ECEN 4634/5634, Microwave & RF Laboratory, Fall 2016  
Taylor Barton and Zoya Popovic  

PRELAB AND CAD LAB 7 AND 8: PASSIVE MICROSTRIP CIRCUIT DESIGN AND SMALL-SIGNAL AMPLIFIER DESIGN (5634 STUDENTS ONLY)

<table>
<thead>
<tr>
<th>Assigned</th>
<th>October 10</th>
<th>Goal: to learn how to design some basic microstrip circuits using NI/AWR Microwave Office</th>
</tr>
</thead>
<tbody>
<tr>
<td>Due</td>
<td>Monday, October 31st in class Start on time!</td>
<td></td>
</tr>
</tbody>
</table>

In this CAD lab, you will design several of the microstrip circuits you have already measured in Lab 6, and components you will measure in Lab 10. For all microstrip implementations and layout, you will use an FR4 substrate that is 30 mils thick (1 mil = 25.4 µm) and has a relative permittivity of 4.2. You are also required to write a 5-6 page paper in IEEE format, given on the class web page (or by visiting the IEEE web page).

Part 1: Passive circuits

1. Design a bandpass filter centered at 3GHz with a 2-GHz passband and at least 20dB attenuation at 1 and 5GHz and as small of an insertion loss in the passband as you can accomplish. You are given freedom to use any topology you wish. Please explain briefly your design and why you chose the particular topology. Plot all relevant S-parameters and answer the following:
   - what is the insertion loss in the passband?
   - what is the return loss in the passband?
   - how steep is the roll-off?
   - what is the attenuation in the higher stop band and how wide is the higher stop band?

After you finish the circuit design, obtain a layout of your filter. If you decide to use any lumped components (capacitors or inductors), you should leave a gap that is the correct size for the particular package you are using. If you use metalized vias as short circuits, you can count on about 0.2-0.5nH inductance for the via hole. If you decide to use lumped components, you should download s2p two-port S-parameter files from the manufacturer since these components are far from ideal. Some manufacturers are ATC (capacitors) and CoilCraft (inductors).

2. Design a feed network at 2.4GHz for a two-element antenna array matched to 50 ohms. The antenna impedance at 2.4GHz is 

   \[ Z_A = 73 + j10 \Omega \]

   Plot the relevant parameters and perform a layout for your circuit. If the antennas are dipoles, how would you connect your matching circuit to them (open question, there are more than one answer)? Is your circuit (a) lossless, (b) matched and (c) reciprocal?

3. Design a 0-180 degree 3-dB coupler centered at 3GHz and plot all relevant S-parameters. What is the bandwidth of the coupler? Investigate what happens to the performance if the coupled and through ports are connected to loads that are not balanced (not equal) and not perfectly impedance-matched (not purely 50Ω). For the last part, you can assume a ±10% impedance variation.
**Part 2: Designing a small-signal linear amplifier**

In this part of the CAD lab, you will design a small-signal amplifier centered at 2, 3 or 4GHz and matched for gain. The device we will use is an Avago GaAs HEMT with specifications given here: [http://www.avagotech.com/products/wireless/transistors/fet/atf-34143#overview](http://www.avagotech.com/products/wireless/transistors/fet/atf-34143#overview)

The .s2p files for several bias points can be downloaded from the web page.

2.1. Choose a bias point and find the input and output impedances of the FET at the design frequency from the S parameters, keeping in mind they are normalized to 50Ω.
   - Design two separate matching networks, one to match the input impedance (a s1p file made from the s2p file with a 50-ohm load on port 2) and one to match the output impedance (another one-port network, i.e. complex load). This effectively means that you are assuming a unilateral transistor. You can do the design on paper (Smith chart), or using a CAD tool.
   - You can design either a lumped-element or a transmission-line (e.g. open stub) match. Please specify if you are making the choice for size, cost, performance, etc. When doing this, make sure you chose reasonable values for the lumped elements, and use their measured parameters (i.e. including the parasitic). A good source for capacitors is ATC, and for inductors Coilcraft, but you can use other manufacturers as well. In your report, the matching circuit performance should include both s11 and s22 on the same plot.

2.2. Connect the matching networks you designed in 2.1 at the input and output of the transistor and simulate the performance. Discuss the results.

2.3. Now re-design your amplifier as a bilateral device, so that it is matched for gain at your chosen design frequency. This means that you meet your specs at the design frequency and in your specified band. The reason you need to modify the design is that the unilateral matching changes when you have a bilateral device, i.e. your transistor. You can either use bilateral design rules, or just do it in the circuit simulator by tuning. Plot the S-parameters and summarize the performance in a table. Check the stability and comment on it.

2.4. Next change the device s2p file to a s3p file by enabling access to the source (S-parameters are measured in a common-source configuration). When a device is mounted in a microstrip circuit, there is inductance associated with the lead that goes to ground. This inductance typically has values between 0.2 and 1nH, depending on the substrate thickness and device package. Resimulate your amplifier for 0.1, 0.5 and 1nH values of the source inductance. Discuss the results, especially stability. You can use a figure or a table to do this.

2.5. For a 0.5-nH source inductance, re-design your amplifier for best match and gain at the design frequency. This is just an adjustment to the design from 2.4 when the inductance is added. Plot the S-parameters.
Adding bias lines and completing the amplifier design

2.6. Design a bias-Tee network for an amplifier that operates at 3GHz with a 20% bandwidth. To design a bias line, you will need to prevent the DC to flow to the RF input port using a blocking capacitor. ATC – American Technical Ceramics – has a good product note (called RF Capacitor Handbook) that you might want to register to receive. List the relevant specifications given by manufacturers for surface-mount capacitors, along with a range of typical values. Choose a capacitance value that will be a good block at your operating frequency and simulate the effect of placing it in series in a 50-Ω transmission line. First simulate an ideal series capacitor, and then add the parasitics and compare the S-parameters.

To complete the bias line design, you will also need to prevent the RF from flowing into the DC input port. This is done either with a lumped element inductor (choke) or with a quarter-wave shorted line, as we discussed in class. Collect specifications for microwave inductors from two different vendors (e.g. AVX, Piconics, Coilcraft). List the relevant specifications given by manufacturers for surface-mount inductors, along with typical values. What is the highest Q and resonant frequency you found? Choose a commercially available inductor and combine it with your capacitor to design a bias-tee network at your design frequency. State clearly which surface mount lumped elements you used in the design and show a plot of the relevant S-parameters. You are also welcome to use a combination of lumped elements and transmission lines.

2.7. Now add your bias lines to the amplifier design from Part 2.5 above. The bias lines can be connected in different places in the input (gate) and output (drain) networks. Does the position of the bias lines affect the amplifier design? Discuss your results and plot your final design S parameters.

Make sure you include a readable schematic of your complete amplifier design in your report. Your guideline can be to include all information that your lab mate would need to repeat your design.