1 Laboratory 5

Power Matched Amplifiers and Filters

Summary: Students will design, assemble, and test their own low-pass filter, with a cutt-off frequency of 6 GHz, and a power matched HBT amplifier at 4 GHz. They will also learn how to use a microwave power sensor and use it to measure the P1dB of their amplifier.

1.1 Useful resources

https://www.rogerscorp.com/documents/725/acs/RO3200-Laminate-Data-Sheet-RO3203-RO3206-RO32
pdf

1.2 Low-Pass Filter Preparation

1. Open up a new MWO file or copy an existing template.

2. You will be designing a low-pass, odd-order, stepped impedance filter. The maximum line width will be 120 mils and the minimum line width 6 mils.

3. You are required to design at least a 5 element LPF, but if you would like to design a higher order filter, you are free to do so. Your only limitation is size, provided at the end of this section. For higher order filters you can use bends in the high impedance sections of line to make your design more compact.

4. Design a 5 element, lumped, low-pass prototype with either a Buttworth response (using your class notes), or the much more aggressive Chebyshev response, the design equations provided in 1.3 to 1.7. Note that the terminating impedances for both cases are equal.

5. Using the scaling relations from your class notes, create the lumped equivalent of your filter in MWO and simulate it, verifying that your design is working as expected. Include the schematic and simulated response within your report.

6. Using MWO TXLINE, calculate the characteristic impedance of lines with widths of 120 and 6 mils. We will be using the same Rogers 3210 substrate, 50 mils thick, as used in Laboratory 2 and 4.

7. Now, create the stepped impedance equivalent of your filter, using the maximum and minimum impedances above, and the relations from your class notes, to calculate the electrical lengths of the lines. Once the electrical lengths are determined, you will need to use MWO to calculate the physical lengths of the high and low impedance lines.

8. To model your filter as realistically as possible, include in it the effects of the steps in impedance, from thin to thick sections of microstrip line. These can be modeled using the **Elements** ⇒ **Microstrip** ⇒ **Junctions** ⇒ **MSTEP** element. Note that there are several versions, one that is closed form, one that is EM based, and several intelligent versions, where you are not required to enter in the adjacent widths of lines. The EM versions are more accurate, but take slightly longer to simulate.

9. Manually tune the line lengths of your design as necessary. **Note that your circuit is symmetric, so you only need to tune 3 line lengths for a 5 element filter**.

10. Plot the results of your stepped impedance filter along side your lumped equivalent filter and include this graph in your report. Comment on the differences between the two.

11. Go to the layout window and snap your circuit elements together.

12. Your entire design must fit within a 1 by 1 inch square including the room required for the connectors.

13. **Email your file to drussell@caltech.edu no later than Thursday at noon.**

The g_k values ($k = 1 \ldots N$) for the Butterworth reponse are given by:

$$g_k = 2sin\left[\frac{(2k-1)\pi}{2N}\right] \tag{1.1}$$

The g_k values ($k = 1 \ldots N$) for the Chebyshev reponse are given by:

$$g_1 = \frac{2a_1}{\gamma} \tag{1.2}$$

$$g_k = \frac{4a_{k-1}a_k}{b_{k-1}g_{k-1}} \tag{1.3}$$

where

$$a_k = sin\left(\frac{(2k-1)\pi}{2N}\right) \tag{1.4}$$

$$b_k = \gamma^2 + sin^2\left(\frac{k\pi}{N}\right) \tag{1.5}$$

$$\beta = ln\left(coth\left[\frac{A_m}{17.37}\right]\right) \tag{1.6}$$

$$\gamma = sinh\left(\frac{\beta}{2N}\right) \tag{1.7}$$

and A_m is the maximum pass band ripple in dB, which is also the attenuation provided at the cutt-off frequency. For your design a value of 0.1 dB is suitable.

1.3 Power Matched Amplifier Preparation

1. Using your MWO file and PCB from Laboratory 3 as a basis, design an amplifier at 4 GHz to maximize the transducer power gain, G_T. For this design you are interested in power matching both the input and the output, while simultaneously achieving the maximum possible gain. From your class notes, you know that this is accomplished by creating two matching networks. The first, for the output, transforms 50 Ω to $S_{22}^{\star}$. The second, for the input, transforms 50 Ω to $\Gamma_{in}^{\star}$.

2. As a design goal, aim to achieve better than 12 dB input and output match, and greater than 14 dB of gain.

3. As you might have guessed, the gain, match, and stability of the transistor are dependent on the bias condition. You will likely want to use a much higher bias for this design, than that used for your LNA. You can treat this as a free parameter, if you run into a bind in the design, say with matching or stability, choose a different bias point and iterate.

4. Determine the necessary bias and LC networks at the input and output of the transistor. Remember to think about biasing the transistor when you do so, make sure you do not have any continuous DC path from the base or collector ground.

5. You can use either lumped L's and C's for your analysis, or the .s2p files for the Johnanson passives.

6. Bring your final component values with you on Friday, **There is not enough time to do the design in class.**.

1.4 Required Equipment

Descrition	Model	Quantity	Notes
FFox	N9917A or N9918A	1	-
Cables	TM26-3131-36	3	-
Capacitor/Inductor Design Kit	S402DS	1	Johanson
RF Generator	HMC-T2220	1	Hittite
Power Sensor	PWR-8FS	1	Mini-Circuits

Table 1: Required Equipment

1.5 Circuit Assembly

1. Deburr your low-pass filter and solder on its SMA connectors.

2. Locate the necessary inductors, capacitors, and resistors and make the necessary changes to your original LNA. Note that we have a hot-air pencil that will make soldering of your 0402 components much easier.

1.6 LPF Measurements

1. Using the methods from the previous laboratories, calibrate the FFox in preparation for measurement of your filters S-parameters. Set the input power to ≥ -15 dBm. Frequency range should be 0.1-18 GHz.

2. Take the S-parameters of your filter and save the data to .s2p file for your report.

3. As always, remember to take a photo of your test setup.

1.7 Amplifier Measurements

1. Using the methods from the previous laboratories, calibrate the FFox in preparation for measurement of your amplifier S-parameters. Set the input power to -30 dBm. Frequency range should be 0.1-18 GHz.

2. Set up a power supply with the necessary OVP and OCP.

3. Take the S-parameters of your amplifier and save the data to .s2p file for your report.

4. If you see results widely different from what you expect, such as very low gain, set the FFox to SA mode and check its spectrum for oscillations.

5. After capturing your amplifiers S-parameters (remember to note down the bias) and saving the data to file, switch your FFox to SA mode.

6. Using the instructions from Laboratory 1, you will now measure your amplifiers P1dB point at 4 GHz. Instead of using the FFox to measure power, you will use a power sensor.

7. The power sensor, in general, provides a more accurate measure of power than that obtained with the FFox in Spectrum Analyzer Mode.

8. Install the power sensor software onto your computer using the USB-DVD drive.

9. Hook up the power sensor's USB output to your computer and bring-up its application.

10. Use the power sensor to calibrate the output power from the signal generator, in 1 dB steps, from -20 dBm to 0 dBm, as was done in Laboratory 1.

11. Following calibration, measure the output power from your amplifier at each of your calibrated input powers. **You do not need to include a 10 dB attenuator between your amplifier and the power sensor.**

12. As always, remember to take a photo of your test setup.

13. Measure and record the wide-band spectrum of your amplifier terminated with your LPF. Do so with the FFox in spectrum analyzer mode, 0.1-18 GHz, with a 10 dB attenuator at its input. Measure the spectrum with your LPF at the input and output of your amplifier. You can terminate the input with a 50 Ω load for these measurements. Does your circuit oscillate?

14. Record the spectrum and make sure to remove the effect of the 10 dB attenuator from your measurements.

15. Before finishing, remove the LPF and measure the output spectrum at -20 dBm and 0 dBm input powers.

1.8 Data Analysis and Report

Reports are due by noon October 31st, emailed to drussell@caltech.edu and to dwinker@caltech.edu
Remember to follow all the general instructions from previous reports. You should include any derivations and/or circuit simulations from the pre-lab, including schematics and simulated results. For you amplifier, remember to include the bias values (voltage and current) in your plots.

1. Plot out your lumped (LC) and stepped impedance simulations on the same plot.

2. Plot out your stepped impedance simulation and measurement on the same plot. Comment on any differences. How closely does your simulation and measurements match? Comment on any differences and the possible reasons why.

3. Plot our your simulated and measured amplifier S-parameters. Comment on any differences or struggles you encountered during the lab and what you would do to improve your design given more time.

4. Plot out the sweep of input vs output power for your amplifier in determination of P1dB. What is the P1dB?

5. What is the efficiency of your amplifier at its P1dB point?

6. Include in your report your wideband spectrum measurements at -20 dBm and 0 dBm input powers. What percent non-linearity do you see at 0 dBm in the second harmonic?

7. Include your measured spectrums with the LPF at the input and output of your amplifier, was your amplifier stable under both conditions? Does it match what you would expect from your simulations?