ECE476 DESIGN PROJECT #1 (Spring 2022)

In this first project of the semester, each student in the class must design a common-emitter (CE) amplifier with emitter degeneration.

Learning Objectives:

- Students will learn how to design, simulate, build, and test a CE amplifier with emitter degeneration.

- Students will learn the advantages of using emitter degeneration (negative feedback) when designing amplifiers.

Design Specifications:

- Your circuit must operate from a **single** 15 Volt lab supply. The circuit should function correctly for supply voltages in the range: 14.5 VDC – 15.5 VDC.

- You may only use parts from your ECE476 parts kit. You should use the 2N3904 NPN transistor. Your LTspice schematic should correspond EXACTLY with the circuit you intend to build. For example, if you plan to connect two resistors in series then draw two resistors in series in your schematic.

- The amplifier should, at the minimum, be capable of passing (-3 dB bandwidth) signals between 100 Hz and 100 kHz. The passband gain of the amplifier should be approximately 8 (*i.e*. 18.1 dB) and should be insensitive to changes in transistor parameters, temperature, *etc*.

- The amplifier should be able to drive loads greater than or equal to 10 kΩ.

- Choose appropriate base and collector DC operating voltages. Use a collector current between 1 and 10 mA.

- Hint: In order to desensitize the gain, I recommend that $g_m * R_{E1} >> 1$.

- You should test the circuit using the function generator in the lab. The function generator in the lab has a source impedance of 50 Ω. Apply a 1 Volt (peak), 10 KHz sinewave but attenuate the input signal by a factor of 100 before applying it to the amplifier. This can be accomplished by using a resistive voltage divider.

- You should be AC-coupled at both the input and the output of your amplifier.

- When I use the term “approximately”, I mean in the vicinity 8 i.e. 8 +/- 10% (so about 17 db to 19 dB is perfectly acceptable).

You must do the following:

a) Design the requested circuit (i.e. select appropriate component values).

b) Simulate your design using an electrical simulator (LTspice) and demonstrate successful simulation to the teaching assistant (TA). The simulations should include DC, transient, and ac analyses.

c) Construct and test your circuit. The circuit should be constructed on a solderless breadboard.

d) Demonstrate a fully-functional circuit to the TA during your prescribed lab period.

Evaluation:

1. Demonstration that circuit simulates correctly: *40 points*
2. Demonstration of working circuit: *40 points*
3. Report: *20 points*

   You must write and submit a brief 3 page report (*i.e.* no more than 3 pages of text but you should also include a circuit schematic and all relevant simulation plots) describing your design and its performance. Justify all design decisions and the choice of all component values. The use of equations, given to you in class, should be used to justify your design decisions. Make sure you discuss input impedance and voltage gain and how it compares to an amplifier without emitter degeneration. Discuss how well your simulations and your test results agree.

Week of January 24:
Demonstrate to TA successful simulation of your amplifier by the end of the lab period.

Week of January 31:
Demonstrate to TA a working circuit (by end of lab period).

Tuesday February 08:
Submit design report to instructor at the start of class.