EE313 Laboratory #3

Push-pull Class-B Power Amplifier

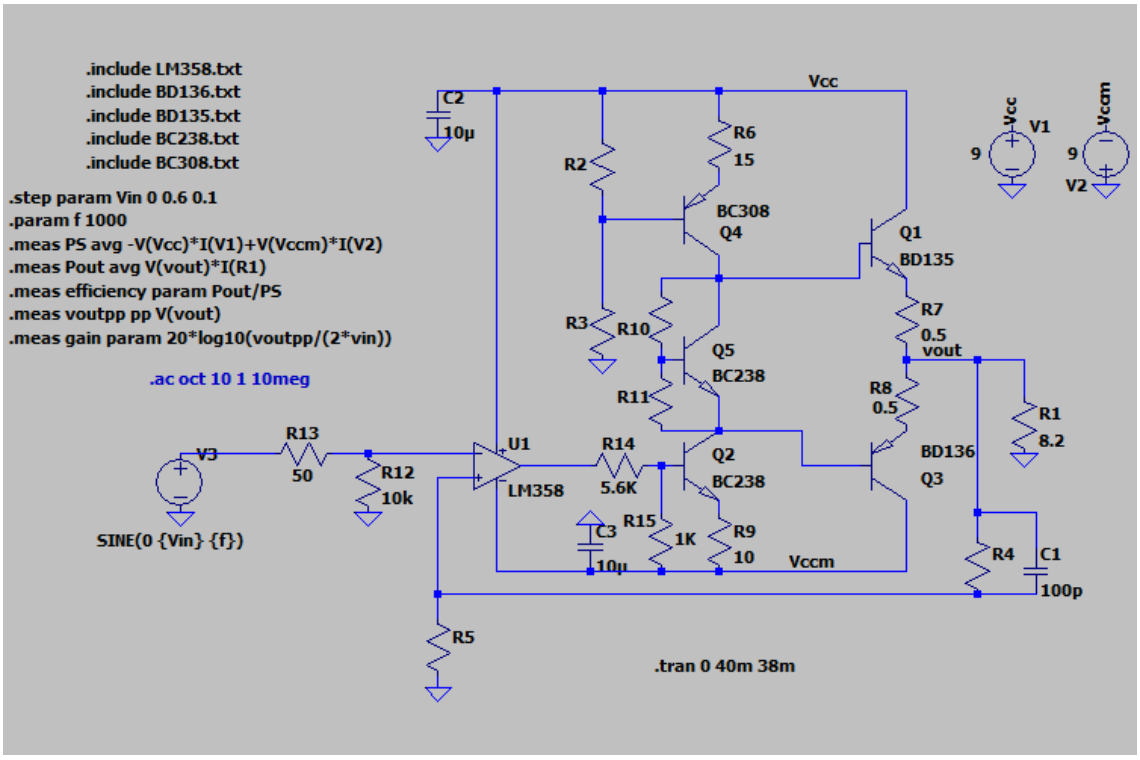
Design a complementary push-pull Class-B power amplifier capable of delivering at least 2.25W to an 8.2Ω resistive load. The supply voltage is at most ±9V. It should operate with sinusoidal voltages between 10Hz and 20KHz with a gain you choose between 15 dB and 25 dB. The SRS DS345 signal generator provides the input without an offset.

Specifications:

1. The amplifier should deliver at least a 2.19W power to an 8.2Ω resistance ($12V_{pp}$ to an 8.2Ω power resistor) starting from 10Hz to 40KHz at the chosen gain value.
2. The harmonics (the highest is possibly the third harmonic) at the 2.25W output power level should be at least 40 dB lower than the fundamental signal at 1 KHz.
3. The power consumption at quiescent conditions should be less than 500mW.
4. The amplifier's overall efficiency (output power/total supply power) should be at least 45% at max power output at 1KHz.

Preliminary work (Due March 25, 2024)

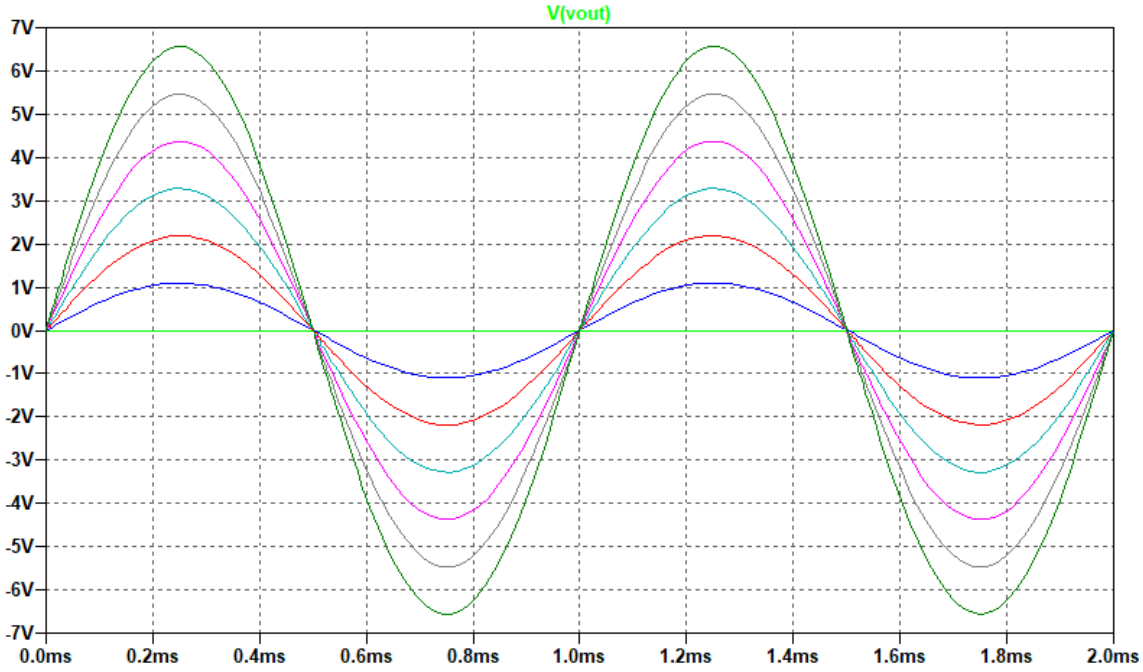
Provide a schematic of your design, showing a component list. Use Diptrace to generate the schematic. Available transistors: BC238 (npn, small-signal), BC308 (pnp, small-signal), BD135 (npn, power), BD136 (pnp, power). A suggested circuit is as follows:



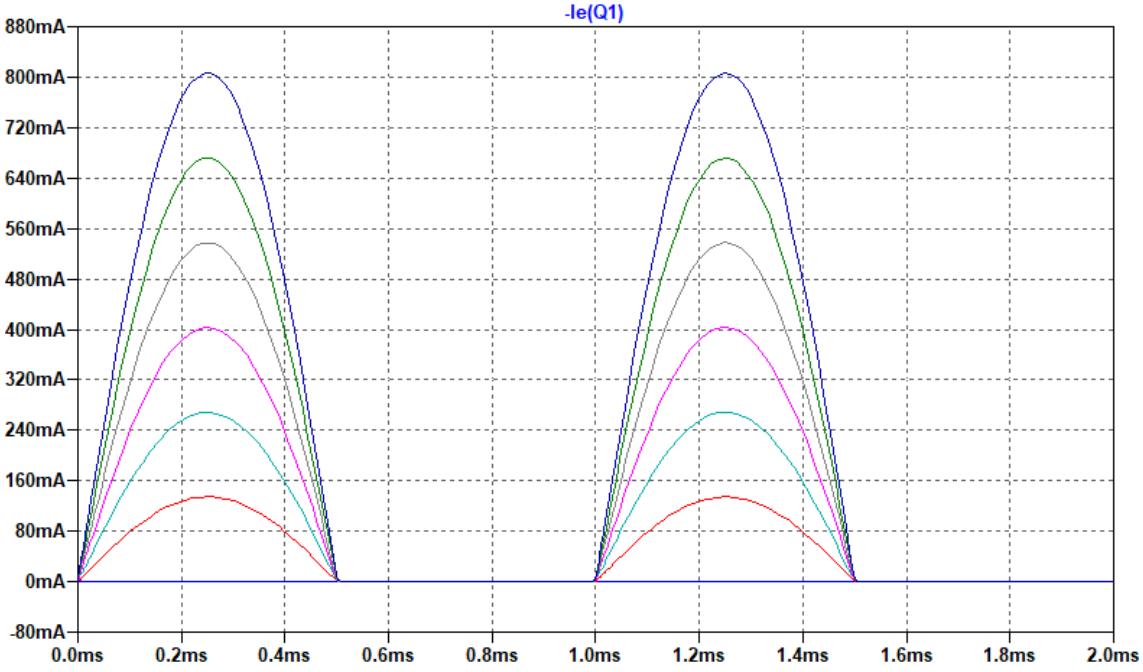
- The use of two supply voltages removes the need to use a large-value DC block capacitor between the output of the amplifier and the load resistor (Otherwise, you would need a large size 20,000 µF capacitor)
- R1 is the 8.2 Ω load resistor connected directly between the output and ground.

- R7 and R8 are emitter resistors, protecting emitter-follower configuration output transistors (to some degree) against short circuits. When the output load resistor is shorted, the voltage across R7 or R8 grows, reducing the base-emitter voltage, hence limiting the current.
- Q4, R6, R2, and R3 form a current source. The current value should be in the 15-20mA range to supply the base current of Q1.
- Q5, R10, and R11 form a VBE multiplier. The V_{CE} of Q5 should be about twice that of V_{BE} ($V_{CE} \approx (1+R10/R11)V_{BE}$). This value should be adjusted to make the quiescent current of Q1 and Q2 very small (a few mAs). You should choose the resistors R10 and R11 properly to achieve Class-B condition. If the V_{CE} of Q5 is too high, the circuit gets more into Class-AB operation, and the efficiency decreases. If the V_{CE} of Q5 is too low, the output voltage's distortion (harmonic content) will increase due to cross-over distortion.
- OPAMP provides the base current of Q2 through the resistors R14 and R15. R9 limits the current of Q2.
- The OPAMP, along with R4 and R5, form a feedback circuit to set the gain of the amplifier. The voltage gain is given by (1+R4/R5). Note that the feedback circuit is connected between the output and + input of the OPAMP since Q2 provides one more phase inversion.
- R12 connects the negative input of the OPAMP to the ground (in case the input is left open-circuited).
- R13 simulates the source impedance of the signal generator.
- C1 is the stabilization capacitor. Without C1, the feedback circuit oscillates at a high frequency. If a transient simulation takes too long, it is probably due to this oscillation. Stop the simulation and restart with a larger C1. If C1 is too large, the amplifier's high-frequency response will suffer.
- C2 and C3 are power supply decoupling capacitors needed in every circuit to make power supplies act like AC short circuits.
- Find the gain and efficiency of the amplifier as a function of the input level. Use the spice directives as follows:

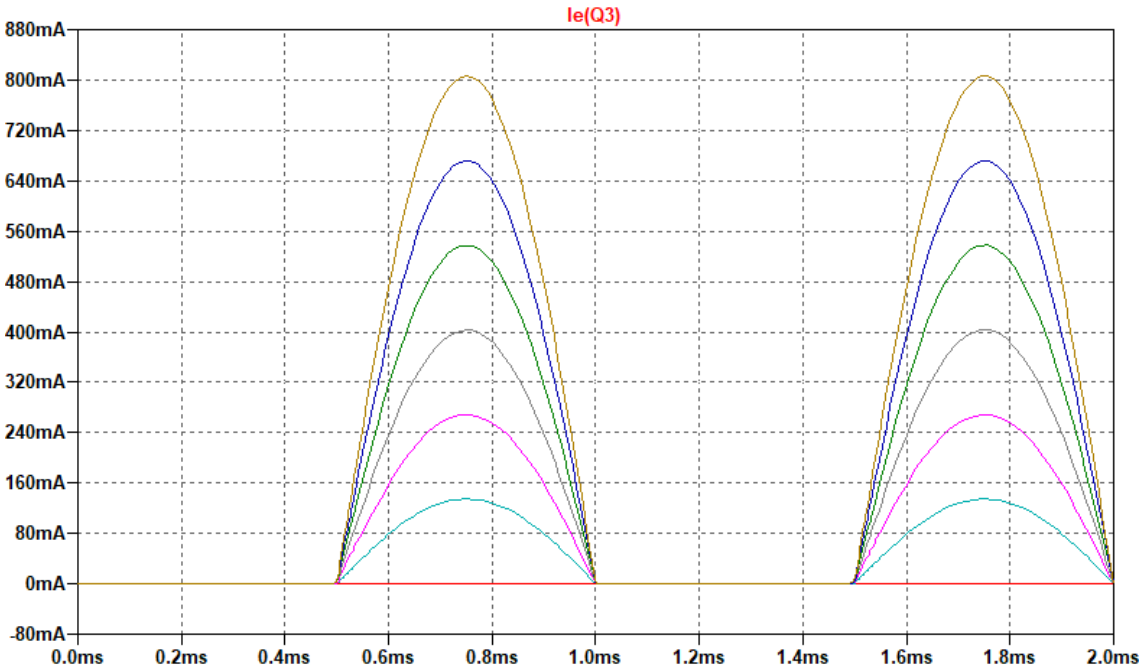
  .step param Vin begin end step (to step the peak input voltage)
  .param f 1000 (to set the input frequency to 1KHz)
  .meas Pout AVG V(vout)*I(R1) (to find the output power, R1 is the 8 Ω load resistor with a voltage label of vout)
  .meas PIN AVG -V(vcc)*I(V1)+V(vccm)*I(V2)
  (vcc is the positive supply voltage label, V1 is the positive voltage source, vccm is the negative supply voltage label, V2 is the negative voltage source)
  .meas Efficiency param Pout/PIN (Efficiency is the ratio of the output power to supply power)
  .meas voutpp pp V(vout) (measure the output load resistor p-p voltage)
  .meas gain param 20*log10(voutpp/(2*Vin) (list the gain in dB at each input level)



The emitter current of Q1 should be like below. Note that the current is zero for half the period.



The emitter current of Q3 should be like below:



After running the simulation, click CTRL-L to see the results at each step. In your report, the graphs should show two cycles of the output waveform, like the one shown above (For an accurate FFT, you need 20 cycles in the plot window).

Power delivered by the supply as a function of input level (between 0 and 0.6V in 0.1V steps)

Measurement: ps

step	AVG(-v(vcc)*i(v1)+v(vccm)*i(v2))	FROM	TO
1	0.309636	0	0.002
2	1.0648	0	0.002
3	1.82681	0	0.002
4	2.58794	0	0.002
5	3.34846	0	0.002
6	4.10766	0	0.002
7	4.86694	0	0.002

The output power as a function of input level:

Measurement: pout

step	AVG(v(vout)*i(r1))	FROM	TO
1	1.2494e-08	0	0.002
2	0.0728726	0	0.002
3	0.29136	0	0.002
4	0.655523	0	0.002
5	1.16546	0	0.002
6	1.82123	0	0.002
7	2.62362	0	0.002

The efficiency as a function of input level:

Measurement: efficiency

step	pout/ps
1	4.03506e-08
2	0.0684378
3	0.159491
4	0.253299
5	0.348057
6	0.443375
7	0.53907

Note that LTSpice can show the instantaneous power dissipated on a component by ALT-Click. You can find the average power by CTRL-Click on the instantaneous power symbol in the graph window if an integer number of signal cycles is shown.

To find the magnitude of the harmonics, simulate with 20 cycles visible on the plot window. Then, right-click on the plot window, View → FFT, to see the amplitude of different harmonics. You can find the dB difference between the fundamental (at 1 KHz) and the third harmonic (at 3 kHz)

- Find the amplifier's gain as a frequency function for the maximum input level using transient analysis and spice directives. Note that small-signal analysis is not suitable for a Class-B amplifier. Use the spice directive:

  .step param f begin end step

  .param Vin 0.6

  with {f} defined as the frequency parameter of the input source.

Experimental work (Due April 1, 2024)

Build your design on the breadboard neatly. Use heat sinks for power transistors. Place the active components so that the required wiring is either short or there is no need for wires. Cut the leads of resistors to prevent dangling, which may cause short circuits between nodes. Check the interconnection wires before plugging them in. Wires may be faulty. Use supply decoupling capacitors of value 10 µF for each supply. These capacitors ensure that the supply voltage sources are shorted to the ground at the operating frequency. Make sure that ground points are connected.

Follow the steps below:

1. Connect 100 Ω resistors in series with each supply line while debugging your circuit. These resistors will be removed later.
2. Do not connect the 8.2 Ω load resistor yet. Apply the supply voltages while gradually increasing them. The supply currents should be small and equal (about 20 to 30mA each). If not, check your connections. There may be missing connections or faulty cables, and the transistor pins may be incorrect. Measure the voltage between the base and emitter for each transistor. Make sure that you measure a voltage of around 0.6V. Measure the collector-emitter voltages of each transistor. All transistors must be in the active region.
3. Measure the voltages at all nodes and compare them with the expected values. If the voltages at different points of the same node are different, you have a faulty wire or a bad connection in the breadboard.
4. Measure the collector-emitter voltage of the V_{BE} multiplier circuit. It should be about 1.1V.
5. Measure the voltages at the input pins of the OPAMP. They should both be near the ground voltage (less than 2mV).

6. Measure the voltage between the output and ground. It should be also very small (less than 20mV). If not, check your feedback circuit.
7. Apply the 1KHz 0.1V peak input signal from the generator. Connect the oscilloscope to the output pin. You are on the right track if you see a 2Vpp signal with zero offset at the output pin.
8. Connect the 8.2Ω load resistor. Apply a 0.1V peak. You should still see the 2Vpp signal with zero offset. The power supply current should increase.
9. Remove the 100Ω resistors from the supply lines. Make sure that 2Vpp sinusoidal output signal is still visible. You may now increase the input signal while observing the power supply current.
10. Measure the gain as a function of frequency at the maximum input level. Determine the upper frequency where the gain drops by 1 dB.
11. Find the amplitude of the harmonics under the full-power condition using the FFT feature of the oscilloscope.
12. Measure the power supply current under quiescent conditions.
13. Measure the efficiency at the maximum input level at 1 KHz.
14. Measure the efficiency as a function of input level at 1KHz.

Grading criteria:

Preliminary work (10 pts)

Nice looking schematic with component list: 2pts

Satisfaction of all four criteria in LTSpice: 8pts, 2 pts each

Experimental work (10 pts)

A neat, easy-to-follow, and easy-to-debug circuit on breadboard. 2pts

Experimental satisfaction of all four criteria: 6 pts, 1.5pts each

Efficiency plot as a function of input level: 1pt

Gain plot as a function of frequency at max input level: 1pt