

UNIVERSITY OF UTAH
ELECTRICAL ENGINEERING DEPARTMENT

ECE 3110

LAB EXPERIMENT NO. 4
CLASS AB POWER OUTPUT STAGE

Objective: In this laboratory exercise you will build and characterize a class AB power output stage. This stage uses a complementary pair of bipolar transistors in the usual "push-pull" configuration. The class AB bias voltage is supplied by a V_{BE} multiplier circuit, which in turn is supplied by a constant current from a current mirror. You are also to simulate this circuit using SPICE and to compare your approximate hand calculations and measurements with the results of the simulation. Refer to the textbook Sections 14.4-14.5 for the design equations for the experiment.

Pre-Lab

1. Read and work through the examples in Sections 14.4-14.5 of the text. This experiment is based on this material.
2. Design your circuit, Fig. 1, i.e., calculate the required value of R_{ref} so that the current mirror will supply a bias current to the V_{BE} multiplier that is about three times the maximum base current expected for Q_1 . Assume that $\beta = 100$.

Note that Q_1 supplies all of the load current when the output is high. The smallest value of R_L to be used in this experiment is 100Ω .

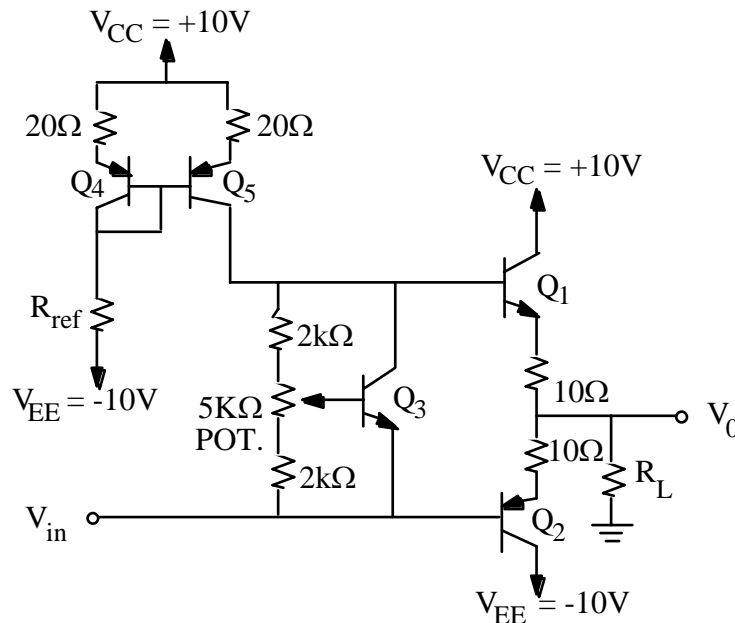


Fig. 1.

Required Parts

- a. A complementary pair of bipolar transistors for Q_1 and Q_2 each of which is packaged in a metal can to allow for higher heat dissipation. One such pair is the 2N2222A and the 2N2907 in TO-18 metal cans which are available from the stock room.
- b. Heat sinks for Q_1 and Q_2 , to increase the allowable heat dissipation in these transistors.
- c. One general purpose, small signal NPN for use in the V_{BE} multiplier. A plastic packaged 2N2222 or 2N3904 is adequate.
- d. Two general purpose, small signal, PNP transistors to form the current mirror. Ideally, these are a matched pair. Use two of the same type, manufacturer, and date code. The stock room has 2N3906 devices in plastic packages.
- e. One 5K pot for adjustment of the V_{BE} multiplier circuit. Use a pot that can be easily and smoothly adjusted over the entire 5 k Ω range.
- f. Assorted resistors as needed to complete the circuit.

Circuit Construction

Any mistake in the construction of this circuit can lead to excess current and very hot transistors. Be careful when touching transistors as they may become *hot!* Set the current limit on your power supply to around 150 mA so that an accidental short does not burn out your transistors or burn your fingers.

Build the circuit as shown using a 1 K Ω resistor for the load, R_L . To begin, build the biasing circuitry (everything except Q_1 , Q_2 , the 10 Ω resistors, and R_L). Make sure this is working properly *before* adding the power transistors. The 20 ohm emitter resistors in the mirror partially compensate for mismatch of Q_4 and Q_5 . Remember that Q_1 supplies all of the current to the load when the output is high. Note that later on in this experiment, the load resistor will be reduced to 100 ohms. Record the actual value of all resistors for later use in SPICE simulation.

This circuit often has problems with high frequency oscillation so use good construction techniques. Keep leads short and keep the input and output signals separated to avoid unwanted coupling capacitance. If present, the unwanted oscillation can generally be corrected by capacitors judiciously placed to bypass the high frequencies. You will have a much better chance getting this circuit to work if you use bypass capacitors on the power supplies. Place 0.1 μF capacitors between VCC and ground, and between VEE and ground. You may also wish to add electrolytic capacitors with values between 10-500 μF across the power supplies as well to suppress noise.

Mount the 5 K potentiometer so that adjustment is easy and convenient.

Troubleshooting

There is a high probability that your circuit will not work properly on first try. In order to minimize the number of burned-out transistors, it is recommended that you test the circuit in sections. Note that Q_1 and Q_2 have no collector resistors to limit I_C in case of wiring mistakes.

First, verify the operation of the current mirror. Remove transistors Q_1 and Q_2 , and ground point v_i . Apply the supply voltages and determine the mirror output current by measuring the voltage drop across the 20 ohm emitter resistor of Q_5 . Adjust this current to the design value by varying the size of R_{ref} . The likely mismatch of Q_4 and Q_5 may make the design equations inaccurate.

Once you have your current mirror working properly, verify the operation of the V_{BE} multiplier. Measure the voltage between the collector and emitter of Q_3 as the 5 K pot is varied from one extreme to the other. This voltage range should be within about 0.1 volt of the predicted value.

When you have this part of the circuit operating properly, set the multiplier output to its MINIMUM value, then insert Q_1 and Q_2 and continue with the circuit characterization.

Low Power Characterization

Use the x-y display mode on the scope to display the transfer characteristic of the amplifier. For V_{in} use either a triangle wave or sine wave of about 1 KHz (but accurately measure the frequency you use). Display the curve for V_{in} varying from -10 to +10 volts. Most of the curve should be nearly a straight line with a slope a little less than 1. (Why is the slope < 1 ?)

Experimentally adjust the pot to make the crossover distortion of class B operation plainly visible. To see the class B operation most clearly, use a wire to short the bases of Q_1 and Q_2 together. This sets $V_{BB} = 0$. Accurately draw this curve in your logbook. Label the axes and put accurate scales on them.

Set V_{in} to 0.0 and measure and record V_{BE} of Q_1 , Q_2 , and Q_3 .

Set V_{in} to whatever value is required to make $V_o = 0.0$ V and then adjust the pot to make I_Q equal to 1.0 mA. The 10 ohm resistors provide a way to monitor I_Q . Record all measurements. What is the purpose of this non-zero value of I_Q ?

Now set V_{in} to 0.0 and record all node voltages in your circuit. You will later compare these values with SPICE calculations.

Again use the x-y mode on the oscilloscope to display the transfer characteristics. Carefully sketch and label the curve in your logbook. Note this time there should be no

visible crossover distortion. (Why?) From this curve, determine the small signal voltage gain.

High Power Operation

Now replace the 1 K load resistor with a 100 ohm load resistor. Be sure to choose a resistor with an adequate power rating. The small resistors you are accustomed to using are typically rated at 1/8 W or 1/4 W. Calculate the maximum power to be dissipated in the load resistor.

Use the x-y mode on the scope to display the transfer characteristic with the 100 ohm load. Accurately draw this curve in your logbook and determine the small signal voltage gain at several points on the curve. Is it the same with the 100 ohm load as it was with the 1 k ohm load? If not, why?

With V_{in} set to 0 volts, measure and record the dc voltage on all nodes of your circuit.

Now with V_{in} set to a dc value that makes V_o equal 0 volts, again measure and record the dc voltage of all nodes. Measure and record the value of I_Q . It should still be close to 1 ma.

Can you explain the origin of the dc offset between V_{in} and V_o ?

Drive the power stage with a sine wave. Adjust the amplitude of V_{in} to give the largest undistorted output waveform. Record both the positive and negative peak voltages of the output wave.

Observe and measure the current wave forms through Q_1 , Q_2 , and the load resistor using the oscilloscope and the 10 ohm resistors. In your logbook, draw all three of these waves on the same plot using the same time base. Accurately label the axes. Do the current wave forms through Q_1 and Q_2 overlap? If they do, your sketches should accurately show it. Your data and sketches are to be complete and accurate enough that you can accurately calculate the power conversion efficiency of your circuit.

Determine the power conversion efficiency of the circuit at this operating condition.

Spice Simulations

You are to simulate this circuit using SPICE. As a minimum, your simulations are to provide the following information. Include the signal generator output resistance in your SPICE input file.

- a. The transfer curves under conditions which show, and do not show the crossover distortion.

- b. The dc voltages of all nodes under both conditions (i.e. $v_o = 0$ and $v_i = 0$) measured in the High Power Operation portion of this experiment.
- c. SPICE simulations of all plots measured in the High Power portion of the experiment.
- d. **OPTIONAL EXTRA CREDIT (5 points):** As an additional SPICE exercise, do a Fourier analysis of the output for the high power conditions of the last part of this experiment. What is the simulated total harmonic distortion?

Report

Include the following in your report:

1. A direct comparison of your experimental plots and the relevant SPICE simulations. For each comparison, put both plots on the same set of axes.
2. A discussion of differences.
3. Suggestions for improving the circuit.