Lab 4: BJTs - Current sources/sinks/mirrors

Introduction

"When you can measure what you are speaking about, and express it in numbers, you know something about it; but when you cannot measure it, when you cannot express it in numbers, your knowledge is of a meager and unsatisfactory kind." - Lord Kelvin

Lab procedure - current mirror

1. Draw the schematic of an NPN BJT current mirror shown below in your lab notebook.

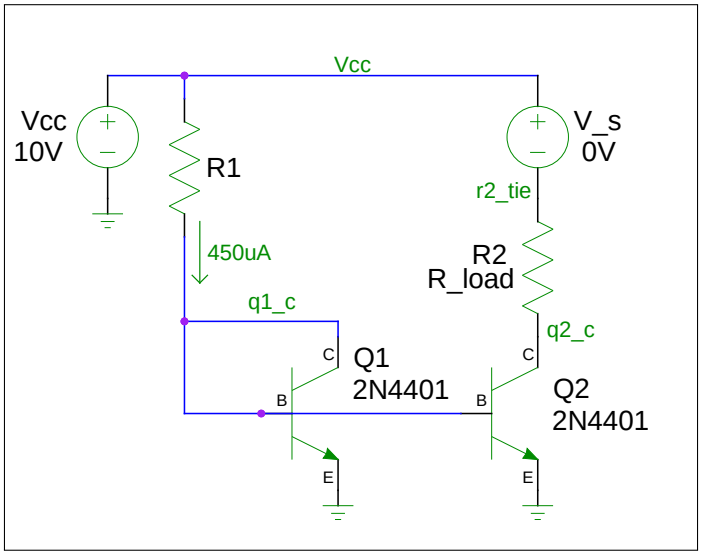

Figure 1: NPN Current mirror

2. With $V_{cc} = 10V$, calculate the value of R1 that sets the current through Q1 to be about 450uA. Assume that for Q1 and Q2, $\beta \geq 100$ and $I_b = 0$ for Q2. This is a hand calculation. Update your schematic with the calculated value of R1. You will need to choose a standard value for R1.

3. With $R_{load} = 10$ ohms, run a Ngspice simulation of the circuit and check the programming and mirror currents (the current through Q1,Q2). Both should be $\approx 450uA \pm 15\%$. You will need the spice model file available at: 2n4401.mod. Record Q1's V_{be}, the programming current and the mirror current.

4. Change R_{load} to 10, 470, 1K and then 10K ohms. Measure the mirror current for all cases. Is it still close to the same value? Note your results.

5. Build the current mirror on the protoboard. The schematic is shown below.

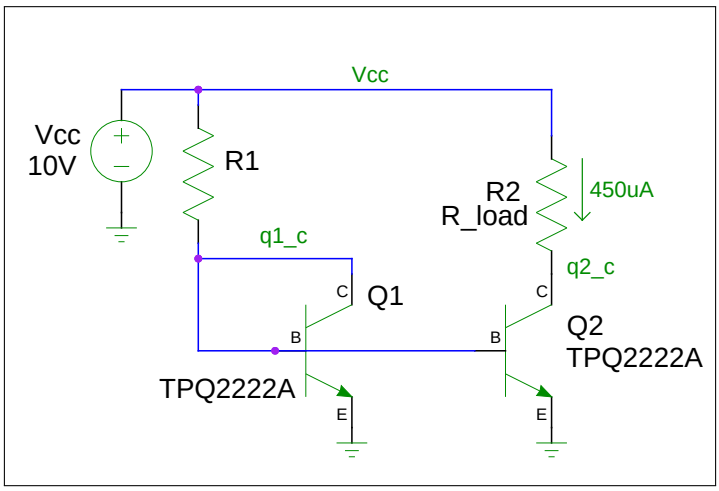

Figure 2: NPN Current mirror prototype circuit

You will use one TPQ2222A IC for this step. This IC has a set of 4, NPN BJTs. Their V_{be}'s and β's will be nearly identical since they are on the same die. They will also track with temperature. You can use any of the four transistors.

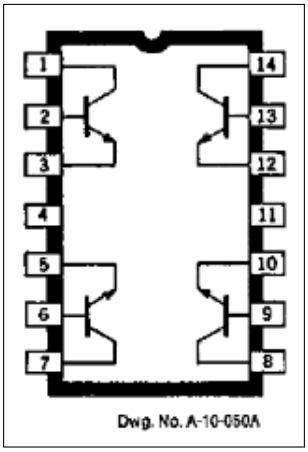

Figure 3: TPQ2222A pin out (top view)

6. You can support the IC above the ground plane by keeping pins 4 and 11 bent down and out like ”legs”. as shown below. This will make for easy soldering. Don't bend the pin (pin 7) shown in Figure 4!

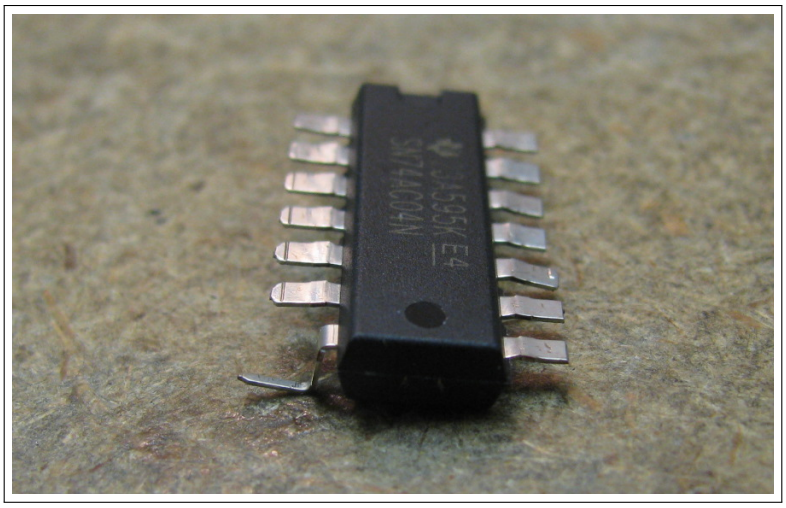

Figure 4: Bending IC legs dead-bug style

7. Test your prototype circuit as before with 10, 470, 1K and 10K resistors for R_{load}. Check the mirror current for each resistor and compare with your spice simulation. Note the programming and mirror currents in your lab notebook. Are they close? They should be!

8. What if you raised R_{load} to 100K ohms? Speculate on what would happen. Confirm your suspicions with simulation or prototyping and report your findings.

Lab procedure part 2 - LED driver

9. Draw the schematic for the LED driver circuit in your lab notebook and simulate with Ngspice with $V_{cc} = 10V$. For the simulation, we are using an LED that we can find a spice model for from Cree. You will need the LED model file xwawht.mod and the spice file w_led_ccs.sp. Using Ngspice, determine:
   1. What is the current flowing through the LEDs?
   2. How far can you increase V_{cc} without Q2 dissipating more than 0.5W?

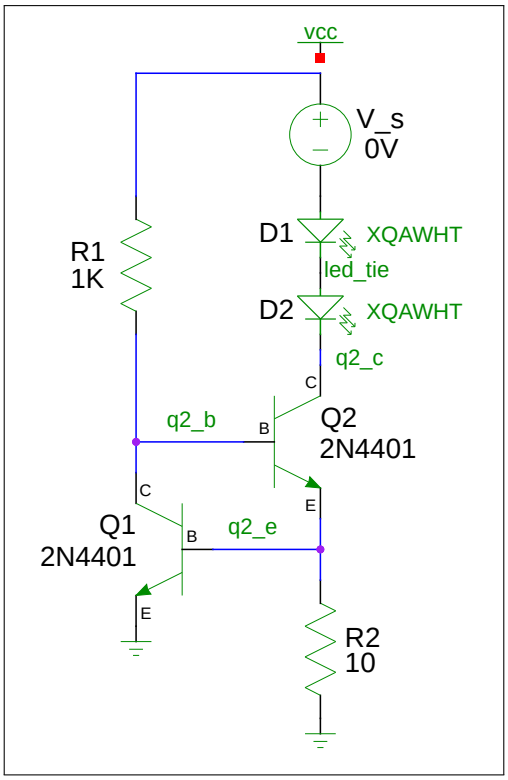

Figure 5: LED driver circuit for simulation

10. For our prototype circuit, we'll use another cheaper LED, a RL5-W10015. You can find its specifications at: RL5-W10015. What is the maximum current that should pass through the LEDs? Does our Ngspice simulation confirm we are operating the RL5-W10015 LEDs at less than this value?

11. Build the LED driver circuit shown below on your protoboard. Note that R2 is different. Use the RL5-W10015 white LEDs supplied in lab for D1, D2. How do you determine the LED's anode and cathode? Measure and record the current through the LEDs with your DMM. Is it close to your simulation? With a 10V V_{cc} is the current well below the absolute maximums for diode current and transistor dissipation?

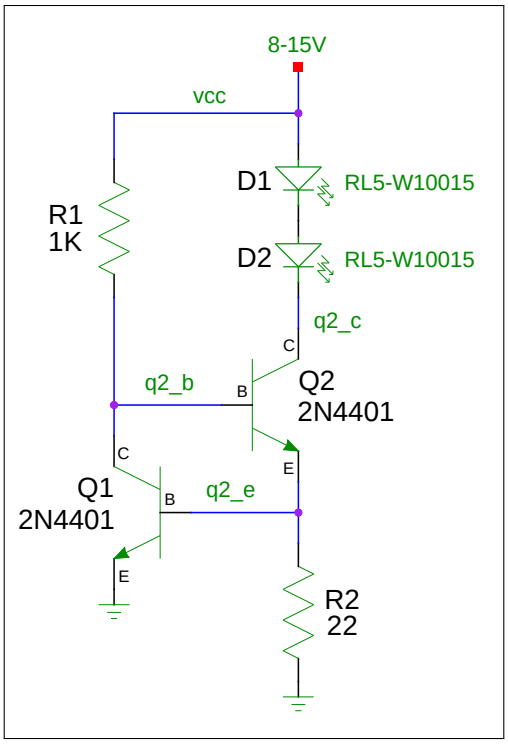

Figure 6: LED driver circuit for prototyping

12. Vary the voltage to the circuit from about 8 volts to 15 volts. Does the brightness change as you vary the voltage? It should not if the LED current remains constant? How do you think this circuit works? Explain to the best of your ability.

Your lab notebook should have the following:

- The spice file and schematic that you used to simulate the current mirror
- Schematic of the hand-built current mirror including node names
- All calculations you used in setting the programming current
- The spice file and schematic that you used to simulate LED driver
- Schematic of the hand-built LED driver
- All calculations used in determining the maximum V_{cc} for the LED driver circuit
- Measurements taken of the LED prototype circuit.