

ECE-205 Lab 4

System Linearity

Overview

For many engineering systems, we can model the system as linear only over a specified region of operation. In this lab you will be constructing the simple common emitter BJT amplifier shown in Figure 1 below, and determining the range of input signals for which we can model the circuit as having linear behavior. Note that this is not a well designed amplifier, but is easy for us to build and learn from.

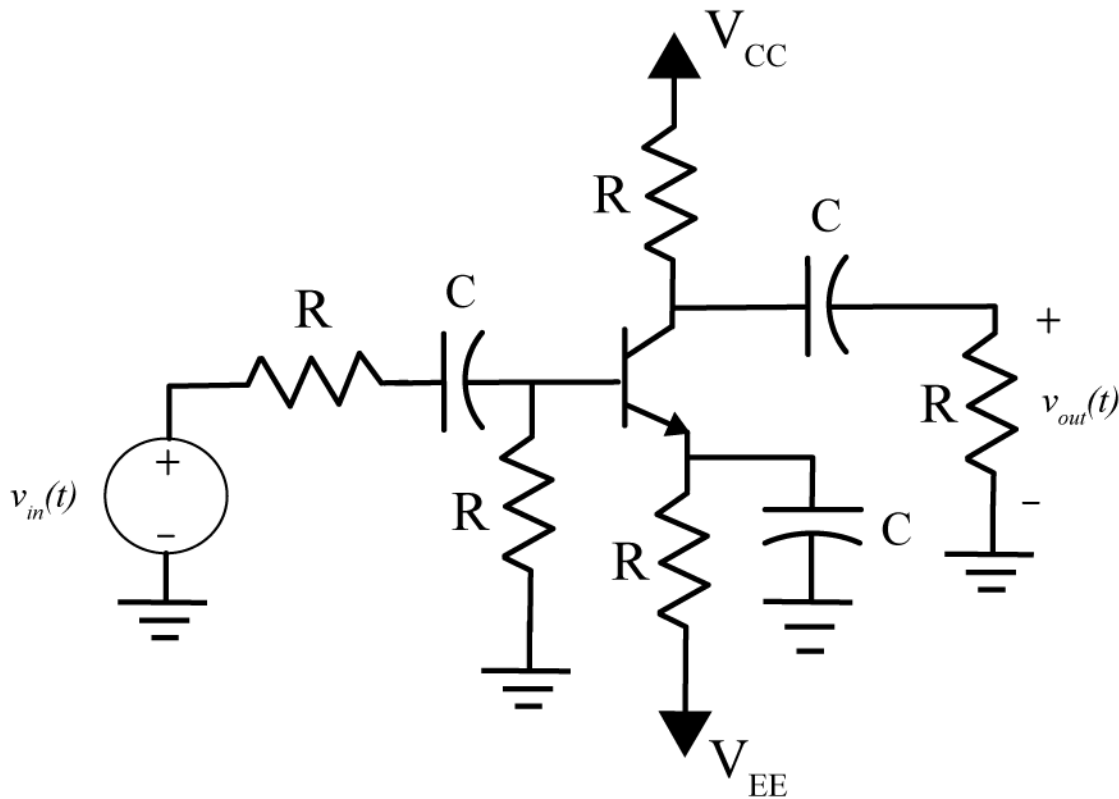


Figure 1: Simple common emitter amplifier.

PART 1: Building a Trigger Signal, and Setting Voltage Rails.

The first thing we want to do is set the ground and the two power rails. First connect one of the grounds (a down arrow) to the blue “rail” and then use another wire to connect the top and bottom blue rails together. It is better to do this somewhere away from the board where the USB and power connectors are. Next connect a wire from VP+ to the top red rail (this will be our V_{CC}) and connect a wire from VP- to the bottom red rail (this will be our V_{EE}). Both V_{CC} and V_{EE} are just to provide the necessary voltages to our transistor, *they are not the input signals*. Figure 2 shows how I connected these. Next start Waveforms (the Digilent software), select the voltage icon, then set Positive Supply VP+ to 9 volts and Negative Supply VP- to -9 volts. Set the maximum currents for the power supplies to 100 and -100 mA. Finally, select Power to be ON.

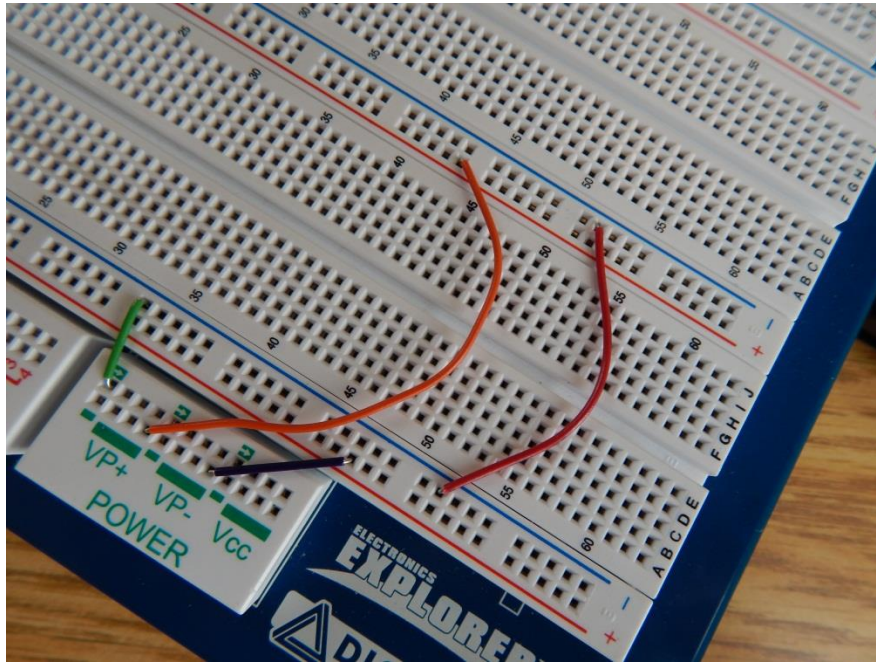


Figure 2. Connection of ground and power rails.

The next thing we are going to build is a very simple resistive circuit to be sure we have something to trigger off of if things go badly. This also helps us locate DC signals which won't cause the system to trigger. Run a wire from the signal generator AGW1 to somewhere on the board, and then connect a resistor from this wire to ground. Set scope channel DC1 to the positive end of the resistor. This is shown in Figure 3.

Finally, set the signal generator to a square wave with a frequency of 100 Hz and an amplitude of 1 volt (2 volts peak to peak). Set the scope to trigger off of channel 1 and make sure you are triggering before you go on. At this point you should use scope channel DC2 to be sure your voltage rails are at approximately ± 9 volts. Be sure you are triggering off of channel DC1. You may also have to change the scaling on channel DC2. Be sure the Explorer Board is turned on!

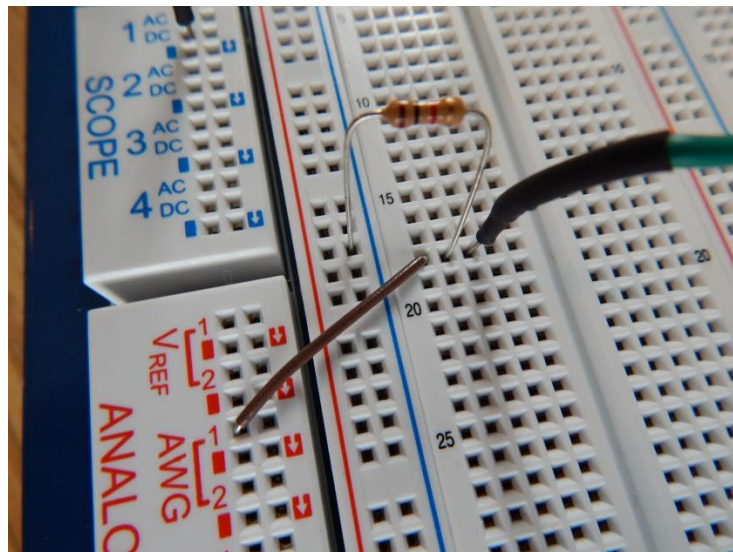


Figure 3. Setting up a simple trigger signal.

PART 2 : Building the Circuit

We will be building the circuit in two stages, do not try to build this all at once. All of the resistors are $1\text{ k}\Omega$ resistors, all capacitors are electrolytic $100\mu\text{f}$ capacitors and the BJT is a 2N-2222A-836. The electrolytic capacitors *must be connected with the correct polarity*, so be careful and read the instructions.

1) The npn BJT is represented by the symbol shown in Figure 4, and it has three terminals. If you hold the BJT with the flat part to the right, then the collector is on top, the base is in the middle, and the emitter is on the bottom.

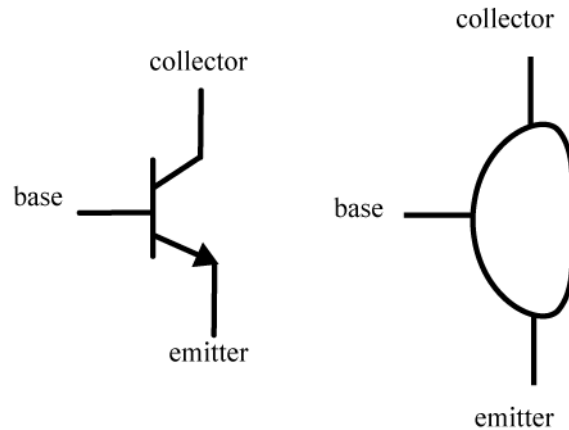


Figure 4. Circuit symbol for an npn BJT (left) and physical device (right).

2) Now we will build the circuit subsystem shown in Figure 5. Turn off the power while wiring, and then turn the power back on when you want to make measurements. Note that the collector is connected to the +9 volt source V_{CC} , the emitter is connected to the -9 volt source V_{EE} , the base is connected through a resistor to ground, and the collector is also connected through an electrolytic capacitor (note the negative sign!) in series with a resistor to ground. It is probably easiest to have the transistor straddle the middle of the breadboard. Note that the resistor connected to V_{CC} is a variable resistor (a potentiometer). Be sure to only connect one of the outer legs and the middle leg to use this as a potentiometer.

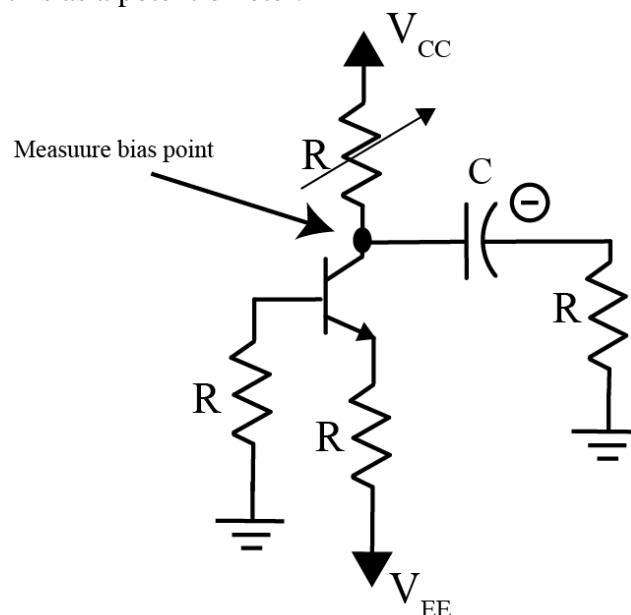


Figure 5: Biasing the transistor circuit.

3) Connect a probe from scope channel DC2 to the measurement point as shown in Figure 5. Turn the power on, and measure the signal at the bias point. Be sure you are triggering off of channel DC1. Adjust the potentiometer so the bias point is approximately 4 volts. Do not go on until you get this correct!

4) Turn the power off and finish building the circuit as shown in Figure 6. Not that the polarity of all of the capacitors, and that everything we are now connecting is connected to ground (not one of the voltage sources). The input signal will be the signal coming from AWG2. We will be measuring our output as the voltage across the load resistor.

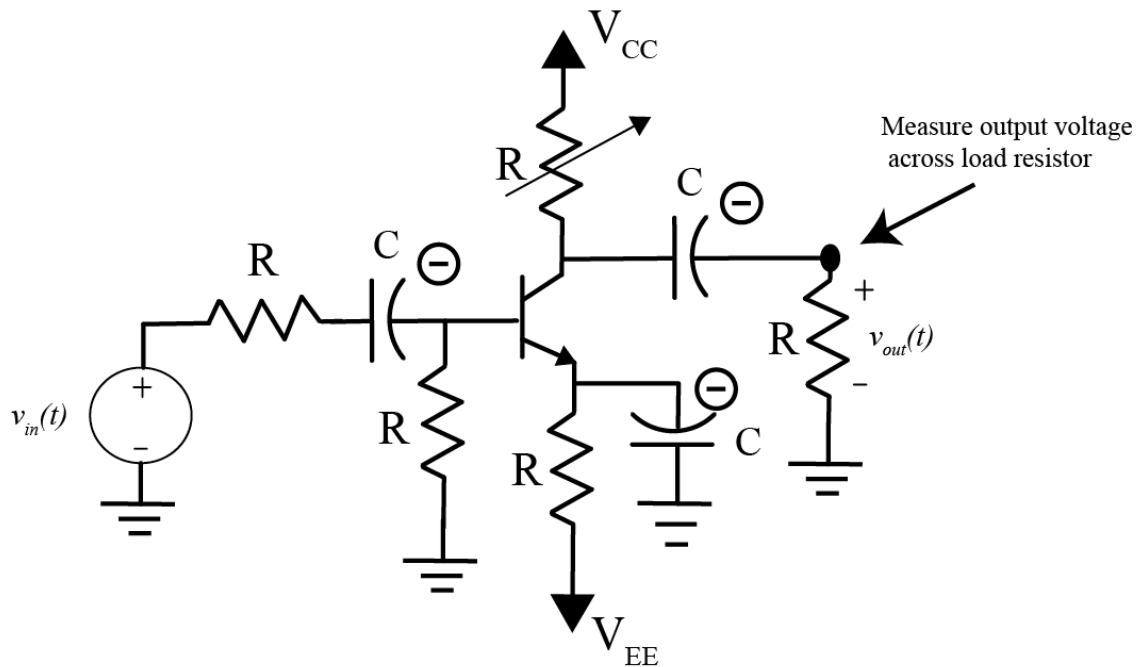


Figure 6. Final circuit. The input is AWG2 and the output is the voltage across the load resistor.

5) Turn the power back on and set function generator AWG2 (use **Select Channels** in the toolbar) to a 100 Hz symmetric sine wave with an amplitude of 0.1 volt. Connect scope channel DC2 to measure the voltage across the load resistor, this is our output signal.

7) Use the cursors to measure the *peak positive amplitude* output signal (**do not** measure peak-to-peak, use the measurement **high** setting). The amplitude of the input signal is 0.1 volt, and the gain is the ratio of the amplitude of the output signal divided by the amplitude of the input signal. The gain for your circuit should be between 1.5 and 4. If it is not, check your circuit and then ask for help. Now we are ready to make our measurements.

PART 3: Measuring and Plotting the Input and Output Amplitudes

Our circuit can only be modeled as a linear system for a limited range of input signals. *One important property of linear systems is that if the input signal is a sine wave, then the output signal must also be a sine wave.* The magnitude and phase of the output sine wave can be different than the input, but it must be sinusoidal or the system is not a linear system. We will measure the input and output amplitudes as the input amplitude changes. We will assume the input signal is a sine wave with a frequency of 100 Hz for all of the following measurements.

1) Measure the peak positive output voltage amplitude for each of the following input voltage amplitudes: 100 mV, 200 mV, 300 mV, 400 mV, 500 mV, 600 mV, 700 mV, 800 mV, 900 mV, 1000 mV, 1100 mV, 1300 mV, 1500 mV, 1700 mV, 1900 mV. Be sure to measure only the *peak positive amplitudes* of the output signal, **do not** measure peak-to-peak amplitudes. You will need to enter your measurements into Matlab (see step 3 below) so you may want to do that as you are measuring. *Change the size of the oscilloscope view screen as necessary to get as much of each signal in the window (while still seeing the entire signal).* **DO NOT DISCONNECT YOUR CIRCUIT UNTIL THE END OF THE LAB**

2) Start Matlab and open a new m-file

3) Enter the input amplitudes and output amplitudes into two arrays in Matlab, and plot the output versus the input amplitudes. To enter values in an array in Matlab, type something like

```
Vin = [1.0 2.0 3.0 4.0]; % not the array used to make the plot below
```

```
Vout = [5.5 6.6 7.7 8.8]; % this creates a row vector
```

You should put a space between array entries. To plot the arrays type

```
plot(Vin, Vout, 'o'); grid; xlabel('V_{in} (mV)'); ylabel('V_{out} (mV)');
```

Your graph should look something like that shown in Figure 7 below:

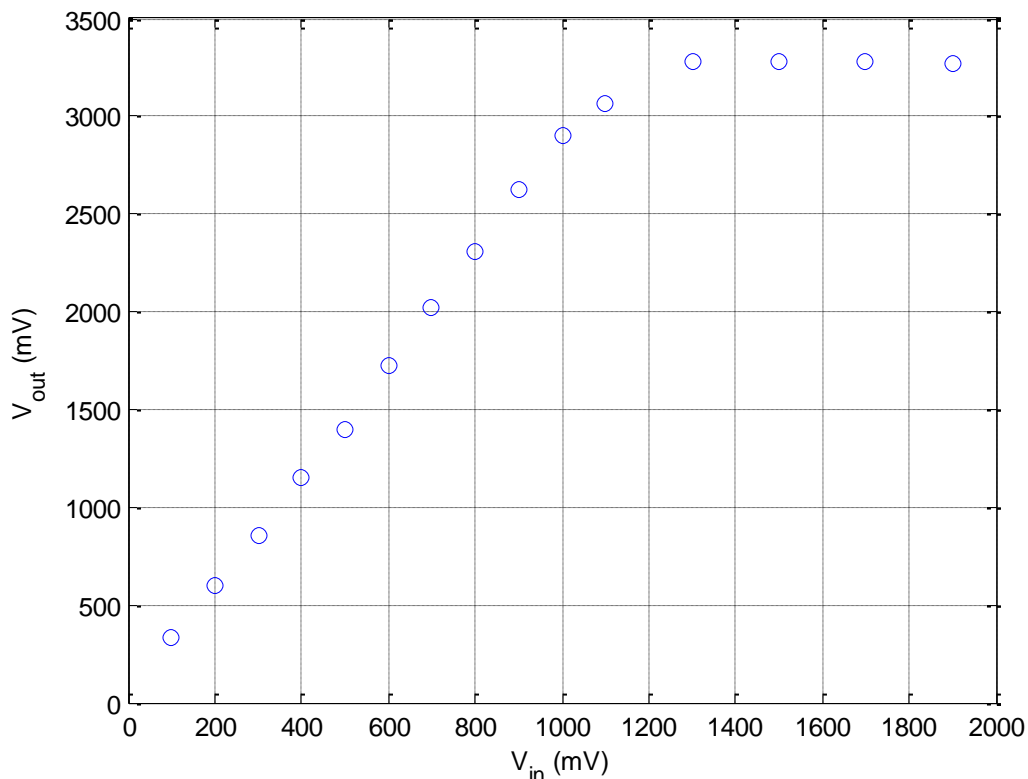


Figure 7. Measured input/output data.

4) We want to fit a least squares straight line between the points that show a linear input/output relationship for the smaller input amplitudes. We can use the Matlab command **polyfit** to do this. Let's assume we have arrays **Vin** and **Vout** that contain the input and corresponding output measurements. Then if we want a least squares line that fits the first **N data** points we will type the following into Matlab:

```
N = 5; % just a guess, yours may be different
p = polyfit(Vin(1:N),Vout(1:N),1);
m = p(1); b = p(2);
est_output = m*Vin+b;
```

Enter these commands into Matlab and generate a plot like the one shown in Figure 8. Vary the parameter **N** to include as many points as you think reasonably fit a straight line. **Include this plot in your memo.** Be sure to use the Matlab **Copy Figure** command (click on **Edit**, do not copy the entire window, only the figure). Be sure to include a figure number and caption. *Include the value of the gain of the circuit in the linear portion as part of your caption. The gain is the value of m in your straight-line fit. Also include the bias point you used.*

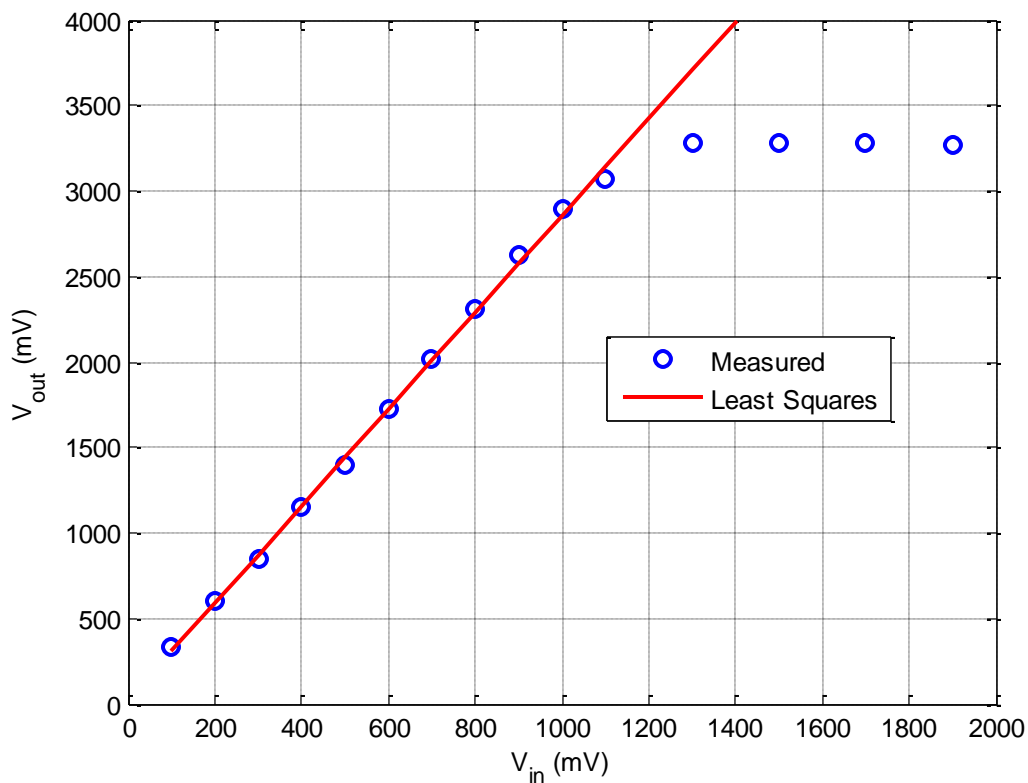


Figure 8: Measured data and least squares fit for linear response.

6) Save one screenshot showing a linear behavior of the system, and one screenshot showing nonlinear behavior of the system. **Include these figures in your memo.** Be sure to include figure numbers and captions.

7) Now disconnect the input (AWG2) from your circuit, and adjust the variable resistor so the bias point is nearly 0 volts. Note that you need to change where you are measuring the output of your circuit now, but *you do not need to change anything else*. Once the bias point is set, **reconnect** the input (AWG2) and **change your output measurement** to the voltage across the load resistor, as before.

8) Repeat parts 1-6 above, and add to your Matlab script (you can comment out your old script if you want, but do not remove it.)

9) Include your m-file in your e-mail since this contains your data.

10) e-mail me your memo, and you are done. Be sure to try and make your memo look like a memo. It should be very short, and include and suggestions for improvement.

Your memo should contain 4 graphs: two Matlabs graph and two screen captures. You should e-mail your memo to me. Your e-mail should also include your Matlab m-file as an attachment.