

Lab 3: Biasing BJTs and looking at AC Gain

Introduction

"One who makes no mistakes, never makes anything."

-English proverb

Overview

You will design a BJT amplifier setting its DC quiescent point and preparing it to be used as an AC amplifier. In this lab, you may or may not have time to actually implement the circuit this week. If you do have time, it would be good to start building it.

Lab procedure

1. Using the schematic below, design an amplifier using a 2N4401 BJT, a collector resistor of 500 ohms to 10k ohms, a V_{cc} between 8 and 15 volts and a collector current between 0.5mA and 15mA. You get to choose, you're the engineer. Don't simply copy the values from lab2. Your amplifier should set the quiescent collector voltage to about $0.5V_{cc}$. Show all calculations in your lab notebook. Take care to not exceed the transistor power dissipation or maximum collector current. Draw the schematic of your design including all resistor values. Check the DC quiescent point by running a ngspice simulation. Adjust your values a bit if necessary to meet the given criterion. Include the netlist for your simulation as well as the results in your lab notebook.

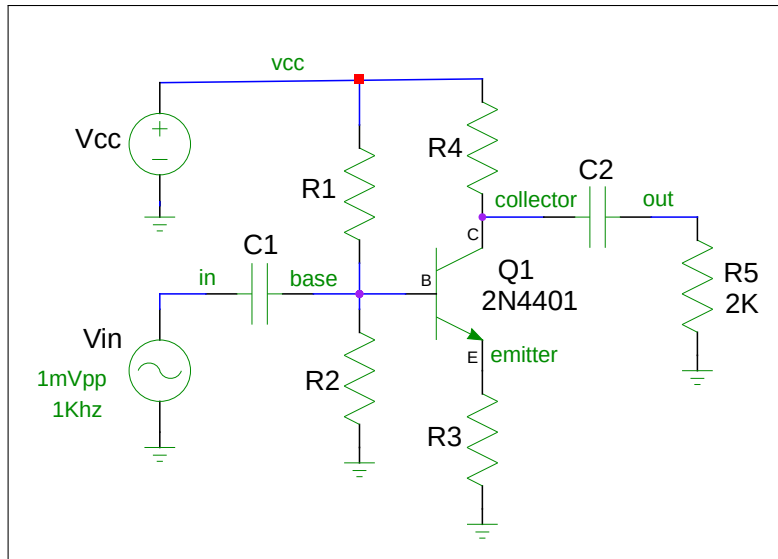


Figure 1: Amplifier Schematic

2. Download the spice model file [2n4401.mod](#) to a working area for lab3. Modifying your spice file from lab2, create a new spice file that now adds C1, C2, R5 and a 1mvpp, 1KHz AC signal source. C1 and C2 can temporarily be 10uF. We will compute the exact values later. Your new schematic and netlist should match.
3. Recheck the BJT's DC operating point by running the op simulation on this new, modified design. Your hand calculated values should roughly match ngspice. Print the voltages at the base, emitter and collector of the BJT from ngspice and record these in your lab notebook.
4. If your DC operating point simulation runs cleanly, measure the input and output peak-to-peak voltages with ngspice and compute the voltage gain. You will have to use the tran and plot simulation directives to allow the transient simulation to be run. A synopsis of the commands is shown below:

```
*transient analysis
* tran tstep, tstop, [tstart]
tran 20u 5m ; transient (time domain) analysis
* plot every 20uS for 5mS
plot v(in) v(out) ; plot AC voltages
```

Paste the plot created into your lab notebook. It should show both input and output waveforms. Did you get an estimated gain close to $\frac{R_c}{R_e}$? Make comments in your lab notebook comparing the actual gain versus the estimated gain.

5. If time permits, build the circuit you have thus far and check the DC operating point. If not, you will be building the circuit next week.

Your lab notebook should have the following:

- Any calculations you used in determining your resistor values
- The spice file that you used to simulate the final circuit with capacitors included
- The spice plot of input and output signals
- The following data:
 - voltages at base, emitter, and collector, collector current, AC small signal gain