



Dual Amplifier Custom PCB

By Hal Silverman, WB6WXO
SOARA Education Director

Thanks to Brian Roode, NJ6N, for writing this report of a SOARA repeater project using an inexpensive Printed Circuit Board service.

We recently suffered a failure of the radio transceiver that provided the RF link from the Temple Hill 147.645 to the Santiago 447.180 and 224.640 repeaters. A temporary mobile transceiver was put into place at the Temple Hill site. Unfortunately, after this change the audio quality through the link turned out to be poor due to the incorrect number of pre-emphasis and de-emphasis stages.

A Motorola Maxtrac transceiver was procured by Tom, AE6SH. Because the pre-emphasis and de-emphasis stages on this radio are configurable, to be in circuit, or out of circuit, we thought this would make a good link radio. The only problem was matching its audio levels with the audio levels of our repeater controller.

An operational amplifier was needed. Howard, KG6GI, designed a circuit and provided me with a schematic. He also calculated the values of the key components. See Figure 1 for the schematic we followed to build the interface.

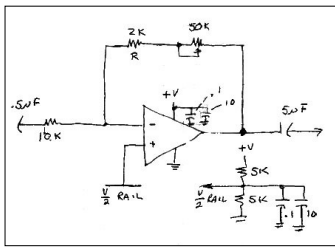
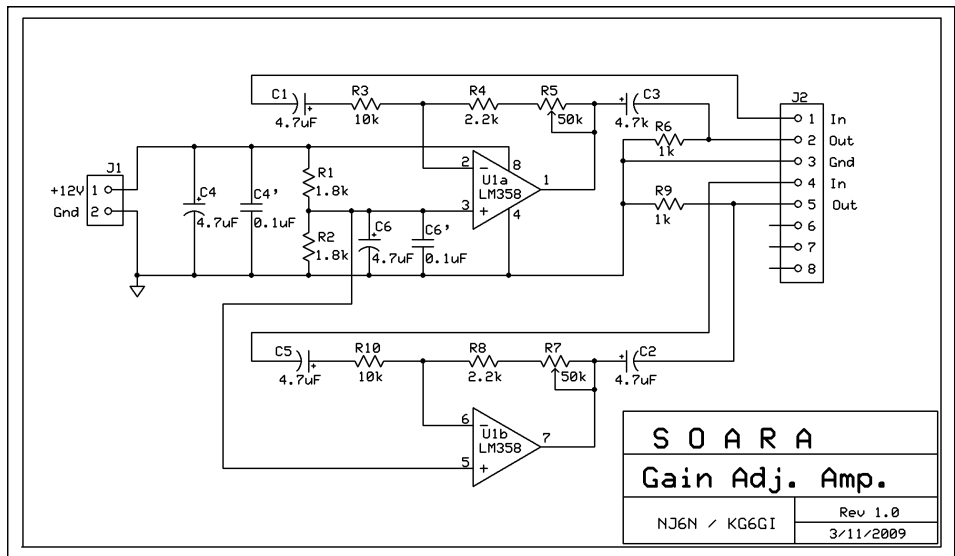


Figure 1

Above: the original circuit sketch.
Right: The full schematic of the dual amplifier as built.



I created a prototype of the circuit for testing; see Figure 2. Using a signal generator, I injected a 1Khz signal at 1Volt peak to peak. Using a 50K ohm, 20-turn trimmer in the feedback path, I was able to adjust the gain of the op-amp to produce a 0.2V through 5V output, just as Howard designed it. A 2K ohm resistor sets the lower limit on the gain to a minimum of 0.2. I re-learned a lot

about operation amplifiers through this process. If you haven't built something with an op-amp recently, you're really missing out!

At some point I mentioned to Dale, W8RRV, that I needed to construct this interface circuit. He told me about a really great printed circuit board manufacturing company, www.expresspcb.com, that has a low-cost PCB manufacturing service.

I downloaded their free schematic and PCB layout software and was able to design a circuit board in about an hour and a half. See Figure 3 for the layout of my first PCB. I refined this one a bit before I submitted it for manufacturing.

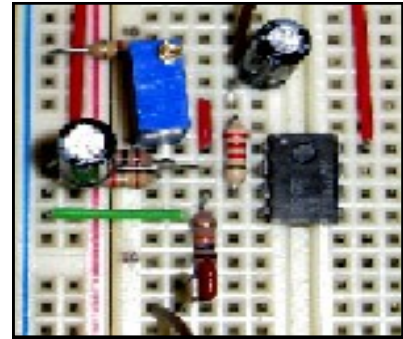


Figure 2
Solderless Breadboard prototype of the circuit.

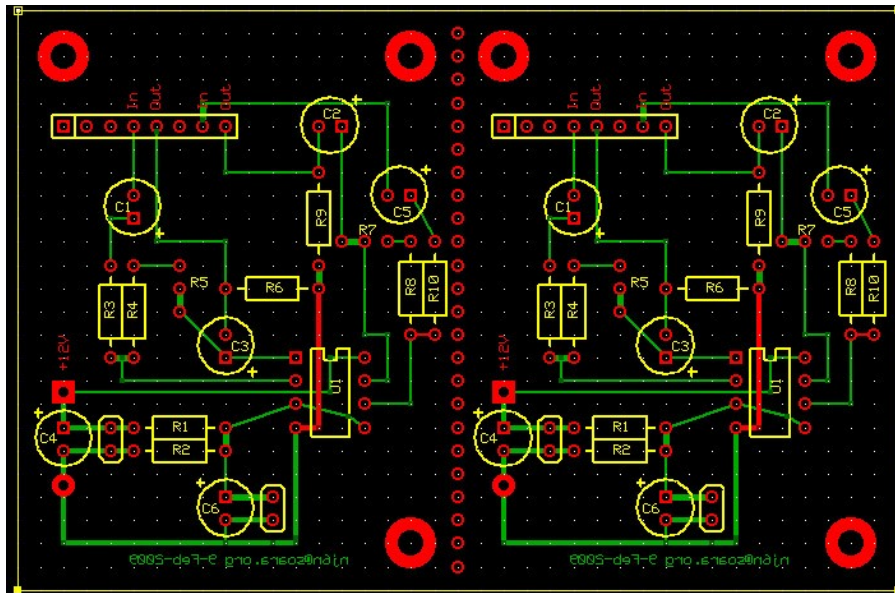


Figure 3
Artwork for the PCB. The top metal layer is shown in red, and the back side layer in green. The "silkscreen" layer is shown in yellow. The finished boards do not get silkscreened. Note the row of through-holes which we used to divide the board and make it easy to cut into two individual boards.

Designing a circuit board using software is much faster than using etch resist "press-ons" and there is no x-acto knife or Sharpie required. Better still, there's no messing with ferric chloride etchant, and you don't have to drill all of the tiny holes yourself. You can specify what type of component is being placed using their built-in library of components or specify custom components so that the spacing, size, and number of holes required is set accordingly. The size of the circuit board traces is also configurable on a per-trace basis.

The PC boards are double-sided, high quality and feature plated through "vias". The design software supports creating a ground-plane that automatically spaces your new and existing traces away from the ground plane and "thermal pads" which make it easier to solder components to the ground plane without dissipating the heat from your soldering iron.

When you're ready to order the circuit boards you've designed, the ExpressPCB program does some "sanity checks" and calculates the total cost. ExpressPCB has a "MiniBoard" service that provides three 2.5" x 3.8" boards for \$51 plus 7.75% tax and \$9.85 shipping. Boards are shipped the next business day by 2nd day air after your order is placed via the internet. The total cost comes out to about 65 dollars.. The neat thing is that you can create as many circuits as will fit on their MiniBoard for the same price, as long as you don't go over the total number of holes. For my first board, I divided the board in half and made two identical circuits on each side. If I had spent more time, I could probably have fit four of them on one board.

The finished product can be seen in Figure 4. I enclosed the circuit board in a die-cast aluminum enclosure and installed it between the Maxtrac radio and our repeater controller and fine-tuned the amplifier gain adjustments. You can hear the results during the next Tuesday night SOARA net on the 147.645, 224.640, and 447.180 repeaters.

73, NJ6N

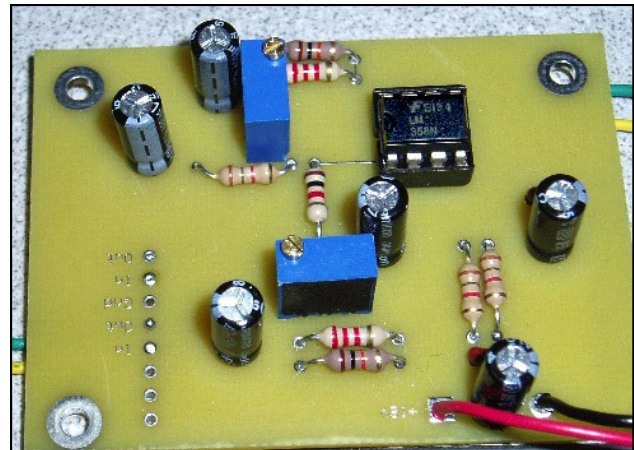


Figure 4

Top view of the completed dual amplifier circuit. The 8-pin dual op amp (LM358) can be seen in the socket at the upper right. Input and output leads are connected at the vertical row of through holes at the left. Note the labeling made in the top metal layer.