Laboratory 6: Multi-Stage Amplifier Design Project

Issued Friday, Nov. 15, 2019 ⇨ Due 5 p.m., Friday, Dec. 13, 2019

A. OBJECTIVE

In this laboratory exercise, you will design, build, and demonstrate a multi-stage amplifier that meets the following specifications:

(a) Midband gain: $A_v = \left|\frac{v_o}{v_s}\right| \geq 4{,}500$

(b) Lower corner frequency: $f_L \leq 100$ Hz

(c) Upper corner frequency: $f_H = 10$ kHz $\pm$ 1 kHz

(d) Input impedance: $R_i > 100$ kΩ

(e) Output impedance: $R_o < 20$ Ω

(f) Undistorted output voltage swing: $V_{opp}(max) > 3V_{pp}$

(g) All specifications must be met while loaded by an oscilloscope probe and a load resistor R_L=100Ω.

(h) No more than 4 transistors total (of either npn or pnp).

(i) Minimize external capacitance, i.e., coupling and bypass.

(j) Reasonable bias stability: $\frac{R_E}{\alpha} \geq 10\frac{R_{BB}}{\beta+1},\ I_{BIAS} \geq 10I_B$

(k) All resistors and capacitors must be standard 5% values (no parallel/series combination).

(l) R_s = 4kΩ

(m) V_{CC} = +12 V (no negative power supply voltage rail).

Note that you must drive a load of 100Ω and limit the bandwidth to 10kHz. This kind of requirement is akin to some audio amplifiers (although most would drive 4, 8, or 16 ohms). To limit the bandwidth, you will need to add a compensation capacitor. For this design, add the *minimum* amount of extra capacitance for compensation. Also, note that the load will influence your amplifier design. For this design, you can choose any topology you want as long as you do not exceed FOUR transistors. Also, for this design you need not include the effect of r_b.

When testing your amplifier, make sure the input signal is kept small to avoid distortion at the output. You should use a signal attenuator similar to that used in Lab 5, except with a higher level of attenuation (to accommodate the higher gain in this design). Notice that to get an output voltage swing of 3V and a gain of 4500, the input signal should be less than 0.7 millivolts. Also, note that the source resistance for this design project is 4kΩ (different from that used in the first project). You should build an attenuator that will give you the necessary attenuation and source resistance.

B. REMARKS

(a) This laboratory provides much more flexibility in the configuration and design of your amplifier than Lab 5. Thus, expect that there will be many more ways to approach the design and many more solutions that meet the specifications.

(b) Assume a source resistance of 4kΩ. (Different than in Lab 5.)

(c) Again, this design is meant to amplify a small signal. You should keep your generator input low enough using a matching circuit to insure the small-signal models used for design are justified. Figure 1 depicts an example circuit. Note that the 51Ω resistor is only necessary if you are operating the signal generator in 50Ω (as opposed to high-*Z*) mode.

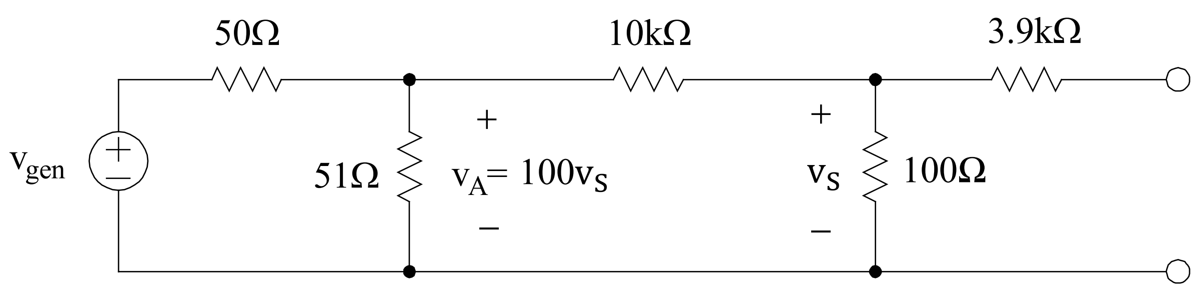

Fig. 1: Attenuator circuit for Lab#6.

To make measurements with this circuit, first calculate v_A/v_S from measured resistor values. Then measure v_o/v_A with an oscilloscope. Your gain, $v_o/v_S = (v_o/v_A)*(v_A/v_S)$.

(d) Verify all specifications by both hand and SPICE simulation.

(e) You will need to verify the maximum undistorted output swing of your amplifier. Be aware that this maximum depends on two limitations:

1) All transistors in the amplifier must stay within the limits of the forward active region.

2) The amplifier must meet the V_{opp}(max) specification with less than 10mV zero-to-peak across any transistor's small signal v_{be}. This ensures the validity of the hybrid-π model at full output swing.

You should determine which one of these two constraints is more limiting to analytically determine the maximum *linear* output voltage swing.

(f) You must drive a 100Ω resistive load in this design. Note that this will significantly influence the overall amplifier design.

(g) It is not necessary to include the effect of r_b in this design.

(h) Compute f_L and f_H using the SCTC and OCTC methods, respectively. When finding f_H, include all capacitors, even parasitics, in your time constant equations. Explicitly show the appropriate calculations to justify your choice of placement for the compensation capacitor (which must be minimized in value).

C. HAND DESIGN

Approach the hand design as you did in Lab 5, once again assuming reasonable values for β and V_A. You can also assume values for C_μ and C_π upon C_μ vs. V_{CB} curves you measure for one device and information from the 2N3906 datasheet. In other words, you need not measure the values for C_μ and C_π for all transistors in your circuit. (You can, of course, measure them if you want.) You will, however, need to measure β and V_A, and incorporate these in your final calculations and simulations.

There are two items that you should be aware of when doing this design project.

1) As you can see from the specifications, the bandwidth of the amplifier should be about 10kHz (±1kHz). Most likely your uncompensated design will have a bandwidth much larger than the specification. The combination of this large bandwidth with large gain may induce oscillation in your circuit. (You will probably see this in your breadboard implementation.) Reducing the bandwidth intentionally using a compensation capacitor helps to eliminate this oscillation. When you use a compensation capacitor, you must place it in the location that minimizes its value.

2) Note that the β of a transistor is a function of I_c. It decreases after I_c gets beyond a certain value. You might need to account for this in your design. Do not automatically assume that β remains high when I_c goes above 10-20mA.

D. TRANSISTOR MEASUREMENTS

Again, you need not measure the values for C_μ and C_π for all transistors in your circuit. Rather, you may assume values for C_μ and C_π in this design project, based upon C_μ vs. V_{CB} curves obtained for one device and information from the 2N3906 data sheet. You will, however, need to measure β and V_A, and incorporate these in your final calculations and simulations.

E. SPICE SIMULATION

Confirm all amplifier specifications in SPICE. Include C_μ's and C_π's explicitly in your schematics. When simulating you should do the following:

1) When doing your AC simulation, make sure to print out the gain at each stage of your amplifier. In other words, find the gain from the input to each node in the amplifier signal path (i.e., each node between gain stages).

2) To determine the input resistance R_i of your amplifier, simply calculate it from resistor values using the final measured parameters for the transistors and using SPICE simulation. Measuring amplifier input resistance is not easy without the right instrumentation, so we will not require you to make this measurement in the lab. For input resistance R_i you should

show both equations and calculated values. Obviously, you will need to calculate, simulate, and measure values for other parameters like bandwidth, gain, swing, etc.

3) You will need to determine the output resistance R_o of your amplifier in several ways. The first method is to just calculate it from your small-signal model. The second is to extract it from SPICE. The third is to measure it (indirectly) on the actual circuit by changing the load, which changes the gain of the last voltage divider stage.

F. LAB REPORT

Your report should be a concise yet comprehensive description and evaluation of your design, simulation, and laboratory measurements. Excessive paperwork and extraneous information is not acceptable. Your report should contain no more than 20 pages of text and figures. Extra pages may be necessary for spice output graphs. Neatness will be a percentage of your grade. The report should look professional, with nice figures and captions.

(a) Format: You should have the following separate sections in your lab report:

1- Design Strategy and Discussion (~10-15%)

2- Hand Design (~35-45%)

3- Computer and SPICE Simulations (~10-20%)

4- Experimental Results (~20%)

5- Summary Including Table (~10%)

Note that the above indicates the approximate weight of each section towards the overall report grade.

The Design Strategy section should include a discussion of how you approached the design problem and what trade-offs in the specifications you encountered. If you had to iterate, describe what guided your iterations. You should use equations to help explain this. This section should contain a clear diagram showing the entire circuit topology, and a table summarizing your complete design, including the operating bias point. Note that this table is different from the one in the Summary section, which focuses on performance (as opposed to design).

The Hand Design section should contain only the final design iteration. The best approach to the Hand Design section is to provide equations in variable form for different parameters and plug in numbers that yield the final design.

The contents of the SPICE Simulation and Experimental Results sections are somewhat obvious.

The Summary section should include a table that compares the specified parameters from hand design, SPICE, and lab measurements. You should compare results and explain discrepancies between the three values. Again, be concise in your explanations.

(b) Calculations: Show all equations in variable form before evaluating. But just show final equations and those that are relevant to your design. We do not need long derivations.

(c) Graphics and Computer Outputs: Include plots that definitively show your results for each specification. Neatly label all plots and spice outputs.