Project 1

Tuned RF Amplifier with 50 Ohm I/O Impedances

Due Date: Friday 10/11/2019 at "COB"

Objectives:

Basic common-emitter amplifiers like that built in Lab 0 generally have insufficient gain at RF frequencies. Moreover, they do not provide the bandpass filtering needed in a radio receiver or radio transmitter.

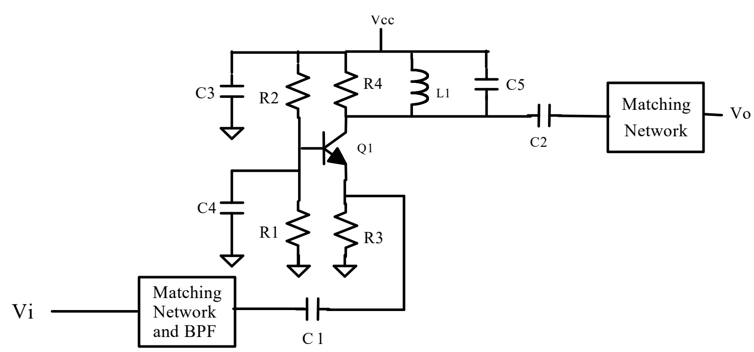
In this project we will correct these problems by:

- converting our common-emitter amplifier to a common-base or cascode configuration (your choice)
- adding a parallel RLC circuit at the output to resonate out the effects of collector node capacitance,
- implementing matching networks to provide good power gain in a 50 Ohm system, and
- designing the resonant circuits to provide a two-pole response with a specified bandwidth

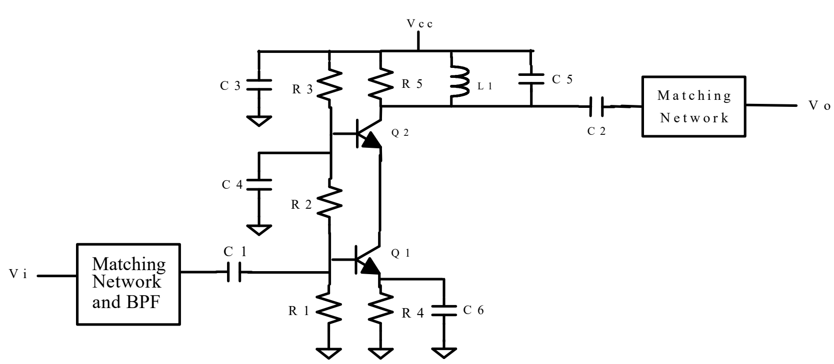
IMPORTANT: *<u>This assignment is a two-person project. You should work with a teammate to reduce your (and my) workload :-)</u>*

Recommended circuit topologies are shown below.

Common Base



Cascode



Specifications:

Your task in this project is to design an MMBT5179 based amplifier together with suitable matching networks and filtering to meet the following specifications:

- Nominal center frequency of 98 MHz.
- Insertion gain of > 15 dB
- Input/output impedances of 50 Ohms (return loss > 10 dB at 98 MHz).
- 3 dB bandwidth of 20 MHz (±20%) when operating with 50 Ohm source and load.
- Two-pole bandpass filter shape, with passband ripple < 3 dB
- Output compression point > +4 dBm
- 60 mW (± 20%) power consumption from a single 5V supply.

These specifications are based on the performance needed for successful implementation of the radio transmitter and receivers you will assemble for the midterm and final.

Teamwork:

You should work with a teammate on this project and divide the workload approximately evenly. I.e., the work should be collaborative, EXCEPT at the writeup stage.

Writeups - division of labor:

When you write up the project, one person must document, in their own words and figures, the design, schematic, and layout - even though the design should be done by the team as a whole. The other person must document, in their own words and figures, the measurements and how they compared to expected performance - even though both should be present and participate in making the measurements and assessing their validity.

In other words, you can and should help each other in getting the project done, but ***the writeup sections should be separately done, and then combined into a single team "deliverable" turned in at the end of the project***.

Number of circuit copies needed:

While only a single circuit must be constructed for this lab, you will each need a copy for the midterm and final - so it is recommended you build the second copy immediately after completing the measurements here, and confirm the second copy has the same performance.

Grading:

Grading will have both a team component and an individual component, so ***you must identify who wrote which part of the report***. In the next project, the roles in the writeup will be reversed.

Suggested design procedure:

A suggested design procedure is outlined below.[1]

Step 0: Amplifier DC Power and Maximum RF Output Power

The collector current and total amplifier current are constrained by the power supply specification. Use this and the transistor datasheet to decide on a reasonable collector current and then design your bias circuits. Also estimate the maximum output power you can get with this DC power. It can be shown that a class-A RF amplifier such as this can achieve up to a 25% efficiency when the output impedance and load impedance are matched - so the largest output power is somewhat less than 25% of the DC supply power used. (The max output power also depends on the Rc value used - but do step 1 below before worring about this...)

Step 1: Amplifier Power Gain

For this project, you may start with an Rc value similar to what you had in project 0. Then compute the power gain this will give (see lecture handout on Power Gain). This should meet the gain spec regardless of which amplififer topology you decide on.

In your computation of voltage and power gain, note that R in "gm*R" is the parallel combination of Rc and R_L', where R_L' is the load impedance seen looking through the matching network to the actual 50 Ohm load. In addition, note that only half of the power to R will reach the output, since half will be dissipated in Rc. Be sure to show in your writeup how you did your calculations and took into account these issues.

Step 2: Tuned circuit Design

The inductor L1 must resonate (at the desired frequency of operation) with the total capacitance at the collector node. This capacitance consists of:

- The transistor output capacitance (Ccb)
- The capacitance of any PC board traces on the output node
- The additional capacitor C6 used to help set the resonator bandwidth.

  Note that coax/wiring and probe loading capacitances we dealt with in project 0 are <u>not</u> present now, since we will be driving a 50 Ohm resistive load through "50 Ohm" coax and will implement a matching network to convert this *resistive* 50 Ohm load value to a *resistive* value of Rc to provide good output matching.

Recalling the expression for bandwidth in terms of center frequency and Q, and for Q in terms of parallel resistance R_p and reactance X_p, find the value L and of the total C needed to achieve the desired bandwidth. <u>Be sure to use the "loaded Q" in your computations</u> by noting that the

[1] The suggested procedure is simplified so that we can gain experience with basic concepts. It is based on our simplified hybrid-pi transistor model. A more exact procedure can be found in various textbooks and uses detailed transistor modeling in the form of Y or S parameters. Additional S-parameter design topics are discussed in ECE 764 and the class textbook.

matching network will transform the 50 Ohm load impedance to a value equal to R_c, and that this transformed load will be in parallel with the physical resistor R_c.

You should also estimate the value of the transistor and PC board capacitances and subtract these from C_{total} to give the value that must be contributed by C6 - although it may not be a substantial change. Note that you should make either C6 or L1 variable so that the circuit can be "aligned" after it is built, and this is likely to address any small adjustments needed from these sources.

Step 3: Matching Network Design

Design matching networks to convert the input and output impedances of the amplifier to 50 Ohms (and visa-versa). As noted above, the output impedance can be considered essentially resistive since any capacitances are "absorbed" in the design of the resonator in Step 2. However the input cannot, especially in the case of the cascode amplifier topology.

Think about how you can handle the situation at the input by applying resonance principles. Also note that your design ***must*** be DC isolated from the source and load. This is done in the suggested circuit with an additional coupling capacitor C1 between the matching network and source/load, although there may be more economical approaches depending on how you implement your matching network.

Finally, modify your input matching network to achieve the specified filtering. You may need to increase the Q from that needed to achieve the match. A loaded Q of 10 is recommended to meet the specified bandpass 2-pole shape factor by stagger tuning the input and output resonant circuits.

Simulation:

After designing your circuit, use a simulator in AC analysis mode (either PSPICE, or the Agilent ADS circuit simulator available in the department's computing lab[2]) to check it. You should use the correct transistor model, since the internal capacitances and the n value matter !

Make a plot in dB versus linear frequency from 20 MHz to 200 MHz to show the gain and bandwidth. Be sure your plot shows "insertion gain" in a 50 Ohm system. Note that the loaded voltage gain from Vi *(not Vs)* to Vo plotted in dB should give this to you when you use 50 Ohm source and load impedances (since R_i should be equal to R_s now if the matching networks are working). At the same time, check the input and output impedances with the simulator and verify that they are indeed close to 50 Ohms resistive at the center frequency. *(Check/plot both real and imaginary parts of Z, not just |Z| ! Or - use ADS and plot using a Smith Chart)*

---

[2] See your instructor for information on using Agilent ADS if you wish to try this commercial-grade RF circuit simulator.

Layout:

Plan the layout of your circuit. When placing inductors, try to avoid allowing the inductors in the input and output circuits from "talking to each other" through magnetic coupling. Otherwise you may end up building an oscillator instead of an amplifier!

Construction and Measurements:

Before beginning, draw your final circuit schematic with part values shown and have this with you as you take measurements and troubleshoot any problems. Having your layout with you is also a good idea.

IMPORTANT!!!

Use an input amplitude of -30 or -40 dBm for all measurements involving an input signal applied to the circuit. If you have a higher level, you will not be using a "small signal" and may get incorrect readings due to saturation of the transistor (at Vbe or Vc).

Measurement 1: DC and stability checks

Apply 9V DC to your circuit and check the current consumption and bias voltages. If all looks OK, connect the output to an instrument suitable for detecting oscillations (i.e. a spectrum analyzer with frequency capability up to at least the transistor's f_T). With no input applied, you should have no output (obviously). If you ***do*** have an output, your amplifier is probably oscillating and will need to be fixed before proceeding. This usually results from parasitic capacitance/inductances in the layout or coupling between coils in the input and output circuits.

Measurement 2: Gain and Bandwidth

Using one of the network analyzers, measure the following. ***Be sure the input power is -30 dBm for both measurements. This requires setting the port-1 power to -10 dBm and the port-1 attenuator to 20 dB, before you perform an S21 through-cal*** !

- The frequency response over the range 1 MHz to 200 MHz in a 50 Ohm system and the 3 dB and 6 dB bandwidths. Record/verify the bandwidth and then plot the response using the procedure outlined on the network analyzer's instruction sheet. Take a photo of the screen.
- The insertion power gain of the amplifier at midband (100 MHz) in a 50 Ohm system. Be sure to align the tuned circuits for max gain.

Measurement 3: Input/Output Impedances

Using the network analyzer, measure the input and output impedances of your amplifier over the frequency range 50 to 150 MHz. For this measurement, you will need to calibrate the network analyzer (S11 One-port cal) for the exact length of coax cables leading to the input (or output) of your circuit. Again, be sure to configure the port-1 power to -30 dBm ***before*** doing this S11 cal.

When recording your final results, use the Smith Chart format and place a marker at the center frequency (100 MHz) before drawing or photographing the screen displays for your writeup.

From your Smith charts, find the impedances in rectangular form (R + jX) at 88, 98, and 108 MHz and write these impedances next to the appropriate locations on the charts. ***In your notebook, you must explain how you can find these values from reading the Smith Chart***. It is not sufficient to use the Network Analyzer's marker function to find the answers. The objective here is to understand and eventually become comfortable with this form of impedance presentation.

In addition, explain why the impedance varies and why you see the values you do at low and high frequencies (well below and well above center frequency). Why does it have the reactance value/sign you see at the two frequency extremes?

Finally, calculate the return loss at 98 MHz. Draw 10 dB return loss circles on your Smith chart plots to show if your impedances fall within spec.

Measurement 4: 1 dB Compression Point

If the input signal to an amplifier becomes too large, the output of the amplifier will begin to saturate and the gain will begin to fall. ***The input signal level where the gain falls by 1 dB is referred to as the 1 dB input compression point of the amplifier.*** This is a very important spec for an RF amplifier[3].

Using one of the RF signal generators, apply a small signal (e.g. -40 dBm) at the center frequency of your amplifier and monitor the output ***power*** with the spectrum analyzer. Increase the input power in steps of 2 dB and record the output for each step. Continue taking measurements up to an input power of at least 0 dBm. Record your results in a table and then compute gains at each input power level, and add this to the table.

Plot the output power in dBm versus input level in dBm and find the input level at which the gain falls by 1 dB from its maximum value. Mark this point on your graph as the 1 dB compression point. The corresponding input signal power is the 1dB *input* compression point. The corresponding output signal power (+ 1 dB) is the *output* compression point. An alternative plot might show gain in dB versus input power in dBm and show the point where the gain has fallen by 1 dB.

---

[3] An equally important measure is the 3rd order intercept point which we will discuss later in the course.

Write-up:

See the instructions on page 1 for required teamwork (who authors what).

Write up your design and measurements in a neat form. In your write-up, you must include, **in the following order:**

- A schematic diagram ***with all part VALUES shown*** ***DIRECTLY NEXT TO THE COMPONENTS***. *These values should be actual, available component values - not raw computed values*.
- An AC small-signal diagram of your amplifier using the high frequency transistor model and *including* ***only*** *elements that determine the response over the range of* 50 to 150 MHz.
- Your layout and a photo of your board
- Narrative and equations to justify your component value selections. It is NOT acceptable to pick values by trial and error, even with the help of a simulator. You should understand how the values are calculated and must demonstrate this understanding by explaining their selection in your write-up. A simulator should be used only for checking for gross errors and for optimization and plotting.
- Plots and discussion of the simulated and measured insertion power gain in dB versus frequency (with 50 Ohm source and load)
- Plots and discussion of the simulated and measured I/O impedances confirming they are close to 50 Ohms resistive.
- Plots and discussion of the 1dB compression point (measured).
- A summary of your circuit's performance and how it compared to specs.

Scoring:

Your writeup scores will be based on Completness, Correctness, and overall Quality (CCQ). Be sure to review all the assignment steps, especially to get a good score on completeness. If you failed to meet any specifications, discuss reasons and ways that the problem(s) could be corrected.