EECE 2412 – Homework 8, 9 (Project) – Fall 2018

Due: Monday, November 19, 2018

Design Project

Please direct ALL PSPICE questions to the TAs (email addresses and office hours are listed on Blackboard). However, general question on electronics and circuit design may be directed to either the TAs or the instructor.

Submission instructions: Prepare a **typed report** with all design descriptions, analyses, schematics, and simulation results; and submit the printed hard copy of the report at the beginning of the lecture on the due date. In addition, prepare a .zip file that contains all PSPICE files and a scanned version of your report in .pdf format. Email the .zip file prior to the lecture on the due date to Mengting Yan (one of the teaching assistants) at yan.me@husky.neu.edu, and copy the instructor who teaches your lecture section.

Report sections:

- *Introduction/Background* (describe potential applications of this amplifier [open-ended, based on your literature search] and related design requirements with references)
- *Proposed Design* (describe/analyze the circuit you selected)
- *Simulation Results* (provide schematics, include plots to demonstrate the performance, and discuss results)
- *Summary* (summarize and provide conclusions)
- *References* (can be textbook(s), journal/conference papers, online resources)

Grade: The project grade will have the same weight as two homework grades.

Goal: Design an amplifier according to the following constraints and specifications, which must all be validated by including the appropriate simulation results in the printed report:

1. The available active devices are Q2N2222 NPN transistors.
2. The power supplies must be +3V and -3V.
3. As shown in the figure below, the amplifier must be coupled to a 20kΩ load resistor (R_L) through a large capacitor.
4. The input impedance (Z_{in}) must close to 100Ω (i.e., matched to R_S) over a 100Hz - 100kHz frequency range.
5. The voltage gain from the source to the output must be at least 30 ($A_{vs} = v_o / v_s \geq 30$) over a 100Hz - 100kHz frequency range.
6. The circuit operates at room temperature.
7. Resistors are in standard values of 5% tolerance, as indicated in Appendix A of the textbook.
8. The capacitors used to block DC and pass AC should be 200μF or less.
9. The circuit must be capable of delivering an output sinusoid with a peak amplitude of 0.6V without severe clipping (i.e., $V_{o\_peak\text{-}to\text{-}peak\_max} = 1.2V$). You must run a transient simulation to show the output voltage vs. time; as well as the input voltage vs. time to demonstrate that the gain is at least 30.
10. The total DC power consumption of your amplifier should be less than 3mW (calculated using the currents from both power supplies).
11. Do not use ideal DC bias current sources directly connected to transistors in a main amplification stage. Instead, use current mirrors to generate DC bias currents, where two transistors mirror the current from an ideal source. An example is shown on the last page of this project description.
12. Do not use ideal DC voltage sources to generate DC bias voltages directly. Instead, use resistor networks to generate DC bias voltages from the power supply voltages and ground. However, the DC power supply voltages should be from ideal voltage sources.

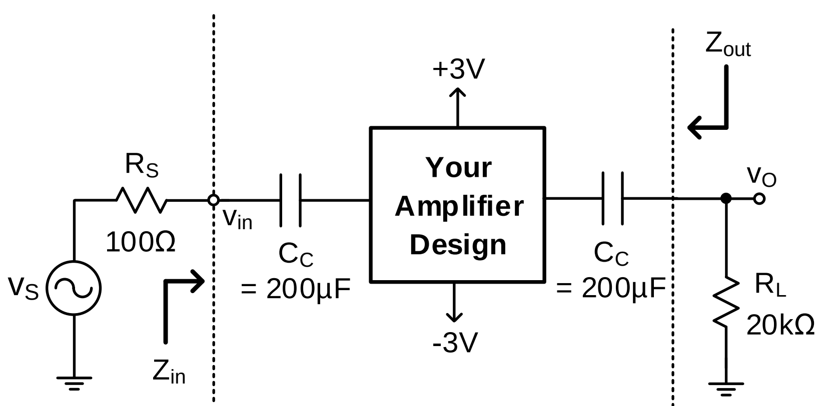
Other comments:

- Within the report, write a design summary that contains your proposed circuit diagram. Provide an analysis (equations, DC analysis, small-signal circuits/analysis, discussion) that explains the reasons why the proposed amplifier meets the design requirements. Analyze the circuit and show your equations as well as calculation results to select the component values that meet the specified requirements. Place boxes around calculation results, or use underlined/bold font. In particular, be sure to address the following:

- Provide an equation for the voltage gain.
- What are the expected input resistance (R_i) and output resistance (R_o) values at midband frequencies based on the hand calculations?

- Simulate the circuit in PSpice (using version 9.1 on COE computers (VLAB) or available for download on the course Blackboard page). Setup the circuit using the Q2N2222 model for the BJT. In the results section, show all important AC and transient voltages and currents, and explain each plot. Label the values of the parameters in the midband region and at the edges of the specified frequency ranges that are given in the above design requirements. In addition, plot the output and input resistances vs. frequency. Include a screenshot of the complete schematic in which the DC voltages and DC currents are displayed. This is particularly important to demonstrate your power consumption. Discuss how the simulation results validate your design choices. Simulation hints:
  - Run an AC simulation with a logarithmic frequency sweep from 1Hz to 100MHz and 20 points per decade. Verify with an AC simulation that your voltage gain agrees with your hand calculation with reasonable error. Submit the plot of the voltage gain (as a ratio, not in dB) vs. frequency in which the midband voltage gain should be labeled.
  - Verify that your simulated input impedance is close to 100Ω by inserting an ideal AC voltage source (without R_S, but with the coupling capacitor C_c) directly at the amplifier input (v_{in}). After running an AC simulation with a frequency sweep, you can plot the absolute value of the ratio of the test voltage source and its AC current at the terminal vs. frequency in order to obtain the plot of $Z_{in} = v_{test}/i_{test}$ vs. frequency. Submit this plot with a label that should show an input impedance magnitude for midband frequencies.
  - With R_S inserted as shown below, select the input voltage amplitude of v_s (a 100kHz sinusoidal signal) that results in the targeted $V_{o\_peak\text{-}to\text{-}peak\_max}$ value with the specified voltage gain. Run a transient simulation with a duration of 50µs and a step size of 0.1µs. Verify with a transient simulation that your output voltage v_o is undistorted. Calculate the voltage gain A_{vs} from the transient simulation to check that it agrees with the gain from the AC simulation.

- Summarize the simulated parameters in a table together with the target specifications. Discuss the following aspects in the report:
  - Do the simulation results match the results from the analysis and calculations? If not, how are they different and what are the reasons for any discrepancies? Change the values of components in your design to meet all specification, or to get as close as you can. Include the optimized simulation results in your report.
  - If you cannot meet all specifications, then explain why it is difficult to meet the requirements.



DC Bias Current Generation Example: