

# Applied Electromagnetics – Final Project

---

*Design of a Class A RF Transistor Amplifier*

*John Reid, Matthew Drago*

*Fall 2017*

## Contents

Table of Figures:	4
Introduction:	5
Background Theory:	5
Biasing the Transistor:	6
Stability Circles:	7
Unilateral Design: Calculating the Gain	8
Unilateral Gain Circle Calculations:	9
Bilateral Amplifier Design:	10
Design and Simulation:	12
Biasing the Transistor:	13
Multisim Simulation:	14
RF Blocking Network:	16
Sonnet Simulations:	20
Stability Calculations:	22
Gain Calculations:	24
RF Matching Networks:	26
Sonnet Simulations:	28
Testing and Analysis:	31
Bias Network:	31
RF response:	31
Other regions of interest:	32
Cascaded System Test:	38
Suggestions for Improvements:	41
Conclusion:	42
Appendix A: Images from the BFG403W Datasheet	43
Appendix B: Matlab code to...	46
Draw Circles on a Smith Chart:	46
Plot the Scattering Parameters as a Function of Frequency:	47
Plot the Stability Circles on a Smith Chart:	48
Plot the Input and Output Gain Circles on a Smith Chart:	48

Appendix C: References	50
Multisim:	50
Textbooks:	50
External Plots:	50

## Table of Figures:

Figure 1: Schematic of a Class A Amplifier with RF blocking networks and DC blocking capacitors.	5
Figure 2: Low and high frequency hybrid- $\pi$ models for a BJT.	10
Figure 3: The amplifier under DC conditions	11
Figure 4: Simulation of the biasing network for the amplifier using computed values.	12
Figure 5: Simulation of the biasing network for the amplifier using actual component values.	13
Figure 6: Simulation using a value for $R_2$ that makes VCE equal to 2V.	13
Figure 7: of the RF amplifier. RF blocking networks are outlined in blue.	15
Figure 8: Frequency response of a 530nH inductor and a 220pF capacitor.	15
Figure 9: Results of the Sonnet simulation for the inductive transmission line.	18
Figure 10: Results of the Sonnet simulation for the capacitive transmission line.	19
Figure 11: Table of magnitude and phase of S-parameters for the BFG403W.	20
Figure 12: Matlab plot of $S_{11}$ (blue), $S_{12}$ (yellow), $S_{21}$ (red), and $S_{22}$ (purple) in dB as a function of frequency.	20
Figure 13: Smith chart showing stability circles for the transistor at 2.4GHz. The stability circles are shown in red and blue.	21
Figure 14: Smith chart of the Source Gain circles showing that 2dB of gain must be sacrificed to bias the transistor.	22
Figure 15: Smith chart of the Load Gain circles showing that 4dB of gain must be sacrificed to bias the transistor.	22
Figure 16: Results of Sonnet simulation for input matching network transmission line.	23
Figure 17: Results of Sonnet simulation for output matching network transmission line.	24
Figure 18: Plot of capacitor self resonance vs. capacitor value.	25
Figure 19: The PCB layout of the amplifier in Eagle PCB.	26
Figure 20: Amplifier with 20dB attenuators.	27
Figure 21: Amplifier forward gain ( $S_{21}$ ) pass-band response for $V_{CC}=6V$ .	27
Figure 22: Amplifier forward gain ( $S_{21}$ ) pass-band response for $V_{CC}=5.44V$	28
Figure 23: Wideband response of amplifier.	29
Figure 24: Reverse gain ( $S_{12}$ ) of amplifier.	29
Figure 25: Group delay of amplifier.	30
Figure 26: Amplifier deviation from linear phase.	31
Figure 27: Magnitude of $S_{21}$ around the large spike in amplifier gain after the pass-band.	32
Figure 28: Phase of $S_{21}$ around the large spike in amplifier gain after the pass-band.	32
Figure 29: Fractal antenna used for transmission and patch antenna used to receive signal	33
Figure 30: Testing configuration for the cascaded system.	33
Figure 31: Wideband response of cascaded system.	34
Figure 32: Passband response of cascaded system.	35
Figure 33: DC current gain as a function of collector current (figure 3).	37
Figure 34: Transition frequency as a function of collector current (figure 5).	37
Figure 35: MSG as a function of collector current (figure 7).	38
Figure 36: Gain as a function of frequency where $I_C = 3mA$ and $V_{CE} = 2C$ (figure 8).	38

## Introduction:

In applications where data is transmitted wirelessly, there is a specific set of sub-circuits that the signal must pass through before being transmitted. First the signal must be filtered to remove any frequencies out of the signal bandwidth to avoid contaminating the information. As it passes through the filter, the signal loses some of its strength and needs to be amplified. Only after it is amplified can it then be transmitted through an antenna.

This project will focus on the design, construction and testing of an RF amplifier that will operate in the WiFi bandwidth around 2.4 GHz. The amplifier design specifications are given below:

Bandwidth (GHz)	Supply Voltage (V)	Minimum Gain (dB)	Variation at BW (+/- dB)	Center Freq. (GHz)
2.401-2.473	+6 V	+10 dB	0.5 dB	2.437

**\*Note that all Matlab Code uses functions obtained from the 'RF Toolbox' App in Matlab\***

## Background Theory:

The RF amplifier in this report can be examined like a 2-port network. As such, the Scattering Parameter (S-parameters) transformation matrix becomes the basis of all the calculations. For simplicity, the design will be a simple Class A transistor amplifier.

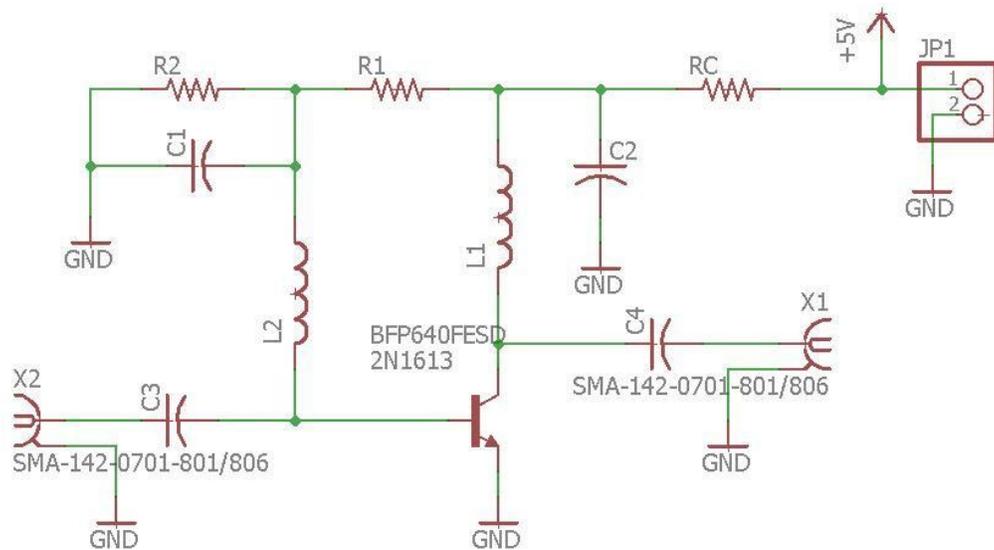


Figure 1: Schematic of a Class A Amplifier with RF blocking networks and DC blocking capacitors.

The schematic for the amplifier that will be constructed in this project is shown in Figure 1. The capacitors C3 and C4 are DC blocking capacitors used to isolate the amplifier from any external biasing. Inductors L1 and L2 form the RF blocking network that will prevent the 2.4GHz signal from escaping into the DC source.

This transistor used in this design will be the BFG403W. It is a 17GHz wideband transistor with a DC current gain of 80. This model was chosen to compare with other RF transistor amplifier designs. In a previous design, the amplifier did not perform as well as expected, having only 3dB of gain in the passband. This gain was then improved to approximately 6dB in the passband by removing a shunt capacitor that was added to increase the amplifier stability. Due to these difficulties, this design will attempt to maximize the amplifier gain by adjusting some of the biasing parameters.

### Biasing the Transistor:

For the amplifier to function properly, the transistor needs to be biased. Using the DC model in Figure 3,  $R_c$  is set using values for  $V_{ce}$  and  $I_c$  chosen from the datasheet. Once these values are chosen, the value of  $R_2$  calculated using the formula

$$R_2 = \frac{V_B}{I_2}$$

where  $I_2$  is the current flowing through resistor  $R_2$ . Typically for this type of biasing network,  $I_2$  is chosen to be approximately five times the base current. Thus

$$R_2 = \frac{V_B}{5I_B} = \frac{V_B}{5I_C} h_{fe}$$

Next the value for  $R_1$  is calculated using the equation

$$R_1 = \frac{V_{CE} - V_{BE}}{I_2 + I_B} = \frac{V_{CE} - V_{BE}}{6I_B} = \left( \frac{V_{cc} - V_{BE}}{6I_C} \right) h_{fe}$$

Finally the collector resistor is calculated using the formula

$$R_C = \frac{V_{cc} - V_{CE}}{I_C + I_1} = \frac{V_{cc} - V_{CE}}{I_C(1 + \frac{6}{h_{fe}})}$$

### Stability Circles:

The next consideration in amplifier design is the stability of the system. Under certain conditions it is possible that an amplifier may apply gain to unwanted reflected signals and cause oscillation. To prevent this, there are different stability techniques which can be applied. An excellent graphical tool which can be used to stabilize an amplifier design is the stability circles method. This works by mapping a circle for both the output stability and input stability, with each circle created using the S-parameters and input

and output reflection coefficients. These circles can be plotted along with a smith chart centered at the origin, with any overlapping regions representing impedances that would produce instability. The amplifier will be designed to have maximum stability across the entire range of operation; therefore our objective is to move the regions of instability, meaning any region of the Smith Chart that falls within the input/output stability circles, completely off the Smith Chart to maximize the stability over the desired bandwidth.

To calculate these stability circles, we must determine their center points and radius values, beginning with the variable  $\Delta$  that is defined as

$$\Delta = S_{11}S_{22} - S_{12}S_{21}$$

The center point for the output stability circle can then be calculated as

$$C_L = \frac{S_{22}^* - S_{11}\Delta^*}{|S_{22}^2 + |\Delta|^2}$$

The radius of the output stability circle is determined by the equation

$$R_L = \left| \frac{S_{12}S_{21}}{|S_{22}^2 + |\Delta|^2} \right|$$

Likewise, the input stability circle can be calculated as

$$C_S = \frac{S_{11}^* - S_{22}\Delta^*}{|S_{11}^2 + |\Delta|^2}$$

The radius of the input stability circle is determined by the equation

$$R_S = \left| \frac{S_{12}S_{21}}{|S_{11}^2 + |\Delta|^2} \right|$$

## Unilateral Design: Calculating the Gain

Through the unilateral design method, computing the gain becomes simple under the assumption that  $S_{12} = 0$ . This is the Unilateral Gain assumption. Although this significantly simplifies the calculations needed to determine the amplifier gain, the tradeoff is that it increases the uncertainty of the gain being calculated. Under this assumption, the total gain of the amplifier is given by the equation

$$G_{tu} = G_S * G_0 * G_L$$

Where  $G_S$  is the Source Gain, and is given by the equation

$$G_S = \frac{1 - |\Gamma_S|^2}{|1 - S_{11}\Gamma_S|^2}$$

$G_0$  is the Maximum Power Gain of the transistor, and  $G_L$  is the Load Gain calculated from the formula

$$G_L = \frac{1-|\Gamma_L|^2}{|1-S_{22}\Gamma_L|^2}$$

where the input, output, load and source reflection coefficients are given by

$$\Gamma_S = S_{11}^* = \Gamma_{in}^*, \quad \Gamma_L = S_{22}^* = \Gamma_{out}^*$$

Because of the increased potential for error in our gain calculations due to the unilateral assumptions, it is necessary to calculate what is called the unilateral figure of merit, which provides boundary conditions to estimate the range of variation in gain calculations due to error. These boundary conditions are calculated by first solving for U with the equation

$$U = \frac{|S_{11}||S_{22}||S_{12}||S_{21}|}{(1-|S_{11}|^2)(1-|S_{22}|^2)}$$

Where the upper bound can be calculated with the equation

$$U_{upper} = \frac{1}{(1-U)^2}$$

and the lower bound can be calculated with the equation

$$U_{lower} = \frac{1}{(1+U)^2}$$

These values can then be converted into decibels in order to obtain the maximum and minimum boundaries for the overall gain of the system.

Another formula that is commonly used to get an indication of the amplifier's stability is known as Rollett's stability factor, or the K-factor. A K-factor that is greater than one means that the amplifier is unconditionally stable. If K is less than 1, then there may be instability problems. Below is the equation for K-factor, which utilizes the  $\Delta$  we discussed earlier.

$$K = \frac{1-|S_{11}|^2-|S_{22}|^2+|\Delta|^2}{2|S_{21}S_{12}|}$$

## Unilateral Gain Circle Calculations:

In RF Amplifier design, one must design an RF Matching Network in order to appropriately match the impedances going into and out of the transistor with the impedances of the input and output SMA connectors. These SMA connectors have a purely resistive impedance of 50  $\Omega$ , therefore to properly match our input and output impedances and simplify our design, we will use gain circles to determine when the transistor source and load impedances will be purely resistive and use a quarter wavelength transmission line to match these resistive impedances to the desired 50  $\Omega$  of the connectors. Gain circles work by centering around the point of maximum gain, with each circle sacrificing 1 dB from the maximum gain point. The objective is to determine how much gain needs to be sacrificed from the maximum in order to achieve a value that crosses the x-axis and is therefore purely resistive at that point.

Under the assumption of Unilateral Gain, the center point of the Source Gain circle is given by the formula

$$C_{GS} = \frac{g_s * \Gamma_s}{1 - |\Gamma_s|^2 (1 - g_s)}$$

And the radius of the Source Gain circle is given by

$$r_{GS} = \frac{(1 - |\Gamma_s|^2) \sqrt{1 - g_s}}{1 - |\Gamma_s|^2 (1 - g_s)}$$

The value of  $g_s$  is determined by the equation

$$g_s = \frac{G_s}{1 - |\Gamma_s|^2}$$

$G_s$  is the source gain discussed previously in this section. Similarly, the center of the Load Gain circle is given by

$$C_{GL} = \frac{g_L * \Gamma_L}{1 - |\Gamma_L|^2 (1 - g_L)}$$

and the radius of the Load Gain circle is given by

$$r_{GL} = \frac{(1 - |\Gamma_L|^2) \sqrt{1 - g_L}}{1 - |\Gamma_L|^2 (1 - g_L)}$$

where

$$g_L = \frac{G_L}{1 - |\Gamma_L|^2}$$

$G_L$  is the load gain discussed previously in this section.

Although the maximum values for  $G_s$  and  $G_L$  are calculated from the formulas above, the actual values for  $G_s$  and  $G_L$  that will be used in the design of the amplifier will be lower. The use of gain circles allows the design to sacrifice some of the amplifier gain in order to use purely resistive components in the

biasing network. This is essential as the transistor must be biased at DC, and only purely resistive components can do this.

### Bilateral Amplifier Design:

Unlike the Unilateral gain amplifier design, the Bilateral amplifier design makes no assumptions about the S-parameters. As a result the calculations are more accurate, but also more complex. This means that the source and load reflection coefficients need to be recalculated using simultaneous conjugate matching. In this design, the assumption that  $\Gamma_s^* = S_{11}$  and  $\Gamma_L^* = S_{22}$  no longer applies. If the simultaneous conjugate match doesn't exist, then the device could be unstable. The formulas for the matched source and load reflection coefficients are

$$\Gamma_{MS} = \frac{B_1}{2C_1} - \frac{1}{2} \sqrt{\left(\frac{B_1}{C_1}\right)^2 - 4\left(\frac{C_1^*}{C_1}\right)}$$

Where B1 and C1 are calculated by

$$B_1 = 1 - |S_{22}|^2 - |\Delta|^2 + |S_{11}|^2$$

$$C_1 = S_{11} - S_{22}^* * \Delta$$

The formula for the matching load reflection coefficient is

$$\Gamma_{ML} = \frac{B_2}{2C_2} - \frac{1}{2} \sqrt{\left(\frac{B_2}{C_2}\right)^2 - 4\left(\frac{C_2^*}{C_2}\right)}$$

Where

$$B_2 = 1 - |S_{11}|^2 - |\Delta|^2 + |S_{22}|^2$$

$$C_2 = S_{22} - S_{11}^* * \Delta$$

And the constant  $\Delta$  is defined previously in the discussion of stability. Thus the equations for simultaneous matching are

$$\Gamma_{MS}^* = S_{11} + \frac{S_{12}S_{21}\Gamma_{ML}}{1 - S_{22}\Gamma_{ML}}$$

$$\Gamma_{ML}^* = S_{22} + \frac{S_{12}S_{21}\Gamma_{MS}}{1 - S_{11}\Gamma_{MS}}$$

## Design and Simulation:

The new design for the amplifier will include some adjustments from previous designs in order to squeeze out as much gain as possible. As was done in previous designs, and for comparison with said designs, this amplifier will use the BFP403W transistor. It is, however, worth noting that this transistor is not the ideal transistor for an RF amplifier at WiFi frequencies (2.4GHz) since the transition frequency is 2GHz.

Before biasing the transistor there are several details that must be addressed. First it is important to note that the transistor current gain,  $h_{FE}$ , given in the datasheet is the DC current gain. At high frequencies, the parasitics of the transistor begin to have significant effects on the current gain which cannot be ignored. Specifically the feedback capacitance between the collector and base,  $C_{\mu}$ , becomes very large and begins to decrease the transistor current gain (see Figure 2).

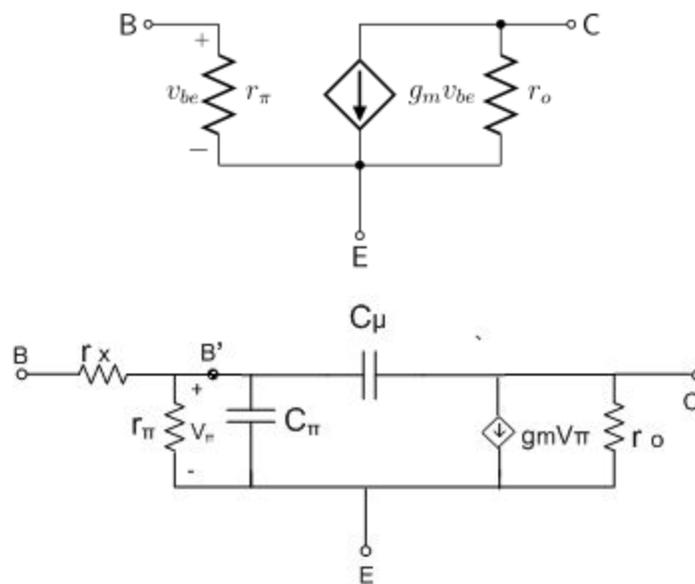


Figure 2: Low and high frequency hybrid-pi models for a BJT.

More concretely the DC current gain,  $h_{FE}$ , is nominally 80, whereas in a previous report the actual current gain at 1GHz was about 40.

Another important not is the maximum gain that the transistor can produce. From Figure 33 (see Appendix A) it can be seen that the Maximum Stable Gain (MSG) at 2.4GHz is approximately 23dB, while

according to Figure 34 (see Appendix A) the Forward Power Gain ( $S_{21}$ ) at 2.4GHz is approximately 14dB. This means that under ideal conditions the amplifier should produce a maximum gain of 14dB.

### Biasing the Transistor:

The first design task will be to bias the transistor. The DC model used for biasing is shown in Figure 3. In order to obtain as much gain as possible, some of the transistor circuit parameters will be slightly modified. The transistor properties were measured with a collector-emitter voltage,  $V_{CE}$ , of 2V, so this will be maintained. The transistor will operate off of a 6V supply voltage instead of the standard 5V supply.

The most significant change will be the collector current,  $I_C$ , which will be 4mA. According to Figure 31 (see Appendix A), increasing  $I_C$  from 3mA to 4mA will decrease the transistor gain,  $h_{fe}$ , from roughly 78 to 75. However increasing  $I_C$  also shifts the unity gain frequency at 2GHz to its maximum value and also increases the Maximum Stable Gain (MSG) by approximately 1dB (see Figure 35). All together this could push the total gain of the amplifier up one or two decibels.

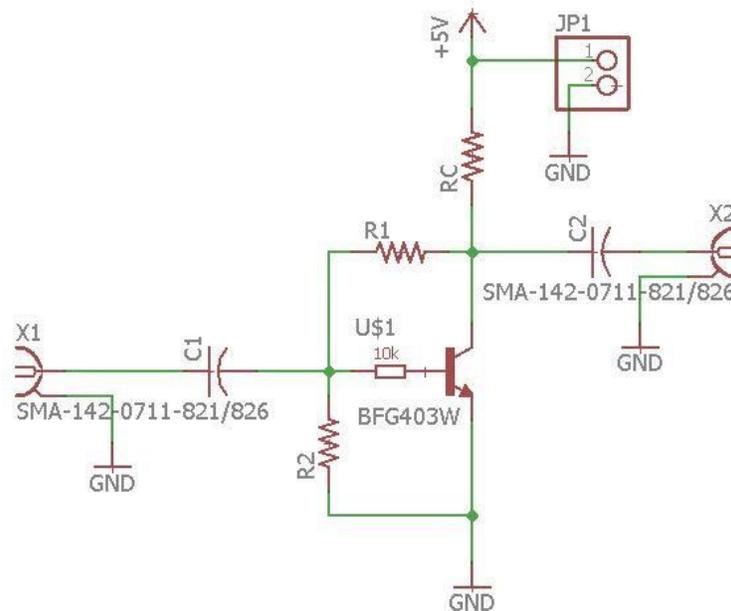


Figure 3: The amplifier under DC conditions

Under the conditions above, and using the equations shown previously, the value of the collector resistor,  $R_C$ , is found to be

$$R_C = \frac{6V-2V}{4mA(1+\frac{6}{40})} = 870\Omega$$

under the assumption that  $h_{FE}$  is 40. Likewise  $R_1$  is found to be

$$R_1 = \frac{2V-0.7V}{6(0.1mA)} = 2.2k\Omega$$

And  $R_2$  is

$$R_2 = \frac{0.7V}{5(0.1mA)} = 1.4k\Omega$$

This combination of resistor values will set the collector-emitter voltage close to 2V.

### Multisim Simulation:

Next the amplifier must be simulated to verify that the biasing is functioning as expected. The simulation was performed in Multisim. Placing the biasing circuit with the calculated component values in multisim yielded the results in Figure 4.

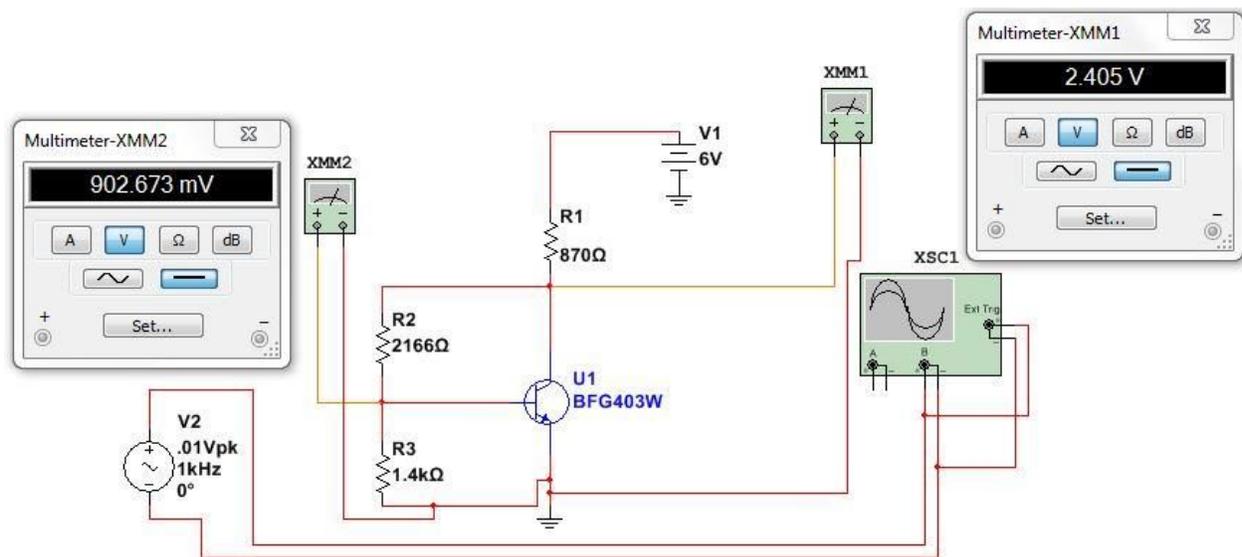


Figure 4: Simulation of the biasing network for the amplifier using computed values.

The closest resistor the ProtoLab had to  $870\Omega$  was  $820\Omega$ , so that resistor value replaced the calculated one. The simulation was redone, and the results are shown in Figure 5. In both of these simulations, the collector-emitter voltage was approximately 2.4V instead of the 2.0V desired. Adjusting the value of the 2.2k resistor to 1.6k fixed this issue and changed the collector-emitter voltage to the desired 2V.

Since the collector-emitter voltage isn't the desired voltage and biasing network won't change from the configuration in Figure 5, the currents need to be recalculated. The simplest current to calculate is the base current  $I_B$ . Rearranging the formula used to calculate  $R_2$  gives the solution

$$I_B = \frac{V_{BE}}{5R_2} = 122\mu A$$

where  $R_2 = 2.2k$ .

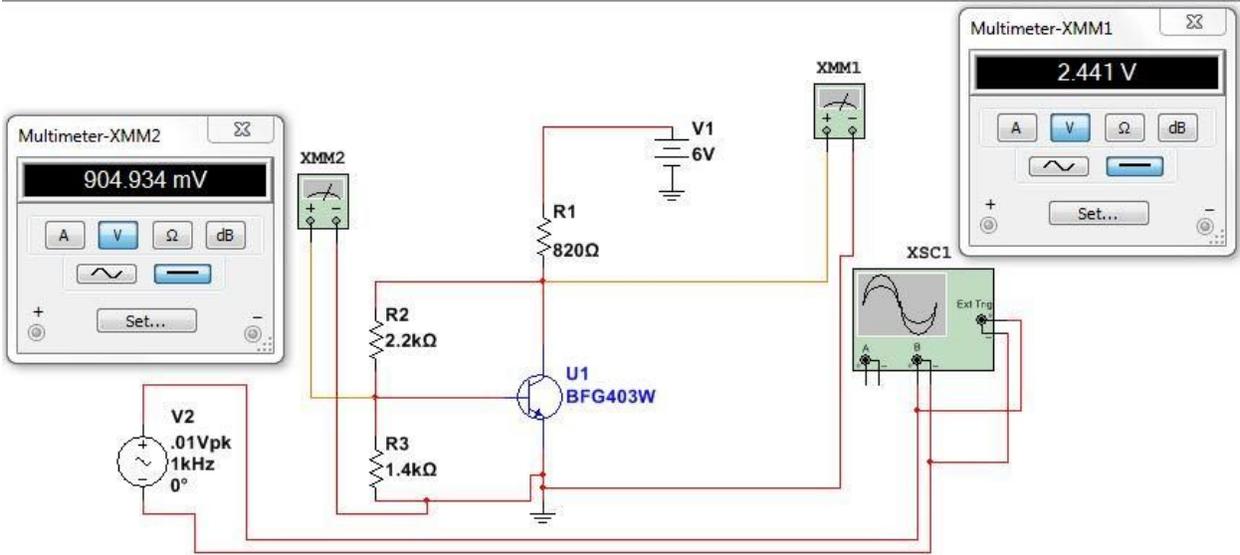


Figure 5: Simulation of the biasing network for the amplifier using actual component values.

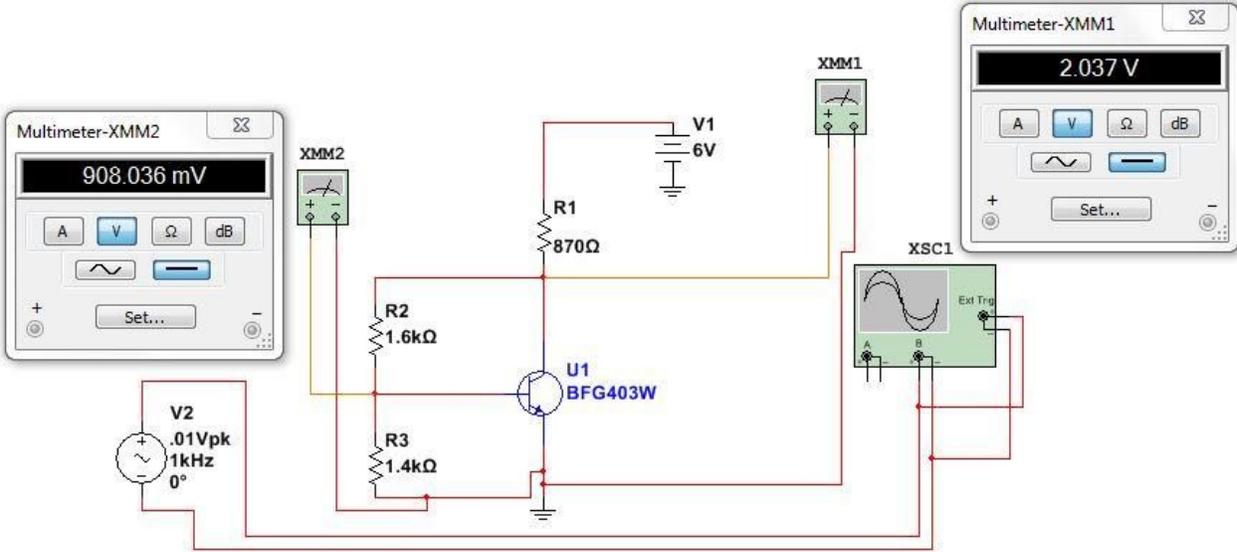


Figure 6: Simulation using a value for  $R_2$  that makes VCE equal to 2V.

Working backwards the formula for  $R_1$  can be rearranged to calculate the collector-emitter voltage  $V_{CE}$  and has the form

$$V_{CE} = V_{BE} + 6R_1I_B = 2.456V$$

Which is in close agreement with the results in Figure 4 and Figure 5. Finally the collector current is calculated using the formula

$$I_c = \frac{V_{cc} - V_{CE}}{R_c} - 6I_B = 3.59mA$$

This is higher than the collector current specified in the datasheet, but not the 4mA desired. This means that the difference in gain from using a higher collector current will not be as noticeable.

### RF Blocking Network:

The RF blocking network is crucial to the amplifier because it prevents the signal from traveling to the power supply and ground plane, ensuring that all of the data passes to the output of the amplifier. The series inductors  $L_1$  and  $L_2$  are normally called RF Chokes (RFC) and serve to isolate the input and output signals from source, while the shunt capacitors  $C_1$  and  $C_2$  short out all the remaining high frequencies (see Figure 7). This amplifier will use distributed components for the RF blocking networks. As such, the values of the series inductors and shunt capacitors will be calculated using transmission line theory.

The series inductors and shunt capacitors will both be quarter wavelength transformers. This is because an inductive transmission line with a length of one quarter of the center frequency behaves like an open circuit for the center frequency and multiples of it, while a capacitive transmission line with a length of one quarter of the center frequency behaves like a short for the frequencies that manage to pass through (see Figure 8). The length of the transmission lines is calculated using the formula

$$\lambda = \frac{v_p}{f} = \frac{c}{f\sqrt{\epsilon_{eff}}}$$

Where

$$\epsilon_{eff} = \frac{\epsilon_r + 1}{2} + \frac{\epsilon_r - 1}{2} \left(1 + \frac{12h}{w}\right)^{-\frac{1}{2}}, \quad \frac{w}{h} > 1$$

$$\epsilon_{eff} = \frac{\epsilon_r + 1}{2} + \frac{\epsilon_r - 1}{2} \left[\left(1 + \frac{12h}{w}\right)^{-\frac{1}{2}} + (0.04)\left(1 - \frac{w}{h}\right)^2\right], \quad \frac{w}{h} < 1$$

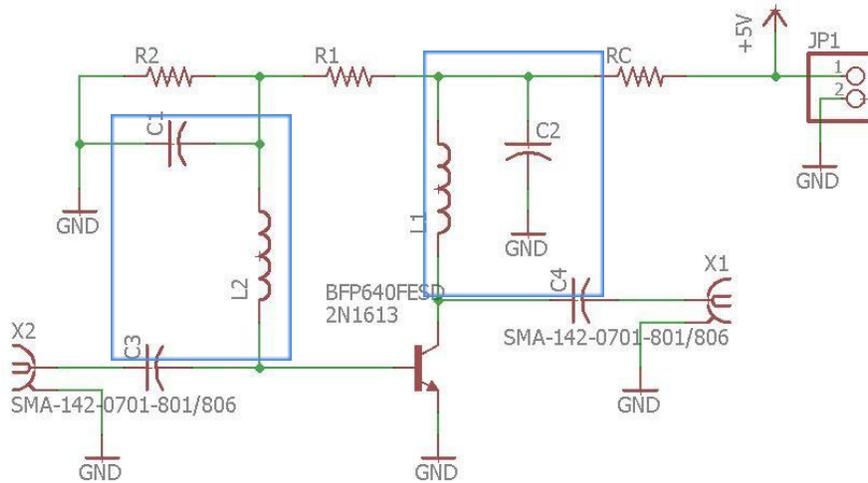


Figure 7: of the RF amplifier. RF blocking networks are outlined in blue.

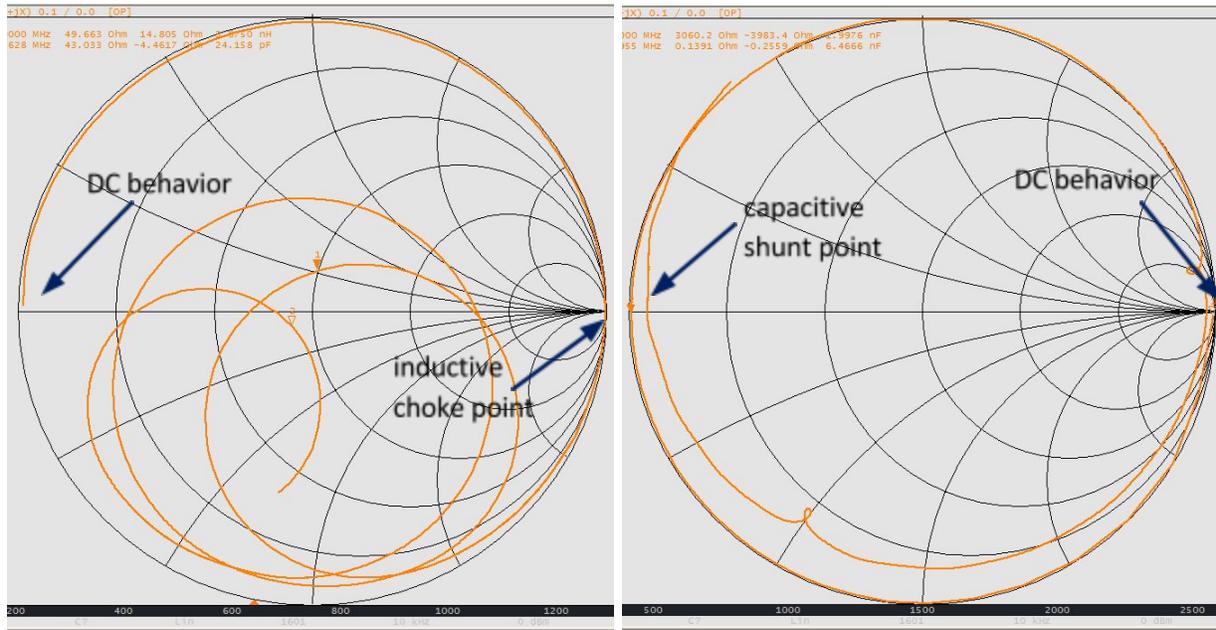


Figure 8: Frequency response of a 530nH inductor and a 220pF capacitor.

and  $\epsilon_r$  for FR-4 is 4.6. Assuming a lossless transmission line model, the characteristic impedance of the transmission lines is given by the equations

$$Z_0 = \frac{Z_f}{\sqrt{\epsilon_{eff} \left( 1.393 + \frac{w}{h} + \frac{2}{3} \ln \left( \frac{w}{h} + 1.444 \right) \right)}}, \quad \frac{w}{h} > 1$$

$$Z_0 = \frac{Z_f}{2\pi\sqrt{\epsilon_{eff}}} \ln\left(\frac{8h}{w} + \frac{w}{4h}\right), \quad \frac{w}{h} < 1$$

Where  $Z_f$  is the impedance of free space,  $\epsilon_{eff}$  is the effective dielectric constant of the PCB (in this case it is FR-4),  $h$  is the thickness of the PCB, and  $w$  is the width of the trace. The impedance of free space is

$$Z_f = \sqrt{\frac{\mu_0}{\epsilon_0}} = 376.8\Omega$$

Calculating the length of the inductive transmission line is straight forward. Since the inductance of a microstrip line increases as the width decreases, the trace should be as narrow as possible. In practice this is limited by the fabrication limits of the foundry. This amplifier will be created by the LSU ProtoLab, and the minimum trace width it has successfully milled was 6mils. However, the machine itself is only guaranteed to work for widths of approximately 12mils, so the inductive lines will be 8mils wide.

Using this width and the fact that the PCB will be approximately 60mils thick, the effective dielectric constant is calculated to be  $\epsilon_{eff} = 3.043$ . From this the wavelength is calculated to be

$$\lambda = \frac{c}{f\sqrt{\epsilon_{eff}}} = 71.66mm$$

Thus the length of the quarter wavelength RF choke is

$$l = 17.9mm = 705.3mils$$

And the characteristic impedance is

$$Z_0 = \frac{Z_f}{2\pi\sqrt{\epsilon_{eff}}} \ln\left(\frac{8h}{w} + \frac{w}{4h}\right) = 140.78\Omega$$

The capacitive transmission line is slightly more difficult. The purpose of the shunt capacitor is to short out all the frequencies that make it through the inductor. Because of this the series inductor and shunt capacitor function as a low pass filter, where the inductive transmission line functions as a resistor with a resistance equal to the characteristic impedance of the inductive transmission line,  $Z_0 = 140.78\Omega$ .

Under this assumption, the corner frequency for the low pass filter is given by the formula

$$f_0 = \frac{1}{2\pi RC}$$

Choosing the corner frequency to be 240MHz, capacitance is therefore

$$C = \frac{1}{2\pi f_0 R} = 4.7pF$$

Using the equation for an ideal capacitor, the impedance of the trace is

$$Z_0 = \frac{1}{2\pi f_0 C} = 14.1\Omega$$

Table 2-20 in the RF textbook gives the impedance as a function of the width-to-height ratio. Using the calculated value of  $Z_0$  and the dielectric constant of FR-4,  $\epsilon_r = 4.6$ , the width-to-height ratio was estimated to be approximately 9. Since the width-to-height ratio is greater than 2, the exact value of the ratio is calculated from the formula

$$\frac{w}{h} = \frac{2}{\pi} \left\{ B - 1 - \ln(2B - 1) + \frac{\epsilon_r - 1}{2\epsilon_r} \left[ \ln(B - 1) + 0.39 - \frac{0.61}{\epsilon_r} \right] \right\} = 10.043$$

Where  $B = \frac{Z_f \pi}{2Z_0 \sqrt{\epsilon_r}} = 19.16$ . Since the thickness of the PCB is set to 60mils, the width is thus 600mils.

This means the effective dielectric constant is  $\epsilon_r = 4.015$ , and thus the wavelength is

$$\lambda = \frac{c}{f \sqrt{\epsilon_{eff}}} = 62.38 \text{ mm} = 2456.0 \text{ mils}$$

This means that the length of the shunt capacitor is

$$l = 614 \text{ mils}$$

As a check for both the calculations and the Sonnet simulations to follow, the characteristic impedance of the capacitive transmission line is

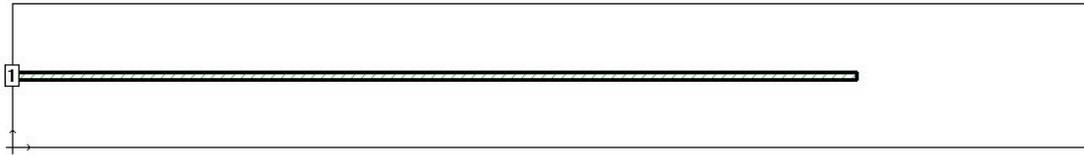
$$Z_0 = \frac{Z_f}{\sqrt{\epsilon_{eff}} \left( 1.393 + \frac{w}{h} + \frac{2}{3} \ln \left( \frac{w}{h} + 1.444 \right) \right)} = 14.395 \Omega$$

which is close enough to the desired  $14.1 \Omega$ .

It is worth noting that Table 2-1 in the textbook by Ludwig and Bogdanov lists formulas for calculating the capacitance and inductance per unit length of a parallel plate transmission line. These formulas do not apply to the calculations for the RF blocking network for the following reasons. The inductive trace is very narrow, so the electric field between the copper trace and the ground plane is dominated by fringe effects, thus violating one of the important assumptions for a parallel plate transmission line. The formula for the capacitance per unit length is fairly accurate for very wide traces, but the inductance of these traces is not. Thus these formulas should only be used as guidelines and not relied upon.

### RF Blocking Network Sonnet Simulations:

In order to verify that the calculations for the RF blocking network are correct, the transmission lines must be simulated. The simulations were performed in Sonnet Suite. The first transmission line to be simulated was the inductive line. The dimensions of the simulated transmission line were  $l=705$  mils and  $w=6$  mils. The simulation was performed with cell sizes of 0.25mil by 0.25mils for greater accuracy. The results are shown in Figure 9. Strictly speaking, since the inductive transmission line is a series component, the simulation should have included a port on both ends of the transmission line. However, when the simulation included a second port, there was no noticeable change in the simulation results.



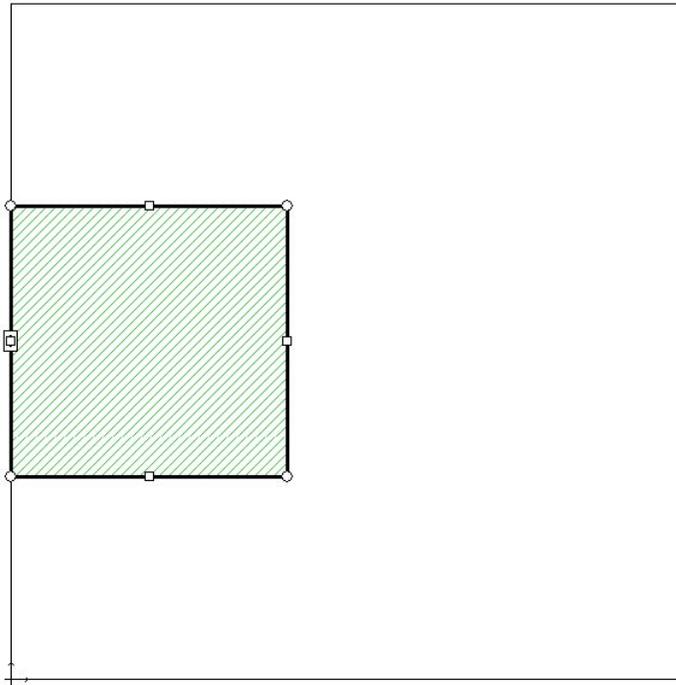
```

Frequency: 2.425 GHz
Frequency completed Sun Dec 03 20:44:35 2017.
De-embedded S-Parameters, 50.0 Ohm Port Terminations.
Magnitude/Angle, Touchstone Format.
2.42500000 0.895447 -176.6
!< P1 F=2.425 Eeff=(2.86620846 -0.0733835) Z0=(138.045975 0.5247687) R=28.962117 C=0.03675192

```

Figure 9: Results of the Sonnet simulation for the inductive transmission line.

Note that the previously calculated characteristic impedance and  $\epsilon_r$  for the inductive transmission line differ from the simulated impedance and  $\epsilon_r$  in Figure 9. This doesn't matter because it is a quarter wavelength transmission line. If the calculated characteristic impedance for the transmission line is plotted on a Smith chart normalized to  $141 \Omega$ , then it is clear that the calculated characteristic impedance is still going from a very low impedance to a very high impedance, and ultimately all the transmission line needs to do is block frequencies in the passband. Thus this is good enough since we don't need to block everything.



```

Frequency: 2.425 GHZ
Frequency completed Sun Dec 03 20:57:39 2017.
De-embedded S-Parameters. 50.0 Ohm Port Terminations.
Magnitude/Angle. Touchstone Format.
2.42500000 0.990987 177.91
!< P1 F=2.425 Eeff=(3.8742294 -0.0806604) Z0=(14.1302493 0.13091787) R=0.39434312 C=2.34192843

```

Figure 10: Results of the Sonnet simulation for the capacitive transmission line.

The capacitive transmission line simulation was less troublesome. The calculated dimensions for this transmission line were  $l=614$  mils and  $w=600$  mils, and the cell size was again 0.25 mils by 0.25 mils. As seen in Figure 10, the simulation matched the calculated values very well.

### Stability Calculations:

In order to do any high frequency design, it is necessary to have the Scattering Parameters (S-parameters). In practice, determining the S-parameters is called “parameter extraction” and, while possible, it is exceedingly difficult. One way to calculate the S-parameters for a device is to use the hybrid- $\pi$  parameters (h-parameters) and then convert them to the S-parameters with Matlab. This method allows the S-parameters to be directly calculated from values given on the datasheet. For this amplifier, this method is unnecessary as the table of S-parameters for the BFG403 can be taken from the RF textbook written by White that was used in the previous report. The table of S-parameters for a wide range of frequencies is shown below in Figure 11.

BFG403W	$V_{BE}=0.8791V$		$V_{CE}=2.0V$		$I_B=43.08\mu A$		$I_C=3.0mA$	
	S11		S21		S12		S22	
Freq (GHz)	Mag	Angle	Mag	Angle	Mag	Angle	Mag	Angle
1	0.787	-29.852	6.441	138.225	0.012	67.938	0.947	-20.577
1.2	0.754	-35.069	6.204	130.774	0.014	64.926	0.931	-24.257
1.4	0.718	-39.992	5.959	123.705	0.016	61.999	0.914	-27.815
1.6	0.68	-44.782	5.714	116.841	0.017	59.838	0.897	-31.212
1.8	0.642	-49.349	5.461	110.315	0.019	57.764	0.88	-34.551
2	0.609	-53.651	5.223	104.247	0.02	56.296	0.863	-37.812
2.2	0.571	-57.579	4.984	98.269	0.022	55.082	0.845	-40.963
2.4	0.539	-61.74	4.775	92.619	0.023	54.271	0.829	-44.054
2.6	0.505	-65.624	4.561	87.157	0.024	53.509	0.812	-47.098
2.8	0.481	-70.015	4.392	82.018	0.026	52.792	0.796	-50.106
3	0.456	-74.228	4.234	77.005	0.027	52.22	0.78	-53.098

Figure 11: Table of magnitude and phase of S-parameters for the BFG403W.

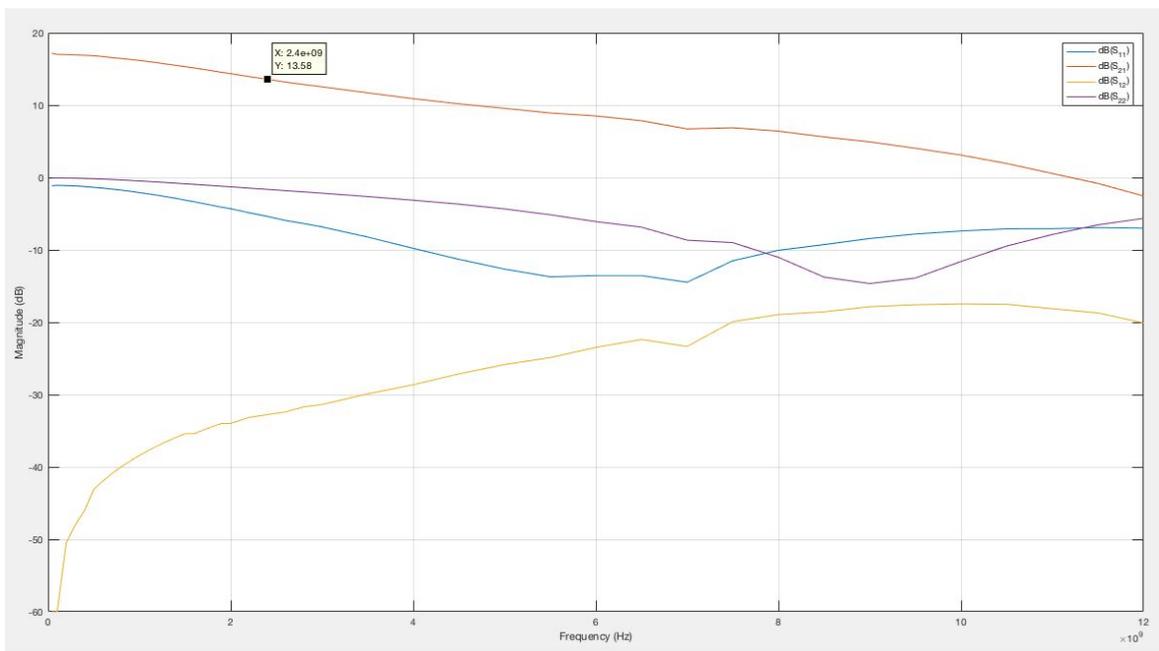


Figure 12: Matlab plot of  $S_{11}$  (blue),  $S_{12}$  (yellow),  $S_{21}$  (red), and  $S_{22}$  (purple) in dB as a function of frequency.

Using the values for the S-parameters at 2.4GHz, it is then possible to determine the values of the input and output stability circles. The Matlab code [Stability\\_Circles.m](#), which contains the previously derived equations for the stability circle calculations, is used to obtain the following plot shown in figure 13.

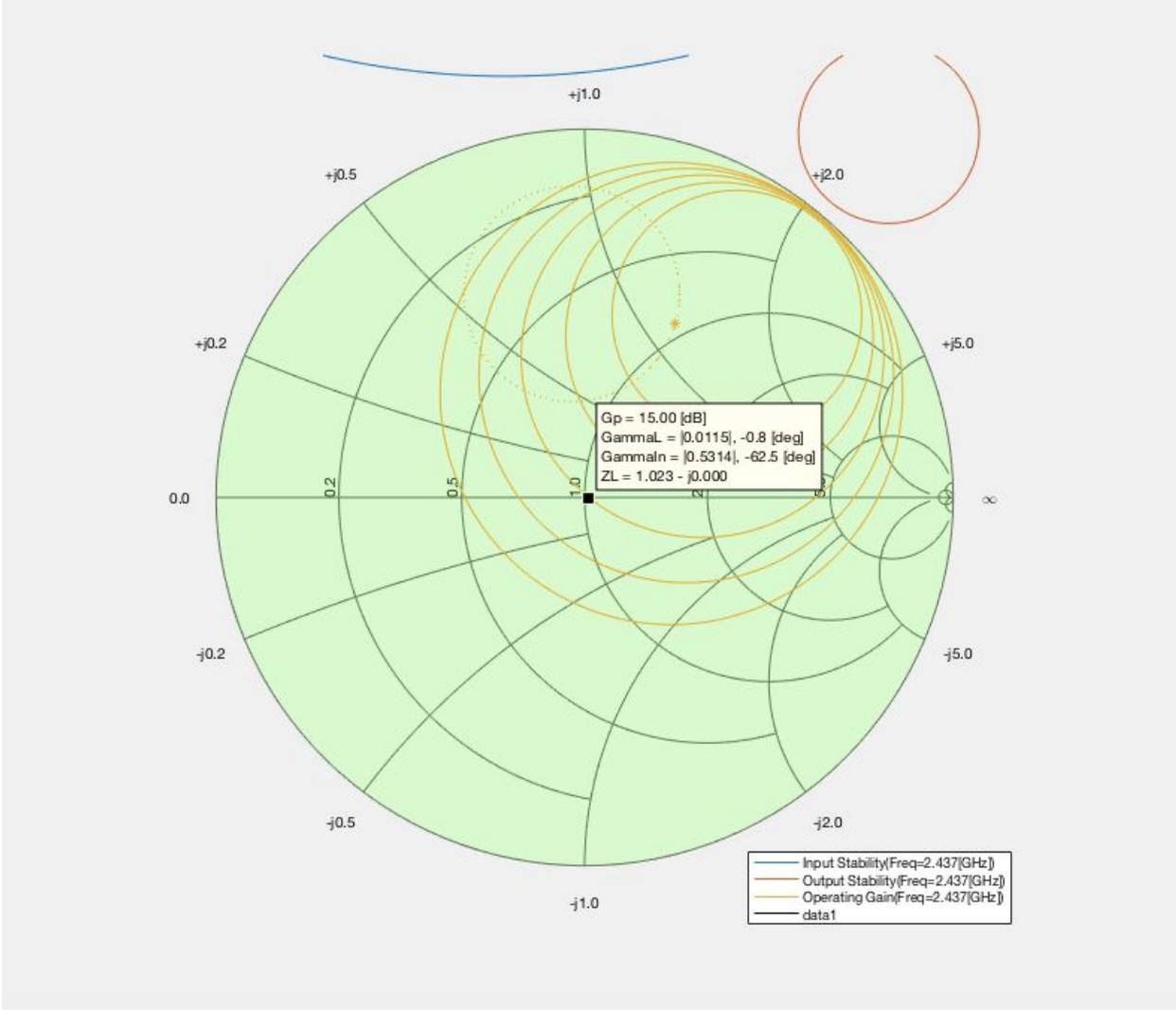


Figure 13: Smith chart showing stability circles for the transistor at 2.4GHz. The stability circles are shown in red and blue.

From these results we can see that at the desired frequency range, the stability circles are located completely off the Smith Chart and therefore allow the transistor to operate successfully over our desired bandwidth at any impedance values without being unstable.

### Gain Calculations:

Using the formulas derived in the previous section for Unilateral Gain Circles, we compute the following values for K, G<sub>s</sub>, G<sub>o</sub>, G<sub>i</sub>:

$$G_s = 1.4095$$

$$G_o = 22.8006$$

$$G_i = 3.195$$

$$K = 1.1982$$

when each of the gain values is converted into dB, we then can calculate the overall gain in dB as

$$G_{tu} = 20.119 \text{ dB}$$

The K-factor > 1 confirms our calculations that our amplifier design will be unconditionally stable over the desired frequency range. Upon running the matlab code [Input\\_OutputGainCircles.m](#) we generated the following plots for Input and Output Gain Circles, where each circle going outwards from the innermost circle corresponds to -1 dB of gain sacrificed from the maximum gain values calculated.

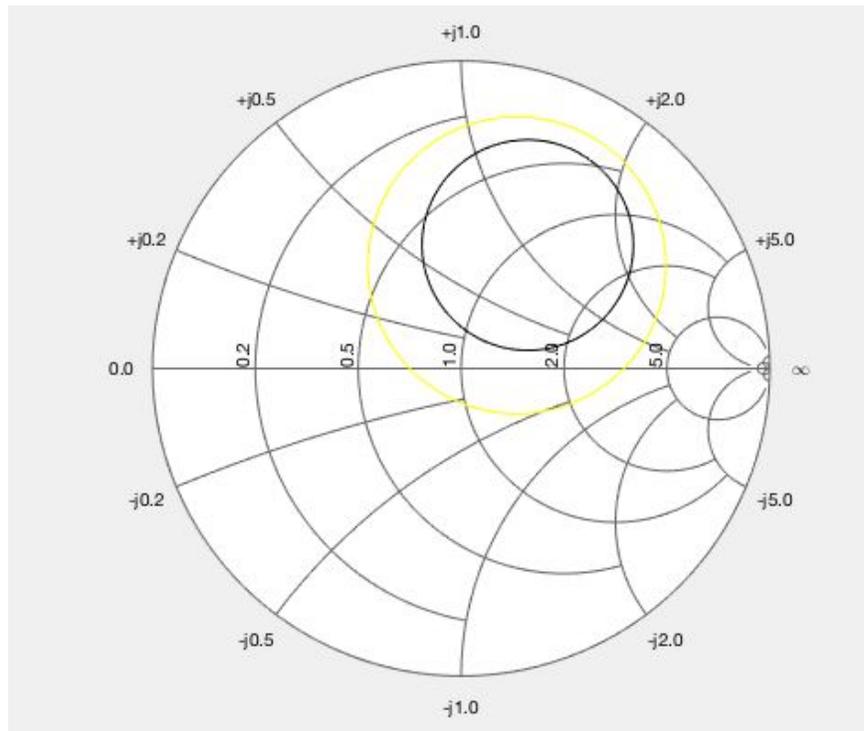


Figure 14: Smith chart of the Source Gain circles showing that 2dB of gain must be sacrificed to bias the transistor.

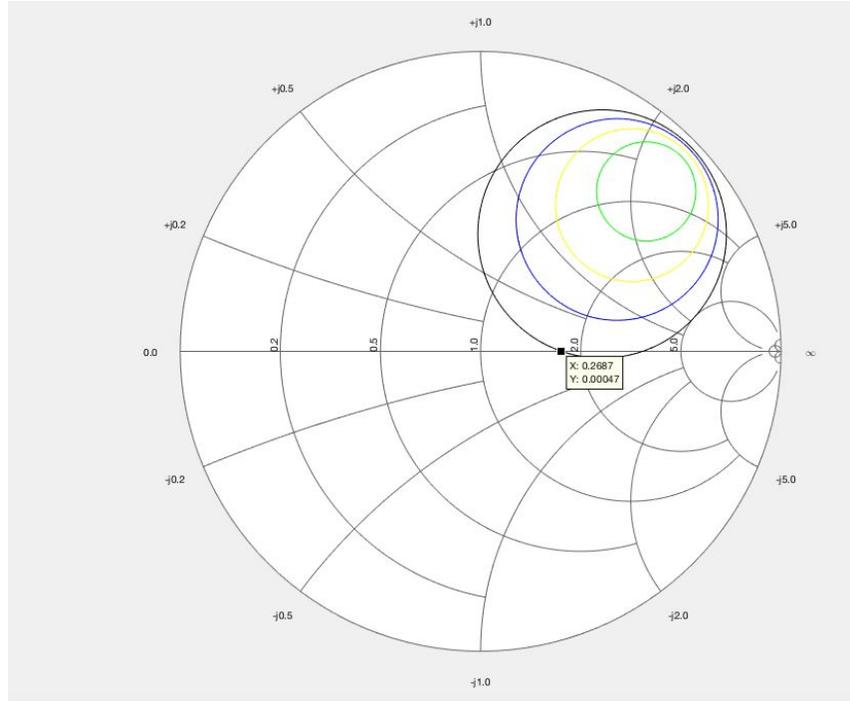


Figure 15: Smith chart of the Load Gain circles showing that 4dB of gain must be sacrificed to bias the transistor.

From these gain circle simulations, we can observe that -2dB of gain is lost to cross the real axis for the input gain circles and -4dB of gain is lost to cross the real axis for the output gain circles. From the Smith Chart plots above we determined the input impedance and output impedances to be

$$Z_{in} = 37.75 \Omega \quad Z_{out} = 83.3 \Omega$$

We now use these new impedances to calculate the dimensions for the Matching Network quarter wavelength transmission lines.

### RF Matching Network:

In order to properly match the impedances from the transistor to the SMA connectors, we will design a quarter wavelength transmission line for the input and output impedances of the transistor to match the impedance of the connectors.

The length of the transmission lines will depend on the wavelength  $\lambda$  which depends on  $\epsilon_{eff}$ , so first we calculate the width using the following formula, where  $h$  is the FR-4 Board depth and  $\epsilon_r$  is the FR-4 permittivity = 4.6

$$h = 60 \text{ mils} = 1.524 \text{ mm} \quad \mu_0 = 4\pi * 10^{-7} \quad \epsilon_0 = 8.85 * 10^{-12} \quad \epsilon_r = 4.6$$

$$W = \frac{h}{Z_{line}} \sqrt{\frac{u_o}{\epsilon_o \epsilon_r}}$$

For the input transmission line,  $w_1 = 133.97$  mils. For the output transmission line,  $w_2 = 69.33$  mils.

We can then calculate for  $\epsilon_{eff}$  and the quarter wavelength  $\lambda/4$  using the formulas discussed previously in this report and attain the following results:

$$\epsilon_{eff\ in} = 3.228 \quad \epsilon_{eff\ out} = 3.0154 \quad \frac{\lambda}{4}\ in = 696.04\ \text{mils} \quad \frac{\lambda}{4}\ out = 702.23\ \text{mils}$$

we can calculate what the desired impedance of the transmission lines will be by using the 50  $\Omega$  impedance of the SMA connectors ( $Z_o$ ) and the  $Z_{in}$  and  $Z_{out}$  impedances calculated previously.

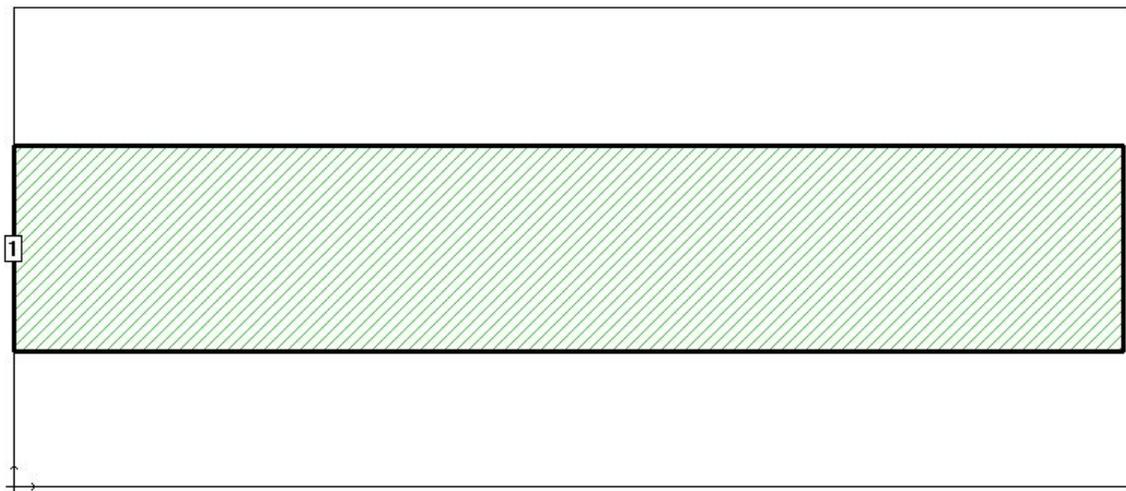
$$Z_{line\ in} = \frac{Z_f}{\sqrt{\epsilon_{eff}}(1.393 + \frac{w}{h} + \frac{2}{3} \ln(\frac{w}{h} + 1.444))} = 43.3\ \Omega$$

$$Z_{line\ out} = \frac{Z_f}{\sqrt{\epsilon_{eff}}(1.393 + \frac{w}{h} + \frac{2}{3} \ln(\frac{w}{h} + 1.444))} = 63.95\ \Omega$$

### RF Matching Networks Sonnet Simulations:

With the calculated widths, lengths and impedances we run Sonnet Simulations of each transmission line to determine the accuracy of our calculations and any changes needed to meet the desired impedance.

The final input matching network dimensions were  $l = 695$  mils and  $w = 130$  mils and produced the following simulation results.



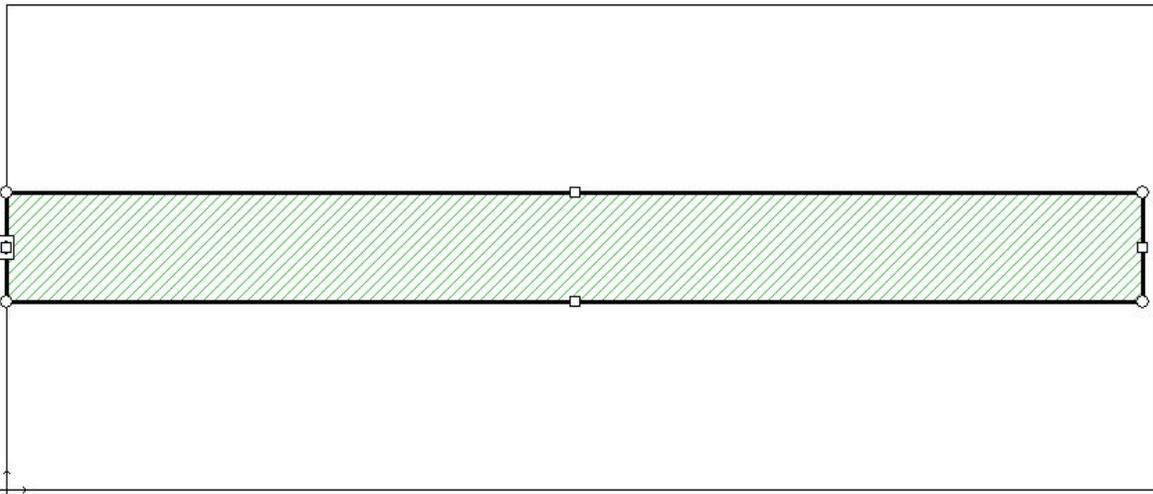
```

Frequency: 2.45 GHZ
Frequency completed Sun Dec 03 18:56:57 2017.
De-embedded S-Parameters. 50.0 Ohm Port Terminations.
Magnitude/Angle. Touchstone Format.
2.45000000 0.974088 172.38
!< P1 F=2.45 Eeff=(3.15328219 -0.0616327) Z0=(43.3184313 0.3482339) R=1.56781787 C=0.65377697

```

Figure 16: Results of Sonnet simulation for input matching network transmission line.

For the final output matching network, the dimensions were  $l = 702$  mils and  $w = 68$  mils and produced the following simulation results.



```

Frequency: 2.425 GHZ
Frequency completed Sun Dec 03 20:33:38 2017.
De-embedded S-Parameters. 50.0 Ohm Port Terminations.
Magnitude/Angle. Touchstone Format.
2.42500000 0.961277 162.34
!< P1 F=2.425 Eeff=(3.0664639 -0.0607315) Z0=(64.1018966 0.48800425) R=2.71151085 C=0.38252567

```

Figure 17: Results of Sonnet simulation for output matching network transmission line.

The sonnet simulation results compared to the calculated results are shown below:

	Impedance (ohms)	Width (mils)	Length (mils)	Effective
<b>Calculated Input</b>	43.12	133.97	696.04	3.228
<b>Sonnet Results</b>	43.318	130	695	3.153
<b>Calculated Output</b>	63.95	69.33	702.23	3.0154

<b>Sonnet Results</b>	64.1018	68	702	3.066
-----------------------	---------	----	-----	-------

These calculations show our final values to be acceptable in achieving the design specifications and are ready to be implemented in the PCB design. The final calculations necessary before the PCB design begins is to determine the values of the decoupling capacitors, which will be explained shortly.

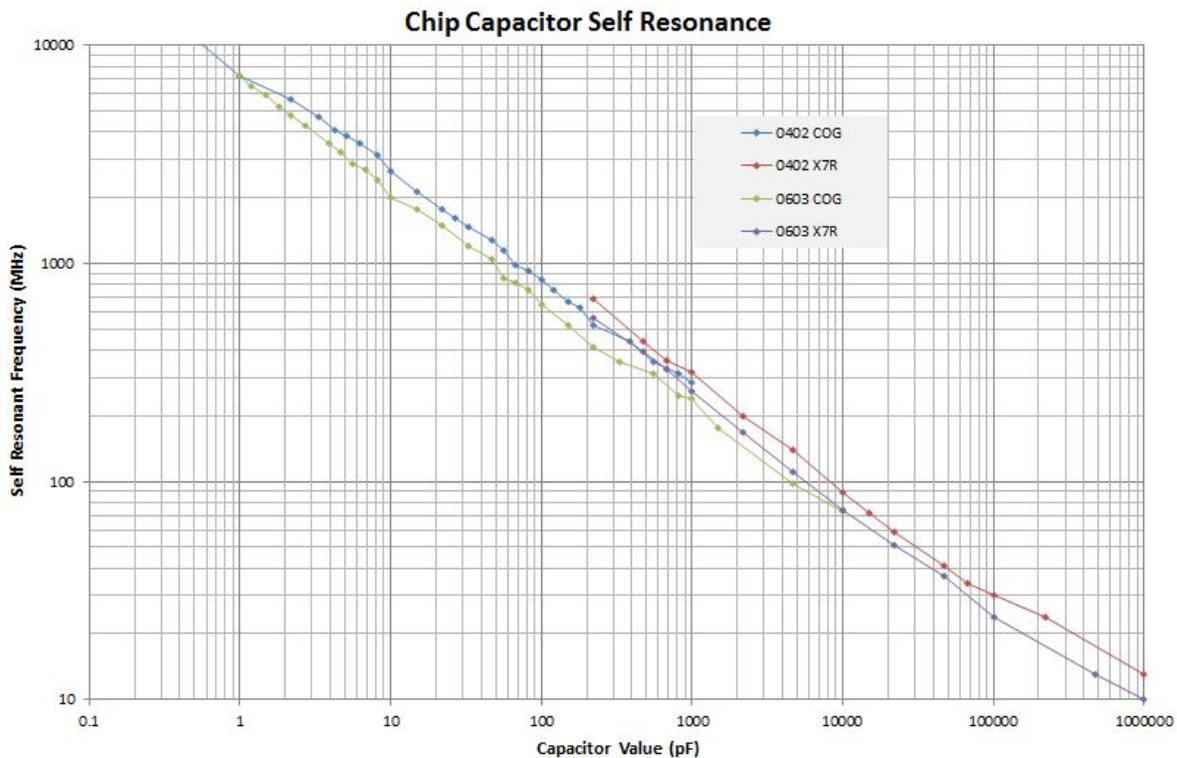


Figure 18: Plot of capacitor self resonance vs. capacitor value.

The decoupling capacitors only need to have a low impedance at the amplifier's operating frequency. The self resonance of the capacitor will lower the impedance further. From Figure 18 it can be seen that the self resonance of the decoupling capacitors will occur at the operating frequency of the amplifier when the capacitors have a value of approximately 10pF. Strictly speaking this level of attention to the decoupling capacitors is unnecessary since the amplifier will work fine as long as the impedance of the capacitors is below roughly 5-10  $\Omega$  at the operating frequency.

Once the transmission lines were fully simulated, everything was ready to put onto a PCB. The PCB was designed in Eagle PCB. The board is shown in Figure 19.

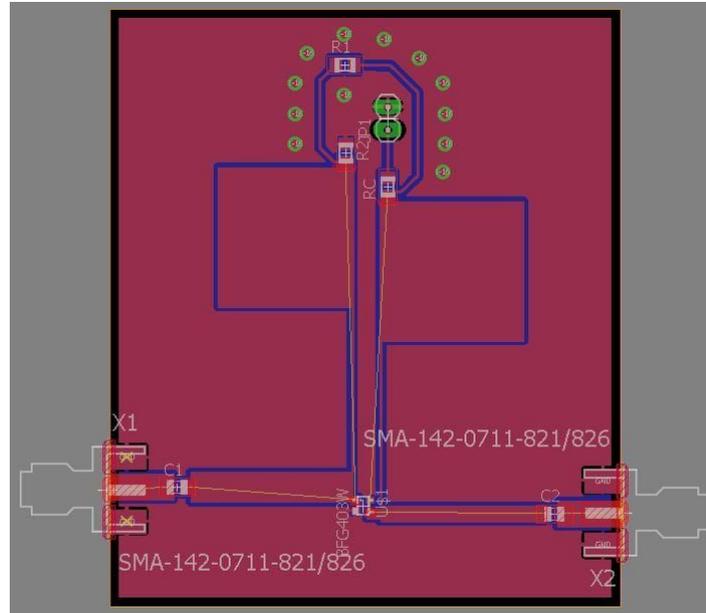


Figure 19: The PCB layout of the amplifier in Eagle PCB.

## Testing and Analysis:

### **Bias Network:**

The first test for any transistor amplifier is the biasing. With the supply voltage set to 6V, the base-emitter voltage was 0.851V and the collector-emitter voltage was 3.3V. This is almost 1V higher than the simulated collector-emitter voltage, but this isn't a bad thing. This means that that the amplifier can make almost full use of the entire voltage range from 0-6V, which is what is normally desired when designing any kind of amplifier. After the biasing test, it was time to test the frequency response of the amplifier.

### **RF response:**

In order to safely test the amplifier circuit, two 20dB attenuators were connected to the PCB, one to the input and one to the output (see Figure 20). This was to prevent any unforeseen gain spikes from damaging the equipment. The first test was the forward voltage gain  $S_{21}$  with the designed supply voltage of 6V. The amplifier had a forward voltage gain of approximately 5.5dB at the center frequency. There was also a ripple of slightly over 0.2dB over the entire passband. In all amplifier frequency sweeps starting with Figure 21 marker 1 is the center frequency of the passband, and markers 2 and 3 mark the lower and upper edges of the passband.

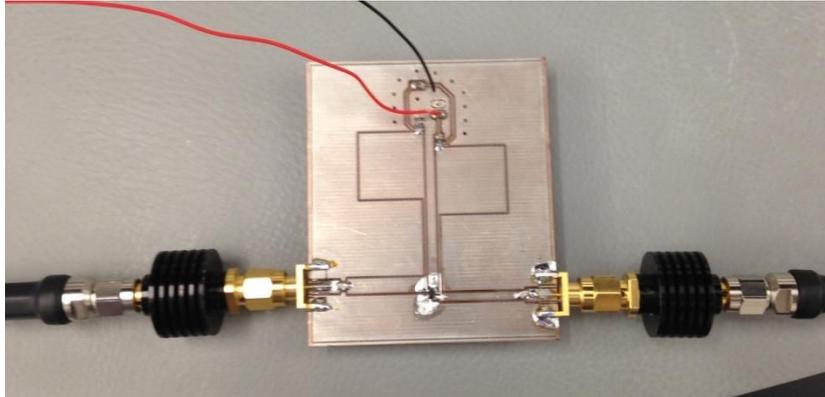


Figure 20: Amplifier with 20dB attenuators.

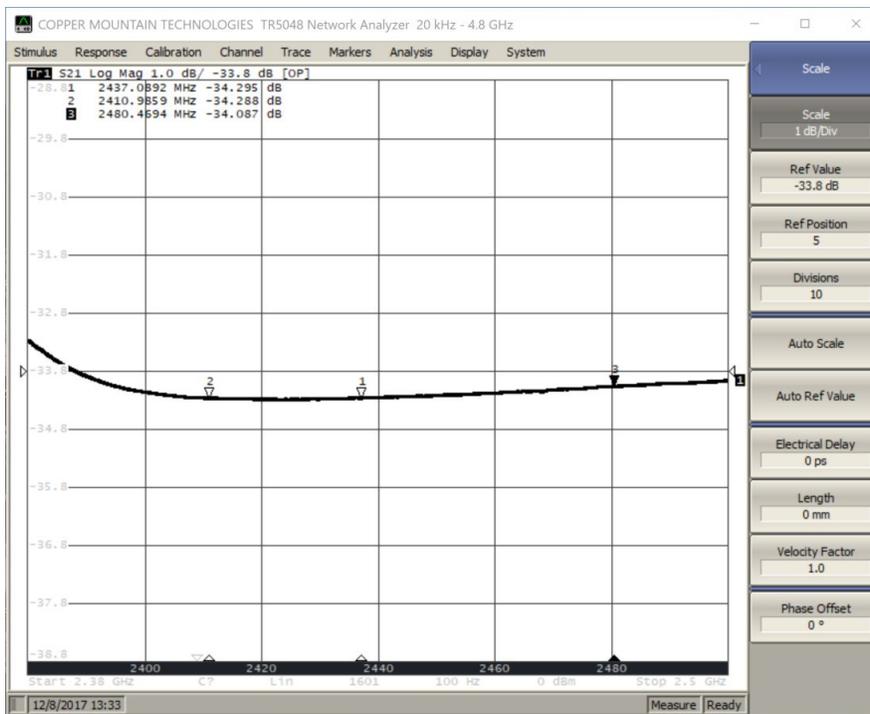


Figure 21: Amplifier forward gain ( $S_{21}$ ) pass-band response for  $V_{CC}=6V$ .

After the initial measurements came the tuning. By varying the supply voltage, the gain of the amplifier also changed noticeably. After adjusting the supply voltage the optimum supply was found to

be 5.44V, and the gain increased to 6.5dB. The voltage ripple also increased to slightly under 0.3dB over the passband.

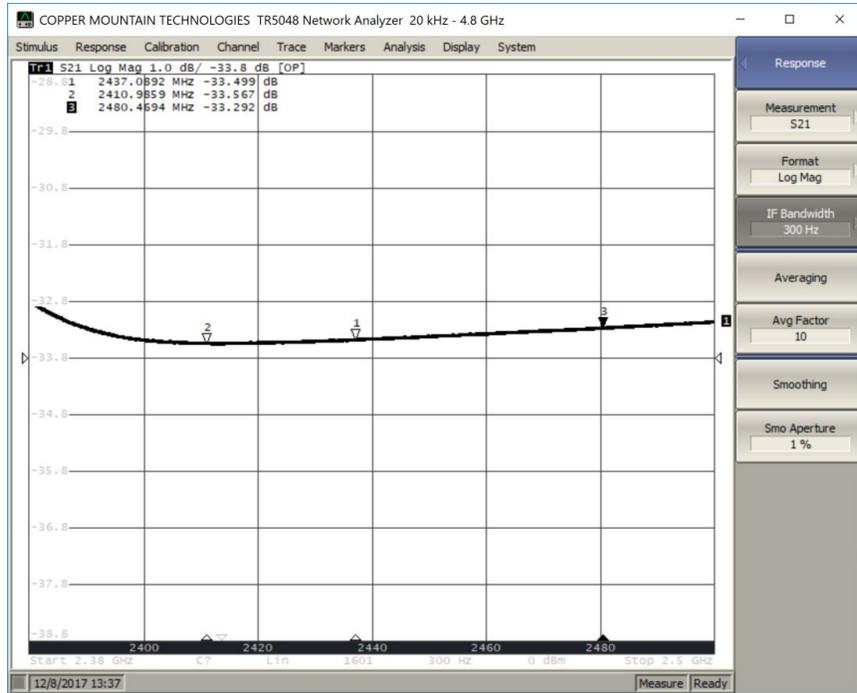


Figure 22: Amplifier forward gain ( $S_{21}$ ) pass-band response for  $V_{CC}=5.44V$

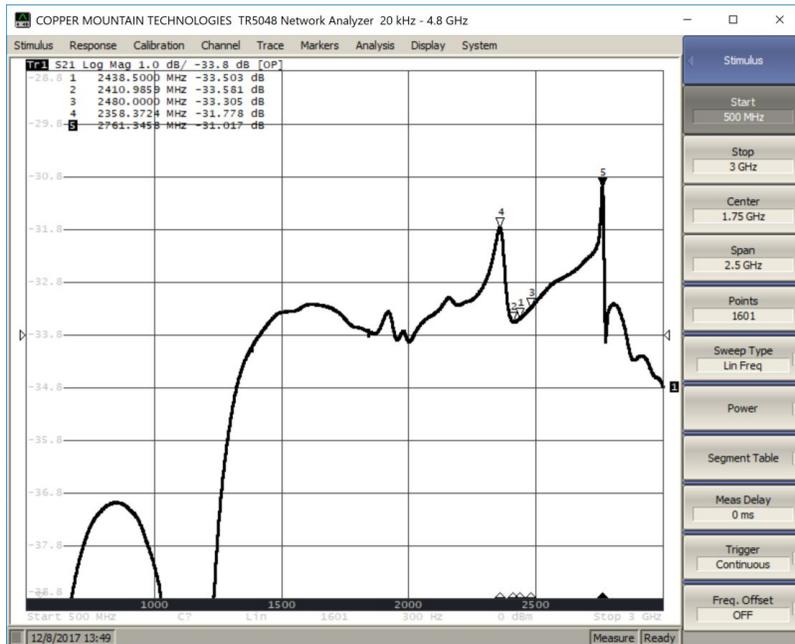


Figure 23: Wideband response of amplifier.

The next measurement was a check to view the wideband response of the amplifier (see Figure 24). With the exception of the peaks at 2.36GHz and 2.76GHz (markers 4 and 5) and the immediate surrounding regions, the amplifier had less than 1.5dB of variation over the range of 1.5GHz to 2.8GHz. The two spikes in gain will be discussed later in the report. The high frequency roll-off is also worth noticing. The BFG403W is a wideband transistor, so the roll-off that starts around 2.8GHz is due to the design of the amplifier. The exact portion of the design that causes this roll-off is unknown.

Another very important quantity to measure is the reverse gain  $S_{12}$  of the amplifier. There is a chance that some of the input signal could be reflected back from the output of the amplifier and then be further amplified and fed to the input device. This would almost definitely destroy the device which is not desired. The measurements of  $S_{12}$  are shown in Figure 24.

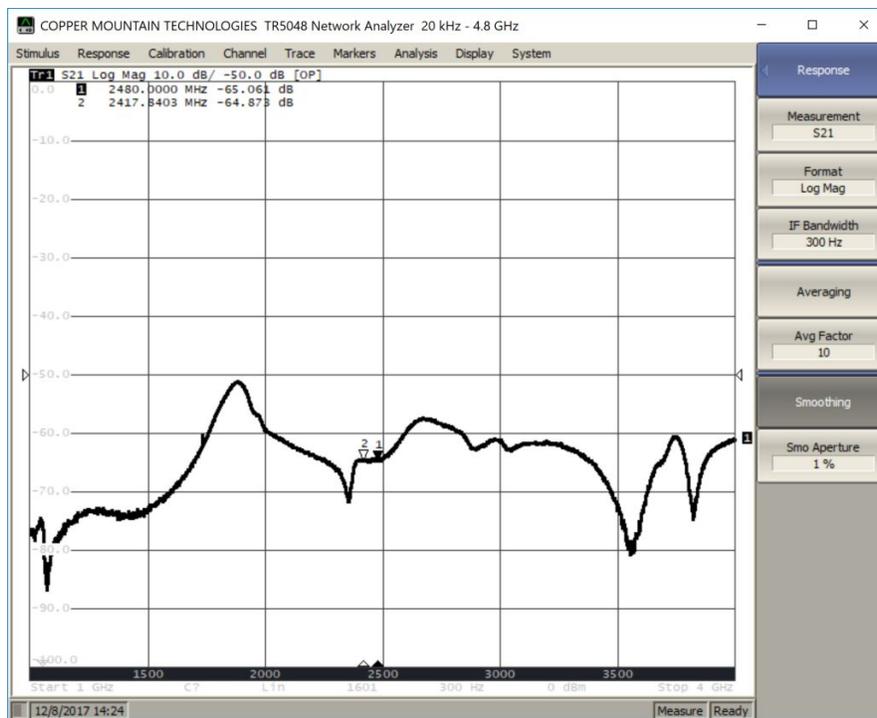


Figure 24: Reverse gain ( $S_{12}$ ) of amplifier.

The measurement of group delay and phase are also very important to the design of a useful amplifier. For obvious reasons, the group delay should be kept as short as possible. As seen in previous labs, it is also crucial that the phase be linear over the passband since a perfectly linear phase means all the information is arriving at the destination at the same time. If the group delay is sufficiently nonlinear, then information could arrive at the destination in the incorrect order. The plots of group delay and phase are shown in Figure 25 and Figure 26.

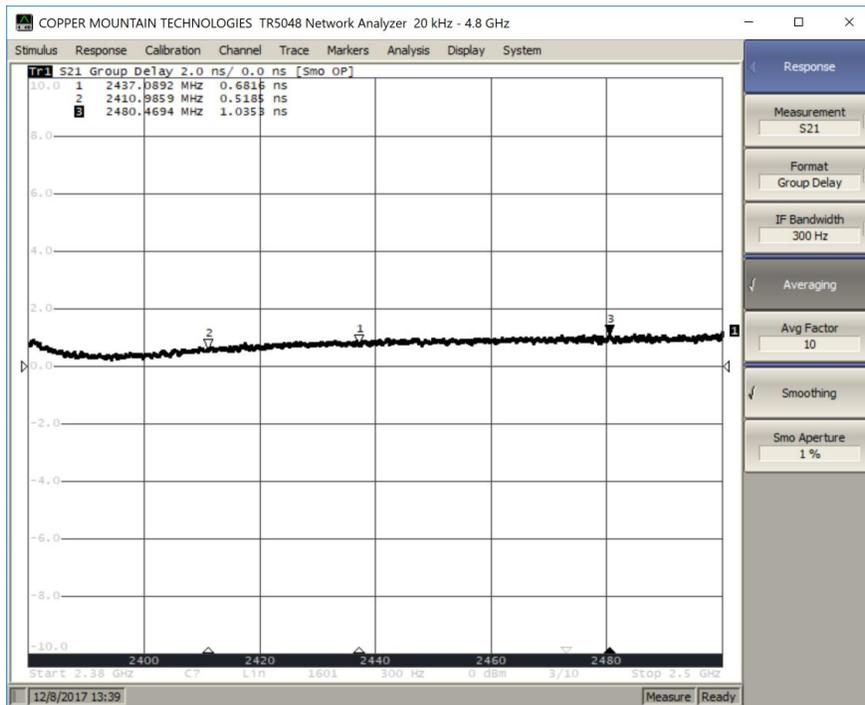


Figure 25: Group delay of amplifier.

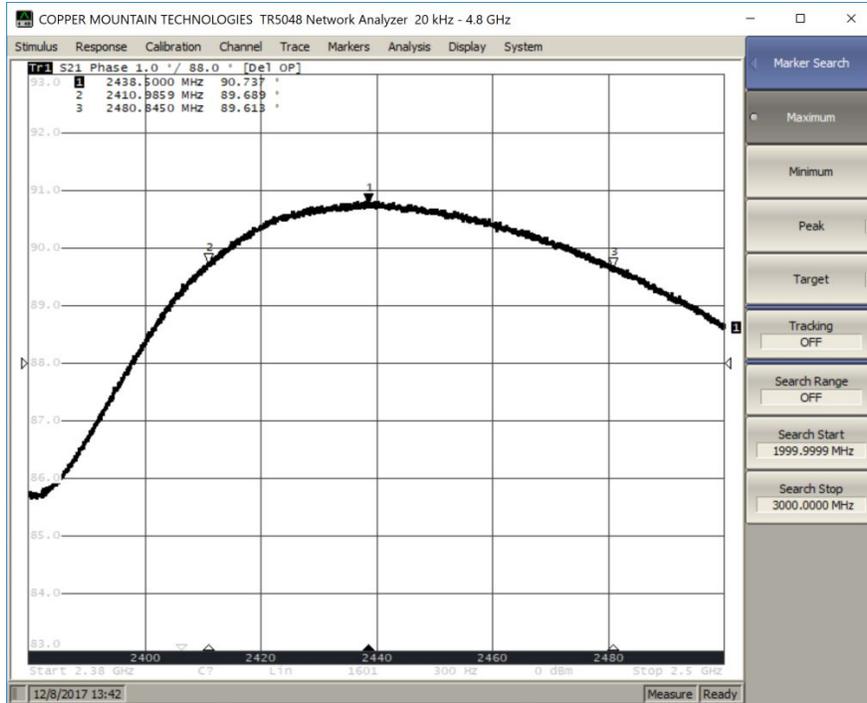


Figure 26: Amplifier deviation from linear phase.

As seen in Figure 25, the group delay at the center frequency is approximately 0.7ns. The phase response in Figure 26 looks very bad at a first glance, but the grid on the plot is in increments of 1 degree. This means that the total phase variation over the bandwidth is approximately 1.1 degrees, which is very linear. This means the amplifier is both very fast and has a phase velocity that doesn't vary significantly over the passband.

### Gain Spikes near the Passband:

The region between 2.72GHz and 2.80GHz has some rather strange behavior (see Figure 27). The gain steadily rises, and then suddenly falls before leveling out again. The phase in this frequency range (see Figure 28) also has a corresponding kink in it. This behavior is most likely due to the self resonance of the capacitive transmission lines. The calculated capacitance of these shunts was 4.7pF. If this value is plotted on the graph of resonant frequency vs. capacitance in Figure 18, it is seen that the point for 4.7pF and 2.4GHz lies very close to the data for self resonance. From this it can also be theorized that the other spike in the gain that occurs at 2.358GHz is the self resonance of the inductive transmission line. This conclusion, however, cannot be supported by a similar plot and therefore is merely speculation.

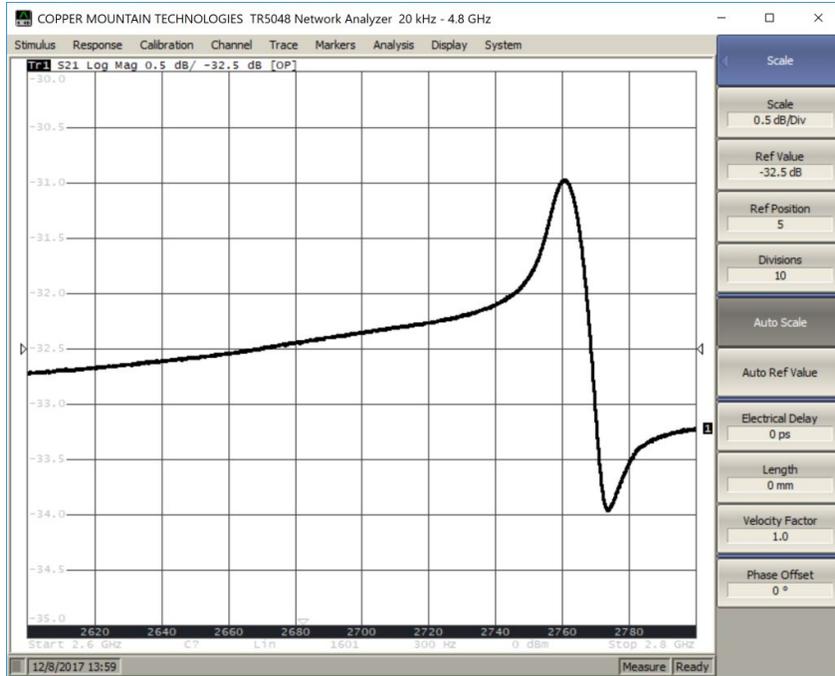


Figure 27: Magnitude of  $S_{21}$  around the large spike in amplifier gain after the pass-band.



Figure 28: Phase of  $S_{21}$  around the large spike in amplifier gain after the pass-band.

### Cascaded System Test:

As a final experiment, the amplifier was cascaded with a filter and an antenna to test if the amplifier could amplify a test signal from the network analyzer enough to be received by a second antenna (see Figure 30). The antennas used were a fractal antenna for the transmission and a patch antenna for receptions. The fractal antenna had a VSWR of 1.3, while the patch antenna had a VSWR of around 1.7 (see Figure 29).

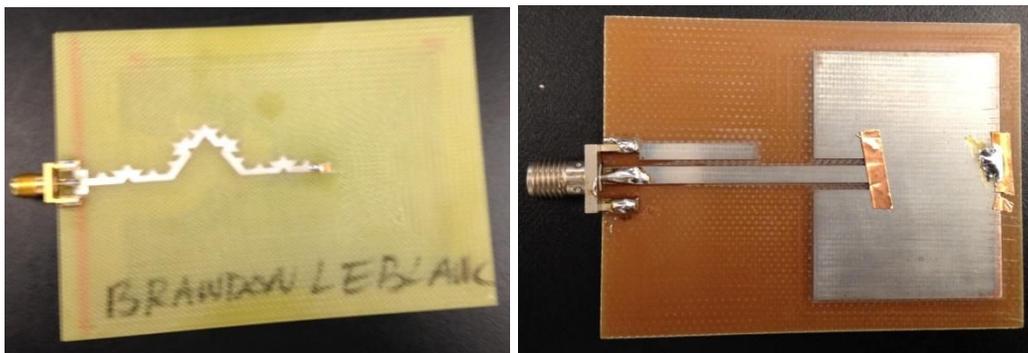


Figure 29: Fractal antenna used for transmission and patch antenna used to receive signal

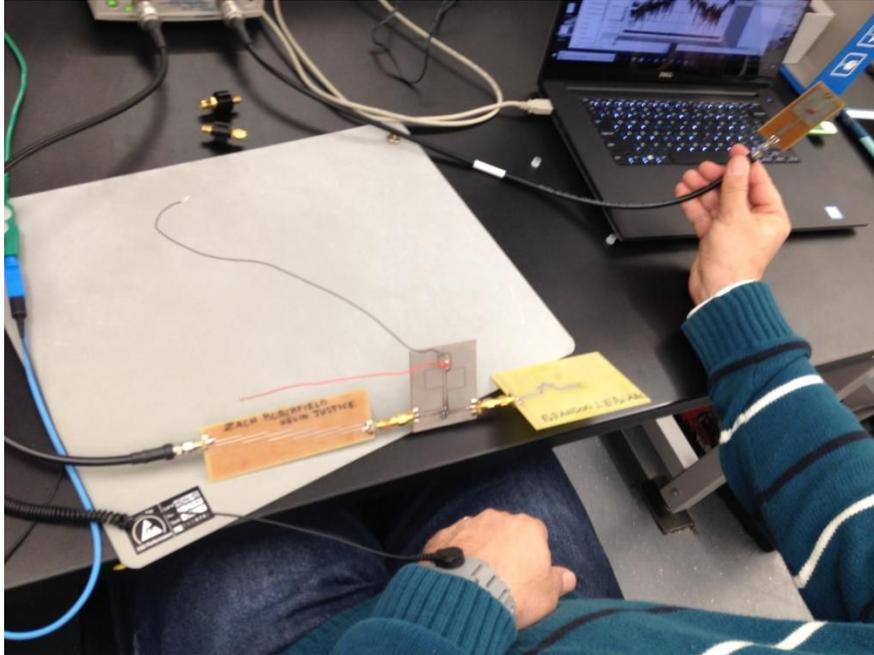


Figure 30: Testing configuration for the cascaded system.

The filter was a Chebyshev filter with 0.5dB of ripple in the passband. The filter missed the center frequency of the WiFi band on the high end, but was to pick up the upper half of the WiFi band.

The wideband response of the cascaded system is shown in Figure 31 with the antenna separated by about 5 inches. The double peak structure is a bit concerning, but upon closer inspection it can be seen that the second peak occurs around 2.7GHz which is close to the resonant frequency of the distributed shunt capacitors. This means that if this cascaded configuration is to be used for any practical purposes the signal must be filtered after being received in addition to before being amplified.

The response of the system in the passband with an antenna separation of approximately 5 inches is shown in Figure 32. From this plot it is clear that there is approximately 3dB of ripple over the entire bandwidth and that the response is very smooth. All together, this system functioned well beyond expectation.

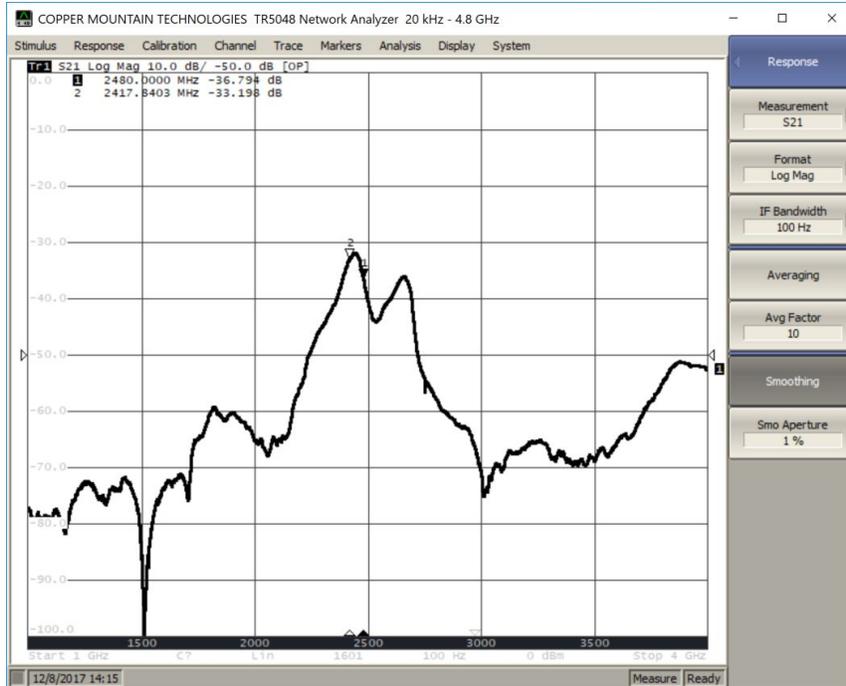


Figure 31: Wideband response of cascaded system.

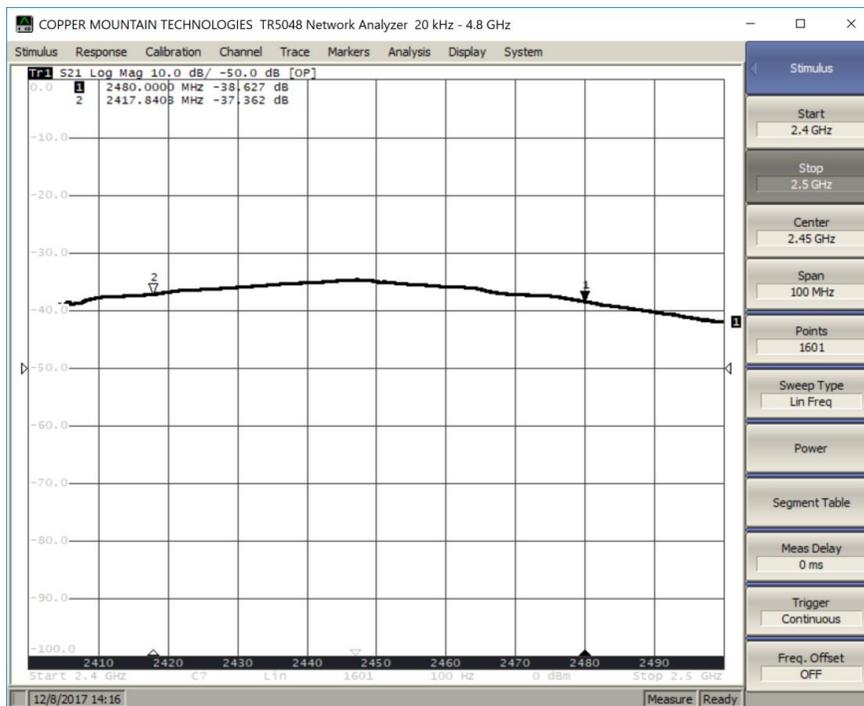


Figure 32: Passband response of cascaded system.

## Suggestions for Improvements:

This amplifier design has squeezed out as much gain from the transistor as it can. Thus changing the transistor will yield the largest increase in gain. In the initial research for the amplifier, the Infineon BFP640FESD was discovered. This transistor has a few advantages over the BFG403W, the most noticeable one being that it was designed to operate at WiFi frequencies. As a result this transistor will exhibit much better behavior over the operating bandwidth.

Another important discovery this semester was made during the testing of the 3dB Chebyshev filter. This group discovered that the milling machine in the LSU Protolab does not accurately cut the PCB traces to the exact specified widths. This is obviously a big problem when the width of a trace determines the behavior of a transmission line. For more information, see the report written for the 3dB Chebyshev filter in Fall 2017.

Finally it is strongly recommended that the designer estimate the self resonant frequency of all distributed components to ensure that a resonance does not occur in or near the region of interest. In the cascaded circuit test, the double peak structure that was received by the patch antenna was due to the self resonance of the shunt capacitors. This shows that filtering before transmission is not always enough. Future designs should also include a filter on the receiver end to filter out spurious signals like the second undesired peak. It also might be interesting to include measurements of resonance frequencies as a function of transmission line dimensions in future labs.

## Conclusion:

The design and implementation of the Class A RF amplifier was a resounding success. The design was able to meet almost all of the desired specifications. The pass-band ripple was 0.3dB, which was less than the required 0.5dB. The phase was linear to within 1.1 degrees over the entire bandwidth. The group delay was approximately 0.7ns at the center frequency. Also very importantly there was no less than 10dB of attenuation for the reverse gain. Furthermore when the amplifier was cascaded with a Chebychev filter and a fractal antenna, the cascaded system was able to transmit the test signal from the network analyzer to a patch receiver antenna.

The only specification that was not met was gain. Unfortunately this design was only able to output approximately 6.5dB of gain over the entire bandwidth. This was due to the choice of transistor. The BFG403W was not intended for wireless transmission over WiFi frequencies as the forward gain ( $S_{21}$ ) begins to roll off around 2GHz. This means that the maximum gain the transistor could provide at 2.4GHz is approximately 14dB, and 7dB need to be sacrificed in order to bias the transistor.



## Appendix A: Images from the BFG403W Datasheet

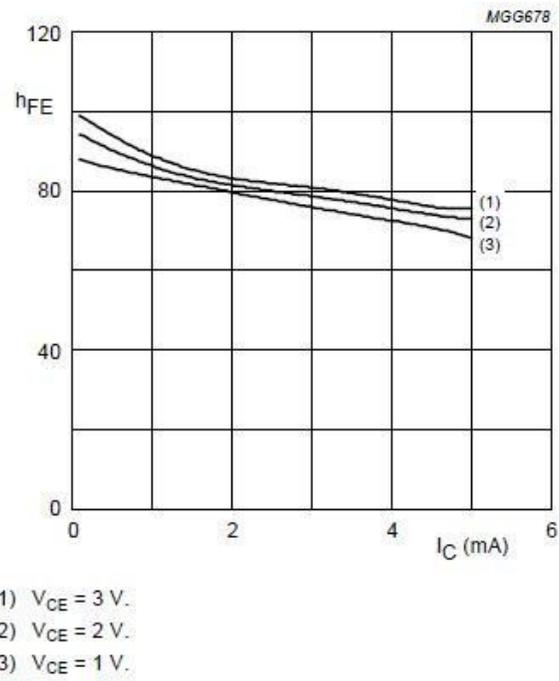


Figure 33: DC current gain as a function of collector current (figure 3).

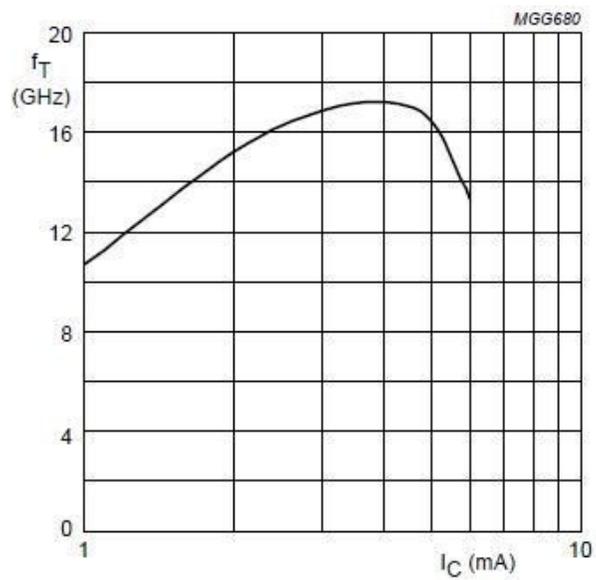


Figure 34: Transition frequency as a function of collector current (figure 5).

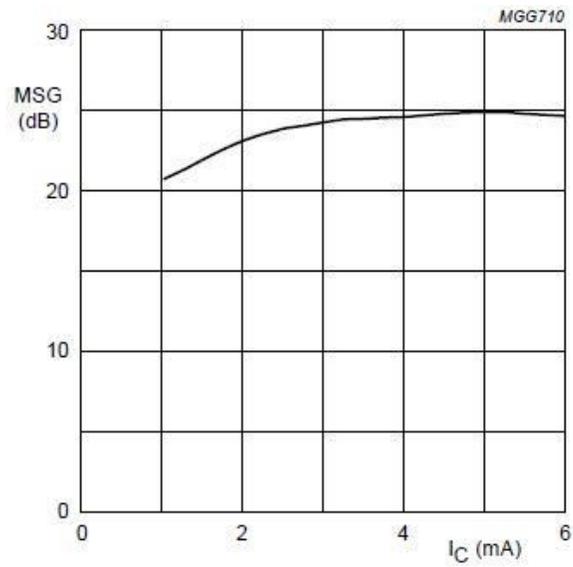


Figure 35: MSG as a function of collector current (figure 7).

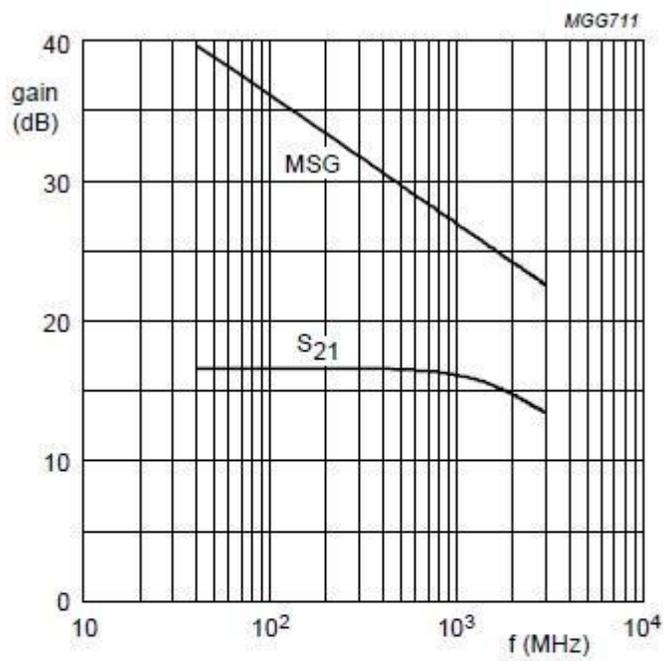


Figure 36: Gain as a function of frequency where  $I_C = 3\text{mA}$  and  $V_{CE} = 2\text{C}$  (figure 8).



## Appendix B: Matlab code to...

### Draw Circles on a Smith Chart:

```
function smith_circles(centre, radius, colour_linestyle, segments)
%SMITH_CIRCLES Draw circles on a Smith Chart.
% SMITH_CIRCLES(CENTRE, RADIUS, COLOUR_LINESTYLE, SEGMENTS) has the
% following parameters:
% - CENTRE: the centres of the circles (a complex number),
% - RADIUS: the radii of the circles,
% - COLOUR_LINESTYLE: one of the standard Matlab colour/linestyle
% combinations that will be used to draw the circles, and
% - SEGMENTS: the number of segments used to draw the circles.
% CENTRE and RADIUS must be row vectors with the same number of elements.
% COLOUR_LINESTYLE and SEGMENTS are optional. RADIUS must be positive and
% purely real, and SEGMENTS must be an integer.
%
% Examples:
% smith_circle(0.5 + j*0.7, 1.5)
% smith_circles([ 0 -0.5 0.5 -j*0.5 j*0.5 ], [ 1 0.5 0.5 0.5 0.5 ])
% smith_circles(randn(1, 100)+j*randn(1, 100), randn(1, 100))
% smith_circle(0.1 - j*0.3, 0.2, 'r--')
% smith_circle(1.5 + j*1.1, 2.1, 'k:', 64)
% smith_circles([ 0 -1 1 -j*1 j*1 ], [ 1 1 1 1 1 ], 'r-.', 2048)
%
% See also: SMITH.M

% Check the number of inputs.
error(nargchk(2, 4, nargin));

% Set default values.
if nargin < 3
    % This is Matlab's default.
    colour_linestyle = 'b-';
end
if nargin < 4
    segments = 256;
end

% This values is used a number of times below, so let us calculate it
% once here rather than many times below.
ac = abs(centre);

% Find the circles that are entirely inside the Smith Chart.
inside = find(ac + radius <= 1);
% Find circles that intersect the Smith Chart boundary.
intersect = find((ac + radius > 1) & (abs(ac - radius) < 1));
% The other circles are all completely outside the Smith Chart or the
% Smith Chart is completely inside the circles - either way, they do not
% have to be plotted.

% The sizes of inside and intersect are used rather a lot, so they are
% stored in variables.
size_inside = numel(inside);
size_intersect = numel(intersect);

% Allocate memory for the points on the circles
z = zeros(segments + 1, size_inside + size_intersect);
```

```

% Circles that are entirely inside the Smith Chart
if size_inside > 0

    % The angles from the centres of the circles to the points to be plotted. I
    % have to add 1 to the number of segments because there is one more point
    % than there are segments.
    theta = linspace(-pi, pi, segments + 1)';

    % Find the points on the circles.
    z(:, (1:size_inside)) = ones(size(theta))*centre(inside) + ...
        exp(j*theta)*radius(inside);

end

% Circles that intersect the Smith Chart boundary.
if size_intersect > 0

    % Find the angles to the intercept points.
    % The angle from the centre to the intercept points relative to the angle to
    % the centre is calculated. This calculation is based on the well-known
    % relationship:
    %  $c^2 = a^2 + b^2 - 2*a*b*\cos(\theta)$ 
    % Note that there is a -1 factor implicit in the next line which
    % ensures that the correct angle is used.
    theta_p = acos((1 - radius(intersect).^2 - ac(intersect).^2)./ ...
        (2.*radius(intersect).*ac(intersect)));
    theta_1 = angle(centre(intersect)) - theta_p;
    theta_2 = angle(centre(intersect)) + theta_p;
    % I need to change the order of these angles to get the part of the circle
    % that is inside the Smith Chart boundary.
    theta_1 = theta_1 + 2*pi;

    % The angles from the centres of the circles to the points to be plotted. I
    % have to add 1 to the number of segments because there is one more point
    % than there are segments.
    theta = ones(segments + 1, 1)*theta_1 + ...
        linspace(0, 1, segments + 1)'*(theta_2 - theta_1);

    % Find the points on the circles.
    z(:, ((size_inside + 1):(size_inside + size_intersect))) = ...
        ones(segments + 1, 1)*centre(intersect) + ...
        exp(j*theta).*(ones(segments + 1, 1)*radius(intersect));

end

% Finally, plot all the circles.
plot(z, colour_linestyle);

```

## Plot the Scattering Parameters as a Function of Frequency:

```

%S Parameters Plotted on Smith Chart
data = read(rfdata.data,'Spar_BFG403W_commonemitter2V3mA.S2P');
freq = data.Freq;
s_params = extract(data,'S_PARAMETERS',50);
y_params = extract(data,'Y_PARAMETERS');

s11 = s_params(1,1,:);
smithchart(s11(:));
hold on
s22 = s_params(2,2,:);

```

```
smithchart(s22(:));
legend('s11','s22');
s21 = s_params(2,1,:);
smithchart(s21(:));
s12 = s_params(1,2,:);
smithchart(s12(:));
legend('s11','s22','s21','s12');
```

## Plot the Stability Circles on a Smith Chart:

```
%Stability Circles and Operating Gain
amp = read(rfckt.amplifier,'Spar_BFG403W_commonemitter2V3mA.S2P');
fc = 2.437e9;
circle(amp,fc,'Stab','In','Stab','Out','Gp',17:-1:13);
legend('Location','SouthEast')
```

## Plot the Input and Output Gain Circles on a Smith Chart:

```
%Uses gs, center of gs and radius of gs to plot input gain circles
s11 = 0.255202 - j*0.474756;
s21 = -0.21819 + j*4.77001;
s12 = 0.01343+ j*0.018671;
s22 = 0.59579 - j*0.57643;

amp = read(rfckt.amplifier,'Spar_BFG403W_commonemitter2V3mA.S2P');
fc = 2.4e9;
delta = s11*s22 - s12*s21 ;
Cout = ( conj(s22)-conj(delta)*s11 ) / ( abs(s22)^2-abs(delta)^2 ) ;
Rout = abs(( s12 * s21 ) / ( abs(s22)^2-abs(delta)^2 ) ) ;
Cin = ( conj(s11)-conj(delta)*s22 ) / ( abs(s11)^2-abs(delta)^2 ) ;
Rin = abs(( s12 * s21 ) / ( abs(s11)^2-abs(delta)^2 ) ) ;

gamma_in = s11 ;
gamma_out = s22 ;
gamma_S = conj(s11) ;
gamma_L = conj(s22) ;

%Calculating Center and Radius of Input Stability Gain Circles
Gs = (1 - (abs(gamma_S)^2))/((abs(1-(gamma_S)*(s11)))^2);
Gs_dB = 10*log10(Gs);
Gs_Max = 1/(1-(abs(s11)^2));
Gs_Max_dB = 10*log10(Gs_Max);
G0 = (abs(s21))^2;
G0_dB = 10*log10(G0);
GL = (1 - (abs(gamma_L)^2))/((abs(1-(gamma_L)*(s22)))^2);
GL_dB = 10*log10(GL);
GL_Max = 1/(1-(abs(s22)^2));
GL_Max_dB = 10*log10(GL_Max);
Gtu = Gs*G0*GL;
Gtu_dB = Gs_dB + G0_dB + GL_dB;
Gtu_Max = Gs_Max*G0*GL_Max;
Gtu_Max_dB = Gs_Max_dB + G0_dB + GL_Max_dB

%Calculating Output Stability Gain Circles
gl = GL_Max*(1-(abs(s22)^2));
cgl = (gl*conj(s22))/(1 - ((abs(s22)^2)*(1-gl)));
rgl = (((1-gl)^0.5)*(1-(abs(s22)^2)))/(1-((abs(s22)^2)*(1-gl)));
%1dB less
GL_1dB = exp(((GL_Max_dB - 1)*log(10))/10);
```

```

gl_1dB = GL_1dB*(1-(abs(s22)^2));
cgl_1dB = (gl_1dB*conj(s22))/(1 - ((abs(s22)^2)*(1-gl_1dB)));
rgl_1dB = (((1-gl_1dB)^0.5)*(1-(abs(s22)^2)))/(1-((abs(s22)^2)*(1-gl_1dB)));
%2dB less
GL_2dB = exp(((GL_Max_dB - 2)*log(10))/10);
gl_2dB = GL_2dB*(1-(abs(s22)^2));
cgl_2dB = (gl_2dB*conj(s22))/(1 - ((abs(s22)^2)*(1-gl_2dB)));
rgl_2dB = (((1-gl_2dB)^0.5)*(1-(abs(s22)^2)))/(1-((abs(s22)^2)*(1-gl_2dB)));
%3dB less
GL_3dB = exp(((GL_Max_dB - 3)*log(10))/10);
gl_3dB = GL_3dB*(1-(abs(s22)^2));
cgl_3dB = (gl_3dB*conj(s22))/(1 - ((abs(s22)^2)*(1-gl_3dB)));
rgl_3dB = (((1-gl_3dB)^0.5)*(1-(abs(s22)^2)))/(1-((abs(s22)^2)*(1-gl_3dB)));
%4dB less
GL_4dB = exp(((GL_Max_dB - 4)*log(10))/10);
gl_4dB = GL_4dB*(1-(abs(s22)^2));
cgl_4dB = (gl_4dB*conj(s22))/(1 - ((abs(s22)^2)*(1-gl_4dB)));
rgl_4dB = (((1-gl_4dB)^0.5)*(1-(abs(s22)^2)))/(1-((abs(s22)^2)*(1-gl_4dB)));

%Calculating K Factor and B parameters
K = (1 - (abs(s11)^2) - (abs(s22)^2) + (abs(delta)^2))/(2*(abs(s12*s21)))
B = 1 + (abs(s11)^2) - (abs(s22)^2) - (abs(delta)^2)

gs = Gs_Max*(1-(abs(s11)^2));
cgs = (gs*conj(s11))/(1 - ((abs(s11)^2)*(1-gs)));
rgs = (((1-gs)^0.5)*(1-(abs(s11)^2)))/(1-((abs(s11)^2)*(1-gs)));
%Calculating Input Stability Gain Circles
Gs_1dB = exp(((Gs_Max_dB - 1)*log(10))/10);
gs_1dB = Gs_1dB*(1-(abs(s11)^2));
cgs_1dB = (gs_1dB*conj(s11))/(1 - ((abs(s11)^2)*(1-gs_1dB)));
rgs_1dB = (((1-gs_1dB)^0.5)*(1-(abs(s11)^2)))/(1-((abs(s11)^2)*(1-gs_1dB)));

Gs_2dB = exp(((Gs_Max_dB - 2)*log(10))/10);
gs_2dB = Gs_2dB*(1-(abs(s11)^2));
cgs_2dB = (gs_2dB*conj(s11))/(1 - ((abs(s11)^2)*(1-gs_2dB)));
rgs_2dB = (((1-gs_2dB)^0.5)*(1-(abs(s11)^2)))/(1-((abs(s11)^2)*(1-gs_2dB)));

Gs_3dB = exp(((Gs_Max_dB - 3)*log(10))/10);
gs_3dB = Gs_3dB*(1-(abs(s11)^2));
cgs_3dB = (gs_3dB*conj(s11))/(1 - ((abs(s11)^2)*(1-gs_3dB)));
rgs_3dB = (((1-gs_3dB)^0.5)*(1-(abs(s11)^2)))/(1-((abs(s11)^2)*(1-gs_3dB)));

%%%%%%%%Plotting Input Stability Gain Circles%%%%%%%%
smithchart;
hold on;
smith_circles(cgs, rgs, 'g', 2048)
hold on
%1 dB less
smith_circles(cgs_1dB, rgs_1dB, 'k', 2048)
hold on
%2 dB less
smith_circles(cgs_2dB, rgs_2dB, 'y', 2048)
hold off
%Plotting Output Stability Gain Circles

smithchart;
hold on;
smith_circles(cgl, rgl, 'g', 2048)
hold on
%1 dB less

```

```
smith_circles(cgl_1dB, rgl_1dB, 'k', 2048)
hold on
%2 dB less
smith_circles(cgl_2dB, rgl_2dB, 'y', 2048)
hold on
%3 dB less
smith_circles(cgl_3dB, rgl_3dB, 'b', 2048)
hold on
%4 dB less
smith_circles(cgl_4dB, rgl_4dB, 'r', 2048)
```

## Appendix C: References

### Multisim:

How to create a custom component in Multisim: <http://www.ni.com/tutorial/3173/en/>

How to Import and Export components in Multisim:

<https://knowledge.ni.com/KnowledgeArticleDetails?id=kA00Z000000P6VLSA0>

### Textbooks:

RF Circuit Design: Theory and Applications, by Reinhold Ludwig and Gene Bogdanov

### External Plots:

Capacitor Self Resonance: [http://leleivre.com/notes\\_cap\\_srf.html](http://leleivre.com/notes_cap_srf.html)