

University of Pennsylvania
Department of Electrical and Systems Engineering
ESE 319
Laboratory Experiment – Current Source Bias BJT Amplifier

1. **Introduction.** This experiment is designed to demonstrate the operation of a BJT transistor amplifier using a current mirror to fix the amplifier transistor emitter current. It also is designed to give you more practice (and confidence) in using many of the design short-cuts mentioned in class and used to design the first BJT amplifier.
2. **Pre-lab homework.** The schematic for a classical current mirror amplifier is shown below. There is a slight difference from the one in Sedra and Smith – in our case, the emitter resistor is kept out of the dc path and thus does not interact with the bias voltage levels of the base and emitter.

Using every analysis shortcut that you have learned, do a quick pencil and paper design of an amplifier to meet the following conditions:

- a. The collector bias current of the amplifier shall be close to 1 mA.
- b. The dc (bias) value of the collector voltage to ground should be about 6 V.
- c. The amplifier gain shall be close to 20 V/V. (26 dB)
- d. R_B can be considerably larger than the input impedance of the transistor (v_{bg}/i_b). Why?
- e. Estimate the base bias voltage to ground (caused by base bias current flowing through R_B).
- f. The lower 3 dB frequency of the amplifier shall be at or below 20 Hz. Suggestion: select $C1$ so that the emitter circuit has a 20 Hz, or lower break frequency. Practical values of $C2$ can be chosen to have a much lower break frequency. (Why?)

HINT: The pole determined by a given capacitance, say $C1$, may be approximated as $f_{C1} \approx \frac{1}{2\pi C1 R_{C1}}$,

where R_{C1} is the resistance seen by capacitance $C1$.

- g. In all cases, adjust your design to use RCA Lab standard values for the circuit resistors and capacitors and then recalculate your bias and gain values. (We may not have all of the values in the table below. When building the circuit in the lab, use the closest one available.)
- h. Can you eliminate the capacitor $C2$ and connect the signal source directly to the base? What will happen if you do?

All students shall complete this design and bring their pencil and paper design calculations, along with a Multisim simulation, when they come to lab to perform this experiment. These materials are to be turned as part of the lab report. When you perform the simulation, do so with and without the parallel RC model of the 'scope input impedance connected from collector to ground.

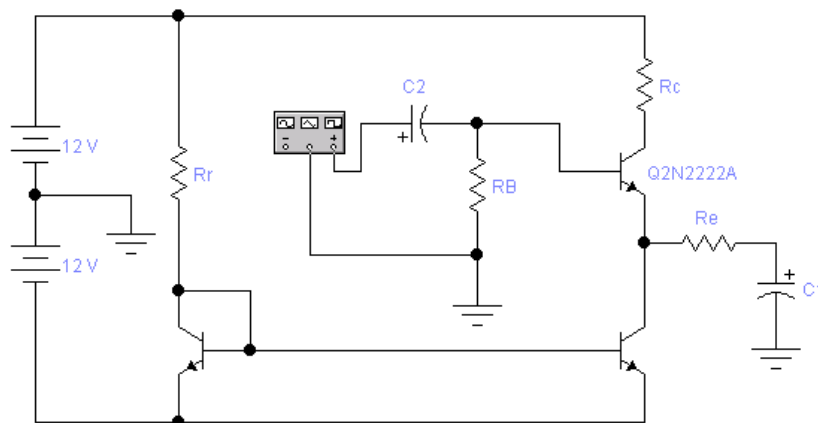


Fig. 1 Current mirror bias amplifier

This design should take only a few minutes. It should not depend on the transistor characteristics. With the possible exception of the capacitor selection, not even a calculator should be needed to complete it. Don't compute values to 3 decimal places when you are working in a 1 or 2 decimal place environment!

3. **Experimental work.** Build the amplifier and verify that it is correctly wired by measuring the bias voltages at the collector, base, and emitter. Then, determine its frequency response using a suitable input voltage source. Measure the input-to-ground and collector-to-ground voltages and calculate the amplifier gain from its low frequency 3 dB point to its high one. To form the current mirror, use either transistor pairs Q1 and Q2, or Q3 and Q4 on a [CA 3046](#) integrated circuit. Remember, for a current mirror to work satisfactorily, the mirror transistors need to be well matched and to be operating at the same temperature. This is why we use the transistor array, instead of two discrete transistors.

EVEN THOUGH Q5 IS NOT PART OF THE CIRCUIT, PIN 13 ON THE IC STILL MUST BE CONNECTED TO -12 VOLTS, THE MOST NEGATIVE VOLTAGE ON THE CIRCUIT!

4. **Conclusions and Discussion.** Does your design gain and your experimental frequency response results agree with those from the simulations? If not, where do the results differ? Try to explain why they differ. What is ignored in the pencil and paper design? What is included in *Multisim* that is not present in the pencil and paper analysis? What is missing from the *Multisim* circuit simulation?

Kenneth R. Laker 05 Oct 2009