Analog 3 - Amplifiers I

The common emitter transistor amplifier.

Preparations:

Please make sure you *do* prepare for this lab, since it is somewhat demanding it will help that you come prepared. This lab has two parts: in the first you build a **a transistor amplifier**, and the second part uses an **OpAmp**. The two combined parts can take a maximum of four lab periods.

- Lecture Notes.
- Read Practical Electronics:
  - Transistors: Chapter 4.3.1 up to page 442. Read the rest of 4.3 is you are interested.
  - Noise: page 311-313 for noise in a resistor. This can be generalized to other components.
- Make sure you understand the Digilent oscilloscope and Network Analysis function.
  - https://reference.digilentinc.com/learn/instrumentation/tutorials/ad2-network-analyzer/start
- If you want to read more about the transistor amplifier:
  - https://www.electronics-tutorials.ws/amplifier/amp_1.html
  - http://www.electronics-tutorials.ws/amplifier/amp_2.html

Goals:

- To learn about a single transistor amplifier
  - Learn that a simple circuit can be deceptively difficult.
  - Learn to deal with noise and unwanted feedback.
- Learn to make measurements on an actual amplifier circuit.
- Learn about noise in a circuit.
- Create Bode plots of amplifier circuits and know what they mean.

Expectations:

- Keep good and accurate notes in your lab book.
  - Make appropriate conclusions about your circuit.
- Carefully follow these instructions so you end up with the right measurements.

Introduction:

The purpose of this lab is to learn how to build a single transistor amplifier and take good measurements on this amplifier circuit, rather than learning how to *design* transistor amplifiers. With a lot more time, using online resources, you should be able to build any type of amplifier you need.

The first circuit in this two part lab series is called a common emitter amplifier, which is a single transistor amplifier circuit. This type of amplifier has an inverted output from the input, so the phase of the circuit is 180 degrees to the input. These circuits are very common, and can go up to very high frequencies if you use fast transistors, which is why they are often used in radio-frequency (RF) applications. If you ever need to *design* one, you can find plenty resources for them in books and on websites, but you are more likely to build someone else's design, like we are doing here.

Please be careful building the following circuit, it is notoriously finicky. Note that since we will be testing this up to fairly high frequencies (>10 MHz), you want to keep ***all your connections as short as possible***, and ***build everything very compact***. Also make sure all your components make a good connection in the breadboard. The circuit itself should probably not use any wires at all. This circuit can be remarkable sensitive to minor changes in the layout or in the environment. It is finicky to create a good one, but *do not spend too much time on this lab*. If it does not work, come and get help! If you build it on one of the mini proto-boards, you can come back to it after the Opamp lab if you want.

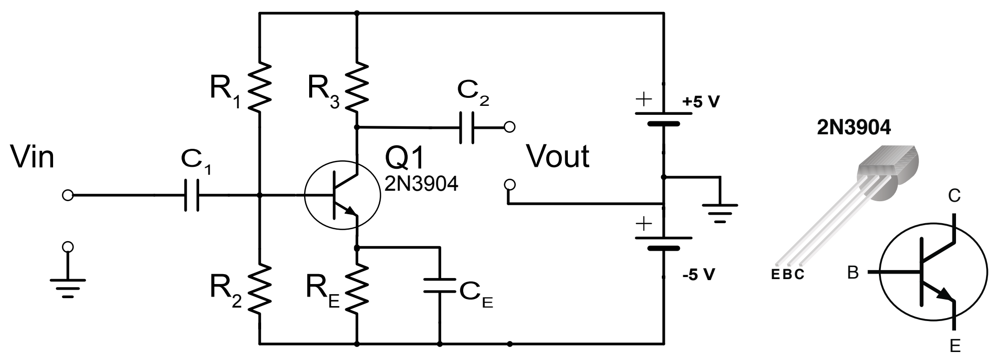

FIGURE 1 COMMON EMITTER AMPLIFIER

A Single Transistor Common-Emitter Amplifier:

The circuit you will be building is shown in the schematic in Figure 1. You will want to power this circuit at 10V, though it should be safe to run it up to 25V. Since we only have the AD to power it, we will run it at 10V by using both the positive and negative power supplies.

Values for the components:

- Q_1 – General Purpose NPN BJT, 2N3904
- C_1, C_2 – 1µF ceramic capacitor.
- C_E – 1µF ceramic capacitor. Initially, leave this out!
- R_1 – 22kΩ ~~27kΩ~~
- R_2 – 4.7kΩ
- R_3 – 1.5kΩ ~~1.2kΩ~~
- R_E – 220Ω

A note about this circuit. C_1 and C_2 form high pass filters. These are needed to allow for the DC (i.e. 0Hz) offset of the input and output signals. The input of you circuit is on a voltage divider to make sure the transistor is properly biased, which puts the base of the transistor at about 1.7V above the -5V. Your input signal will be from -0.01 V to +0.01 V compared to ground. The capacitor C_1 allows for this signal to end up at the transistor base as 1.69V to 1.71V compared to ground. Similarly, you transistor collector will be at about 1.7V, but your output should be centered around 0V. The limited size of these two capacitors limits the amplification of very low frequency signals. If you needed lower signals to be amplified, you could use larger capacitors.

The capacitor labelled C_E strongly increases the amplification of this circuit, up to a bout 70x to 100x for frequencies in the 100kHz - 1MHz range. Because this amplification is large, the circuit may get quite unstable. You should build it without C_E, first, which should give you an amplification of about 5x. So a 100mV amplitude sine wave input at 100kHz gives a 500mV sine wave output. Once this is working stably, you can add C_E, but then reduce the input amplitude to 10mV (maybe 20mV).

Noise:

Noise in electronics circuits are random fluctuations of the signal. Every component will contribute to some extend to noise, some more so than others. Some of this noise is an unavoidable consequence of the quantum mechanical behavior of the electrons. Some of the noise is due to "pickup" or "coupling noise", which means that the components, such as wire, will pickup noise from the environment. In audio equipment, you can hear the noise as a soft hissing in the background. With expensive audio equipment, designers will go out of their way to minimize this noise.

The noise on a signal can be expressed the root-mean-square of the signal V_{rms}, or as the peak-to-peak (V_{pp}) noise, or as a signal-to-noise ratio (SNR). Often noise expressed in decibels (dB), where the ratio of the power (V^2_{rms}) is to some standard measure, such as 1mW, and often the noise is specified as per root-Hz. You need to look up the details of the noise specification for each application (i.e. is you look at the specifications of a commercial amplifier). In this lab, we will simply look at the peak-to-peak (V_{pp}) amplitude of the noise signal since that is easy to read from the scope.

To get a good sense of the noise in any circuit, you first need to establish the baseline noise in the measurement equipment. You do not need to save the data from the scope for *each* step here, but write down your observations in your lab book. It may be nice to have one illustrative picture or data sample for your progress report.

Tasks:

- **Inherent noise in the Scope**:
  - Connect the + and - of the scope to ground.
  - Bring up the Scope window and add the FFT window, which takes a Fast Fourier Transform of your scope signal. You can leave this window for the rest of the lab.
  - Measure the noise. It should be less than 10mV_{pp} but could be more. Measure both the low frequency and the high frequency noise of the scope itself by changing the time base between 500ms/div, 1ms/div and 500ns/div.
  - Notice the yellow band when you are on the low frequency setting. This band indicates the higher frequency noise fluctuations and makes it easy to read the V_{pp}.
  - Look at the FFT of the noise. Are there any peaks? These would correspond to specific frequencies which are more dominant in the noise.
- **Pickup Noise**:
  - Disconnect the + lead of the scope and plug it with a longer wire into one of the unconnected strips of your proto-board. Make sure nothing else is connected to this strip.
  - Repeat the measurements of the previous step.
  - Do you see the 60 Hz signal of the AC main power being picked up? Are there higher frequency ones? (This scope is not fast enough to pick up the WUNH radio station, but a nearby AM station would be visible.)
- **Noise of the Waveform Generator:**
  - Now connect the + of the scope to the W1 signal generator output, and the - to ground. Generate a sine waves with the signal generator of 100Hz, 100kHz and 1MHz, for a small amplitudes (10 mV), and look at the signal on the scope. Your signal should be visible on the FFT as a peak. Note how the peak is wider/narrower depending on the time base setting of the scope. This is a feature of the FFT, where the number of samples per second determines the highest frequency the FFT is sensitive to, and the width of each frequency bin.
  - Now generate a square wave at 10kHz (and if you like some custom waves) and examine the output on the scope and then with "Spectrum", which takes an FFT of your signal. Make sure you test multiple time base settings and notice how the base frequency of 10 kHz repeats at odd multiples. If you set the symmetry to something other than 50%, you will also see the even multiples.

Amplifiers:

An amplifier will have a gain, but this gain often depends on the frequency of the input signal. A Bode plot will give you a clear picture of the frequency response of an amplifier. For a perfect amplifier, that is all it does, it will faithfully amplify the input signal with some fixed gain. An actual amplifier will have a number of distortions which will change the signal in subtle, and not so subtle, ways.

Tasks1, Noise:

- Carefully build the circuit of figure 1. Since it requires +10V, you need both the +5V and -5V of the AD.
- Connect the wave form generator 1 (W1) output to the input of your circuit. Connect scope channel 1 to the input of your circuit as well and scope channel 2 to the output of your circuit. Set the wave generator to zero volt output. You are effectively grounding the input. Measure the noise of the circuit by *comparing the two scope channels*. Note there is noise on the input as well.
- Is the noise of the amplifier, proportional to the signal, much larger than that of the scope?
  - You expect the noise to be a bit larger than just the amplified input noise.
  - If you have very large amounts of noise, you may have an oscillating circuit or other issues. Try to see if you

can reduce the noise, or get help.

- Take an FFT spectrum (using the AD) of both channels and compare the noise spectrum.

Tasks2, Small Signal Amplification:

- Build the transistor amplifier. Test the amplifier without C_E, then insert C_E and take measurements.
- Now set the waveform generator to a very small sine signal, 10 to 20 mV, and the frequency of 100kHz. Compare the input wave form and the output waveform. The output is flipped over because the circuit inverts the signal. You can "unflip" it by exchanging the + and – input of channel 2 (or you can use a Math channel to compute -V_2). This allows for a good shape comparison.
  - How much is the amplification of the signal, i.e. V_{out}/V_{in} ? This is the gain of the amplifier.
  - Does the circuit distort the signal? How much? I.e. does the amplifier change the shape of the input.
  - If you change to a triangle input wave, does the distortion get worse? With a triangle wave it is easier to see if the amplification is not linear.
- Repeat these steps for a 1kHz input signal. You may need to increase the input signal size.
- Usually distortions get larger for larger input signals. Try it. Increase the input signal until you observe stronger output distortions, such as "clipping" of the signal.
- Record the maximum input signal at 100kHz that gives the smallest distortion in the output.
- Generate a Bode plot and a gain plot for this circuit for 10Hz to 10MHz, with at least 500 measurements.
  - You need to set the **WaveGen Amplitude** (top right on the panel) to **the small value (10mV?) you recorded in the previous step.**
  - You also want to set a -180 degree offset to the phase, so it plots nicer.
  - Save this plot in dB, and the phase, for your lab report.

Tasks3, Limit the Gain:

You circuit has two problems. First, it amplifies the high frequencies more than the low frequencies. The "cause" of this is the C_E capacitor. The gain of the circuit is set by the ratio of R_3 to (R_E+R_t) where R_t is the transistor internal resistance (small). The gain for very low frequencies (100Hz) is around 5x. For high frequencies, C_E effectively short cuts R_E, so the gain get large, up to about 200x for signals of 100kHz. We need to limit the maximum gain and allow for lower frequencies. The latter is done by picking a much larger capacitor, 100µF. The former is done with a series resistor R_{E2} of 100Ω with this capacitor. The circuit is now as in figure 2. This amplifier should have a gain of about 15x (23dB) and a frequency response down to about 100Hz. This circuit is now also more stable.

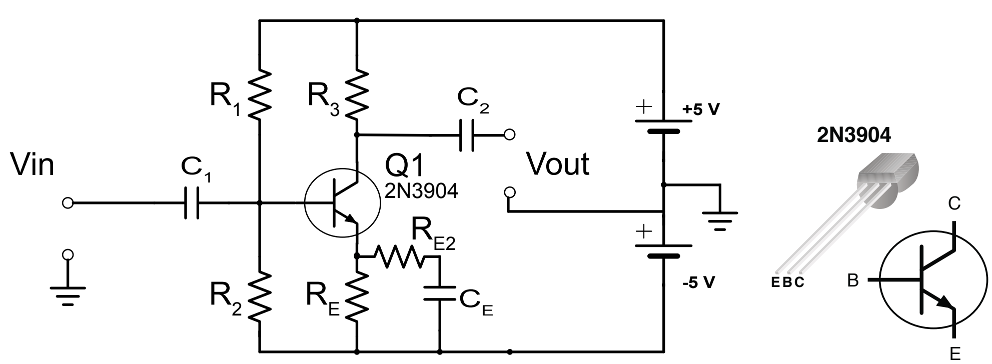

FIGURE 2 COMMON EMITTER AMPLIFIER WITH LIMITED GAIN.

- Following the steps you did before in Task2 with this new circuit, investigating the gain and distortion.
- Save the new plots for your lab report.
- Comment on the differences you observe. Is this a "better" amplifier?

A final note: this is not the "best" circuit for amplifying signals, but it *is* the simplest. You can add more transistor stages to get more amplification, and to get the amplifier to be more uniform. You can also add a "push-pull" dual transistor output stage to get a bigger peak to peak output signal. There are a lot of variations on transistor amplifiers. We will not worry about them, and instead move on to Operational Amplifiers, which are much simpler to use.

For Progress Report:

Required Parts in the report:

- An accurate copy of the schematic **that you build** in the lab, with a very brief description describing *your* understanding of what it does.
- A description of the measurement of the noise in the circuit: how did you measure this, what were the results. This can be a brief paragraph, mentioning the peak-to-peak noise and perhaps the characteristic of the noise. Put your data in a table.
- A description of the measurement of the distortion of the circuit:
  - How did you measure this?
    - Description.
    - A recorded signal that illustrates the distortion you observed would be nice but is not essential.
  - What is the maximum input signal (peak-to-peak) that can be amplified more or less un-distorted? At what frequencies did you measure this?
- A description of the procedure for Bode plots you created.
- Plot the Bode plots for the two circuits (if you did both).
  - If you made multiple Bode plots that you saved, the ideal plot would have the Bode plots for the different input setting super imposed onto one plot, each with a separate color. You will find an example Python script in the Python directory on GitHub to help you out.
- A description of the results.
- What you learned, who helped you, what difficulties did you encounter.

Optional extras, bonus points:

If your lab just did not work, here is a way to make up the difference.

- Repeat the lab by building the simulation of the circuit in partsim.
  - See the almost complete example at: http://www.partsim.com/simulator/#73815.
    (Note that in the simulation V_{in} is 0.01 V_{pp} so it is at -20dB, and V_{out} goes to 25dB, so there is still 45dB of amplification!)
    - If partsim is again not working. Contact your professor for using MultiSim on one of the UNH PCs.
  - Modify and play with the simulation a bit as you did with the experimentation part of this lab, and report on your results.
  - Changing R_E and C_E should change the amplification.
  - Do your measurements from the lab correspond with the simulation?
  - Describe how they differ, how they are similar.