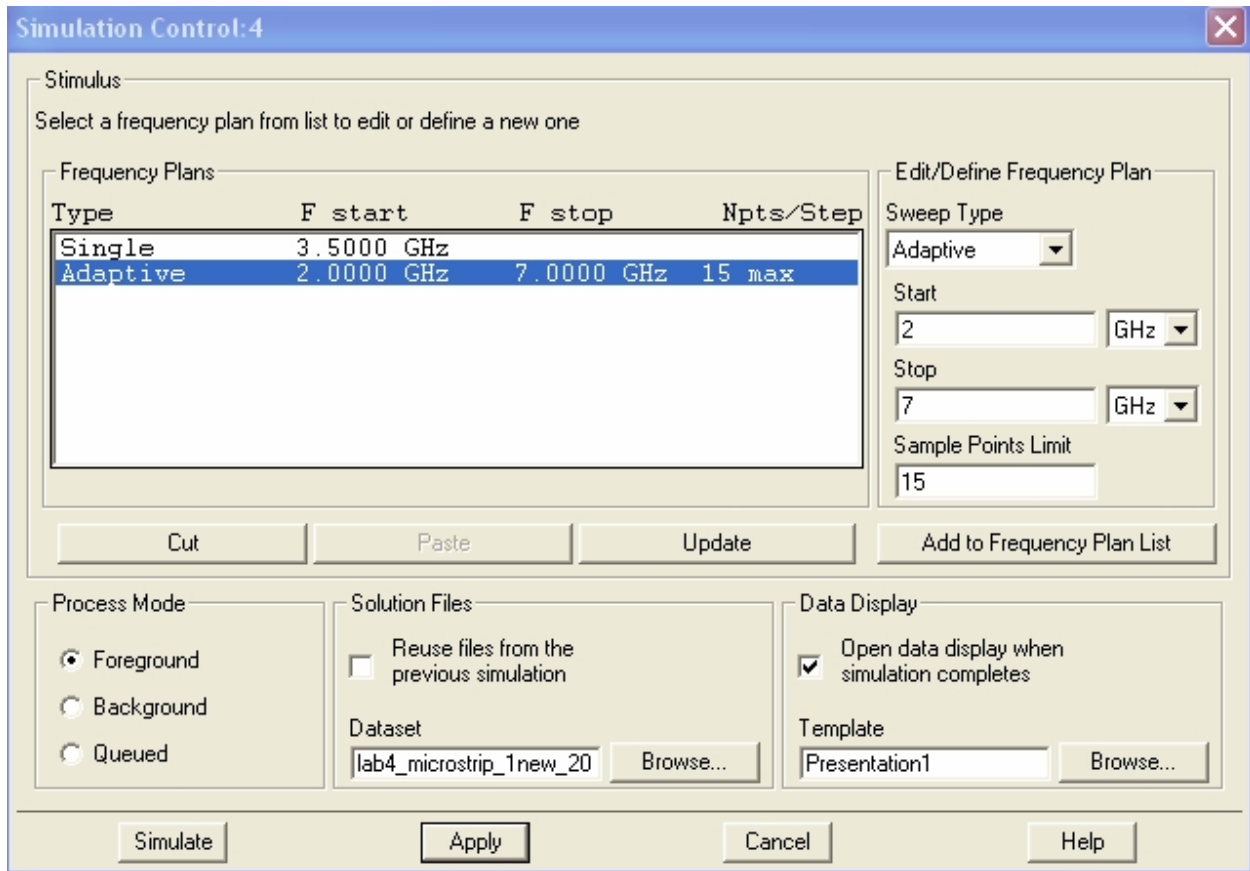
10

Figure 12. Simulation controls used in the Momentum analysis of the microstrip filter. In this lab you should simulate from 2-7 GHz using an "adaptive" pass and also specify a single frequency at fc.

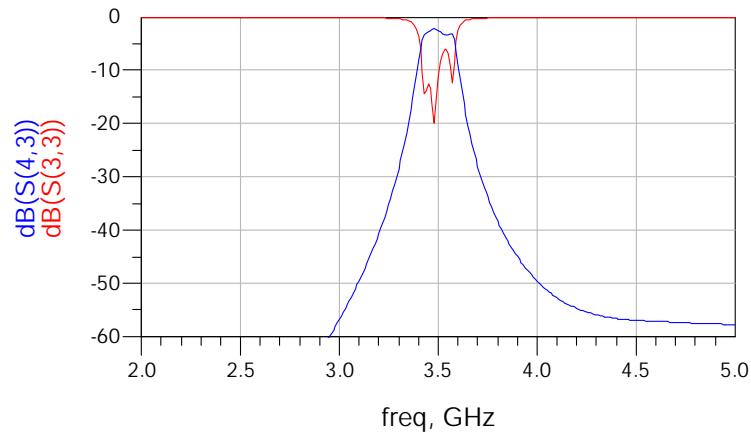*Figure 13. S-parameters for a microstrip filter as calculated by Momentum. Notice that the return loss in the passband has degraded (it is <10 dB in some parts of the pass-band)*
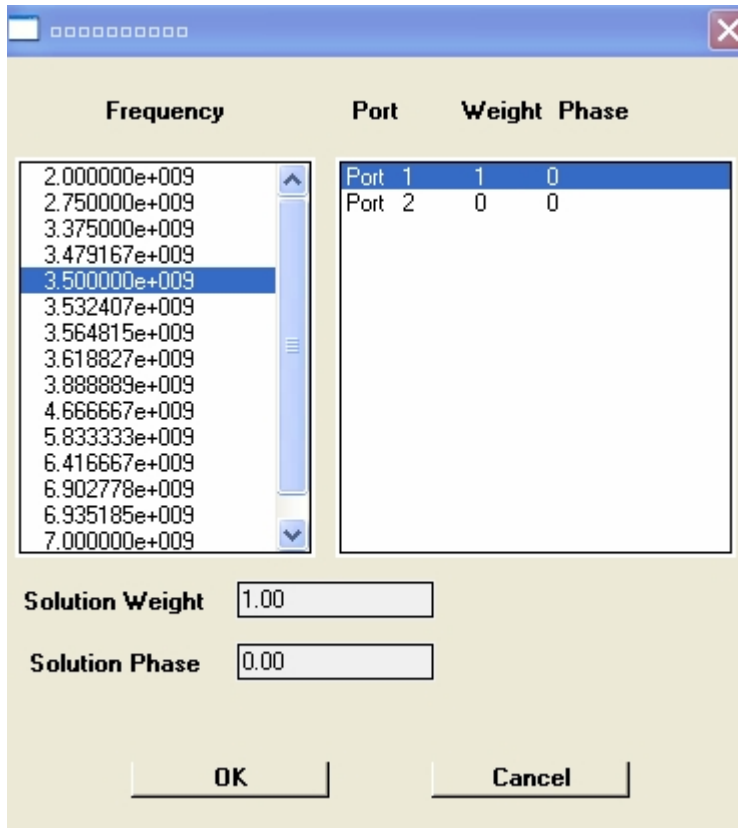
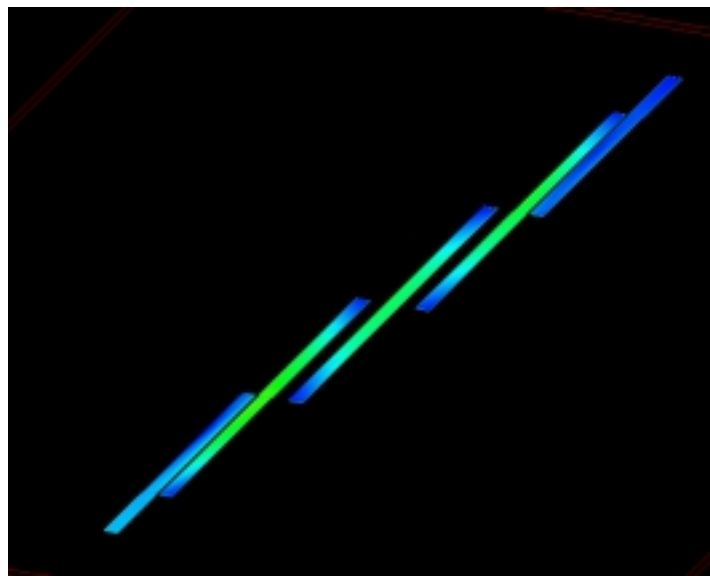*Figure 14. Port solution weights used to generate the plot of the electric current density.*

Figure 15. Representative plot of the electric current density at fc.

Laboratory Reporting Requirements

1) Your report should include the following:

   a) Printouts of the following:

      i) The ideal filter schematic and the simulated S11, S21 plot.

      ii) The microstrip filter schematic and the simulated S11, S21 plot.

      iii) The microstrip filter layout and the Momentum simulation results (S11, S21 plot).

   b) A short explanation of the following:

      i) Create a table that includes the insertion loss and return loss at fc and fstop for the ideal filter, the microstrip circuit simulation, and the microstrip Momentum simulation.

      ii) Describe the difference in the electric current density plots at 2 GHz and ~3.5 GHz. With an excitation at Port 1 for a frequency in the pass-band of the filter, do you see a high current density appearing at Port 2? (Explain why or why not.)

      iii) Explain what you believe to be the dominant loss factor in the filter. (You can investigate this topic using the circuit schematic.)