UNIVERSITY OF UTAH
ELECTRICAL AND COMPUTER ENGINEERING DEPARTMENT

ECE 5320/6322 LABORATORY #4

DESIGN AND TESTING OF LOW PASS MICROSTRIP FILTERS

Introduction and Goal

Filters are used in microwave circuits for the purpose of selecting and controlling transmission of signals that differ in frequency. Filters come in four basic varieties called LP (Low Pass), HP (High Pass), BP (Band Pass), and BS (Band Stop) where the names are descriptive of the functions performed. In reality, however, any circuit, which performs a frequency selective function, can be called a filter; e.g., an impedance matching circuit (or filter) suppresses reflections over a given band. Also, combinations of filter types exist which can legitimately be thought of as a single filter such as a filter designed to operate over two adjacent bands to pass one and stop the other. Such a filter would be designed as a single circuit having the fastest possible transition between the pass and stop functions so that the two bands of interest can be close together.

The **goal** of this lab is (1) to carry out the electrical design of two LP filters in distributed transmission line (TL) form, (2) to use LINECALC to define the geometries needed to implement the filters in our microstrip media and to use ADS to simulate and optimize the result, and (3) to build and test both filters. One of the filters comes from Pozar's example problem for the **"Image Matching Technique"** (Pozar, Section 8.2) and the other comes from the prototype design tables based on the **"Insertion Loss Technique"** (Section 8.3). As a result of this lab, you will learn how to scale the impedance and frequency of filter circuits and how to utilize prototype filter designs. You will also learn how to work with stub and step-Z microstrip elements to implement the filters.

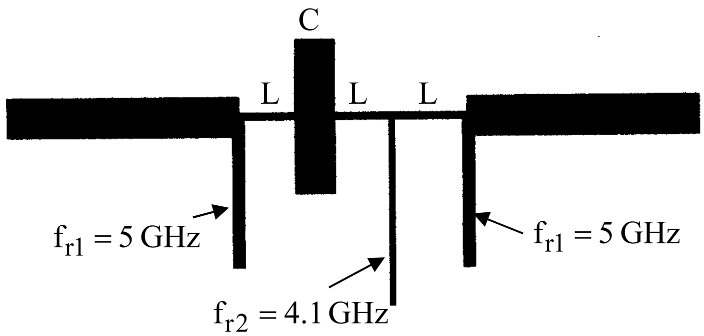
Preliminary Designs

In preparation for this lab, you should read Pozar, Chapter 8, Sections 8.2, 8.3 so that you understand the filter concepts involved.

Filter #1

Re-scale the composite filter designed in Pozar, Example 8.2 (p. 388) and shown in Figs. 8.19 and 8.20 (p. 389). The design frequency and impedance are [2 MHz, 75 Ohms] and the re-scaled values for our lab should be 50Ω and $f' = 4$ GHz. In redesigned filter, make sure to use $f_\infty = 5.0$ GHz for some of the sections. This would help to suppress the second-harmonic output of a 2.5 GHz amplifier from going through the filter.

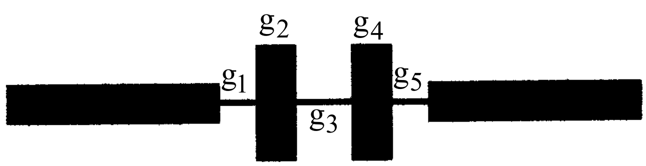
You should combine the pairs of series inductors along the line in Fig. 8.19 into a set of three lumped inductors, and you should identify which shunt resonators are producing the zeros (caused by resonant short circuits) at 2.05 and 2.5 MHz. Then, after scaling to the new frequency, you should verify that the new values of shunt resonance occur at the scaled frequencies corresponding to 2.05 and 2.5 MHz. Use high-Z "step impedance" lines to implement the three series inductors and a low-Z step line to implement the shunt capacitor. Then, use open circuit stub elements for the remaining three shunt resonators. Your filter will look something like the following in microstripline geometry.



You should have the electrical design finished before coming to Lab, and then use LINECALC to find the line dimensions and ADS to simulate the filter response. We will try to use optimization on this filter to sharpen its cutoff performance at the critical frequency.

Filter #2

Use Table 8.4 (p. 396) to design an N = 5, 0.5 dB ripple, LP filter using step elements. Design for a cutoff frequency of 4 GHz. <u>Start and end the filter with a series inductor</u> and use High-Z and Low-Z step elements to implement the series inductance and shunt capacitance values indicated in the prototype filter table. Your design will look something like the following in microstripline realization.



If you desire, you may up the order of this filter to N = 7. Use the design curves in Fig. 8.27a (p. 397) to predict the attenuation of the filter at frequencies above the cutoff frequency. Note that these values are often part of the filter specifications and are typically used to determine the order N of the filter when high attenuation is required at specific frequencies above the cutoff frequency e.g. at 5 GHz for this filter. One can easily find the point where ω is 25% above ω_c, for example, and then find the order N required to give 25 dB attenuation.

Once again, come to lab prepared with the electrical design and use LINECALC and ADS to simulate, optimize, and layout the mask for your circuit.

Laboratory Procedure

In Lab 4 you should proceed as follows:

1. Use your electrical designs for the filters in LINECALC to obtain the physical dimensions required for ADS simulations. Select a cutoff frequency for each filter to be 4 GHz for your designs and make sure that the parameters in LINECALC reflect this selected design frequency.

2. Use ADS to simulate the design of each coupler. Adjust the designs as required to obtain the desired transmission over the pass band and attenuation above the cutoff frequency. You should try to utilize the ADS optimization process to accomplish this. Save copies of your starting design and be prepared to try several different types of optimization and to optimize over different sets of variables. Be sure to collect the printouts needed to demonstrate that the designs are working as desired to document your work for your report.

3. When each circuit simulation shows the desired result, reduce the simulation schematic to a layout drawing and transport the results to AutoCAD. In AutoCAD, you can place the two circuits side by side on a single circuit board and connect each one to a pair of template ports.

4. When your microstrip circuit is returned, you should make careful dimensional measurements on the circuits and see how they compare to your original mask dimensions. Small errors in resonant stub widths and lengths can make a big difference in the filter performance. If you do see differences, you will probably want to make adjustments in the simulation model to make the model as close as possible to the fabricated circuit. Remember that the goal is to see how well ADS predicts the performance of the actual circuit.

Write-Up

The report for Lab 4 will be similar to the previous labs and according to instructions from your TA. Hole-punch your report and put it in a three-ring binder. After it is organized and documented so that it reads like a report, staple it together in the upper left-hand corner and turn it in to your TA for grading. Lab 4 will be due at the end of the next lab. Use a three-ring binder to store your lab reports so that they can be reviewed together, if necessary, at the end of the semester.