Maximally Flat Series Stub Low-Pass Filter Design Lab

Objective:

Design, optimize, build, and test a Low Pass Filter

1. Review the text and design a maximally flat low pass filter with a cutoff frequency of 2 GHz. The filter should provide 15 dB of attenuation at 3 GHz. From the low-pass filter prototype, apply Richard's transformation and the appropriate Kuroda identities to realize a series stub low-pass filter for microstrip lines.
2. HPADS simulation.
   A. Analyze the series stub low-pass filter design assuming ideal lossless lines using HPADS. Make sure that the design is matched at both ports and the proper insertion losses are realized. Print out your schematic and pertinent plots of the scattering parameters.
   B. Build your schematic using microstrip lines. Your substrate - specified using MSUB - is a 30 mil GML 1000 Gil Laminate substrate with relative permittivity of 3.20 and a loss tangent of 0.003. The conductor is 3 μm thick copper (5.7×10^7 S/m). Use MLIN for the uniform microstrip lines, MSTEP for any abrupt changes in line widths, and MTEE for the splits. Use Linecalc to determine the width of the microstrip lines, and the lengths and widths of microstrip lines. Make sure that the line widths are physically realizable. If not, some decisions will have to be made in modifying the circuit.
   C. Add 1 inch of 50 ohm line to both ports to allow the soldering on the connectors and accurate measurement. Print out the pertinent S-parameters predicted for the unoptimized microstrip filter.
   D. Next, optimize the circuit performance with HPADS. You will have to change line lengths and possibly line widths to optimize your circuit performance. That is adjust the resonant frequency to 2.0 GHz. Also, monitor the attenuation in the pass band and the roll-off in the stop band. Decide which parameters are important for your design and prepare to plot them from 1 -3 GHz.
   E. When you are satisfied with the response of the circuit, bring a print out of the schematic of your circuit design to the instructor who will create a layout and mask for you.

Measure your circuit:

1. Fabricate your circuit using the PC Board Etching process from the masks based on your circuit provided by the instructor.
2. Solder on edge mounted SMA connectors to your circuit board.
3. Calibrate the network analyzer from 1-3 GHz (if necessary).
4. Attach ports one and two to the network analyzer.
5. View the different parameters on the Network analyzer for the port pair. Save the data to a PC formatted disk. This is done by pressing the "Save/Recall" hard key. Then, press the "Define Disk Save" soft key. Next, turn "Data Only" to on (again a soft key). Then press the "Return" soft key at the bottom. Next, press the "Save State" softkey. This will save 4 files. Two will end in *.D1 and *.D2. These data

files contain the data for the s-parameters from channel 1 and channel 2, respectively. The other two files *.S1 and *.S2 are also the S-parameters, but stored in a touchtone format, which can be read directly into HPADS. See step 7 below

6. Plot your network analyzer results against your HPADS results on the same graph. To plot the analyzer results within HPADS, you need to copy the *.S1 and *.S2 files to the *data* directory of your project folder. Rename your files with a descriptive name for your circuit. Also, *make sure* you give the files the extension *.s2p. Next, open a new schematic window in HPADS under your lab project. Under the file menu, choose "Insert Template", and insert an S-parameter template. Edit the stimulus to the range of 1.0 to 3 GHz start and stop frequencies and an appropriate number of sample points (the number of points is arbitrary and does not have to coincide with the measurements). Choose the "Data Items" library. Then, place an "S2P: 2-port S-parameter file" on the schematic. Double click on the 2 port to open up the parameter window. Make sure that the "File" parameter is highlighted, and then select "Browse…". Your local data directory will appear listing all the .s2p files. Pick the one you wish to plot, then select "Apply" and then "OK". Next, connect ports 1 and 2 to the term 1 and 2, respectively. Then, connect port 3 to a ground (this port is only used for active device models with DC bias. For passive devices, it must be grounded for proper operation). Save your schematic and then simulate it. You can now plot it and overlay plots in a data display window.

Lab Report:

1. Complete your lab report. Include sections on: 1) theory, 2) HPADS simulations (ideal, unoptimized microstrip, optimized microstrip), 3) fabrication, 4) measured results compared with HPADS results, 5) conclusions.
2. When you draw your conclusions, address the following questions relating the measured results to the theoretical: Did the circuit perform as expected? Were there shifts in the resonant frequency? If so, what mechanisms would result in this? What is the performance of the circuit in the pass band or the stop band? What losses were observed? What loss mechanisms or other parasitic effects deteriorate the performance from the ideal?