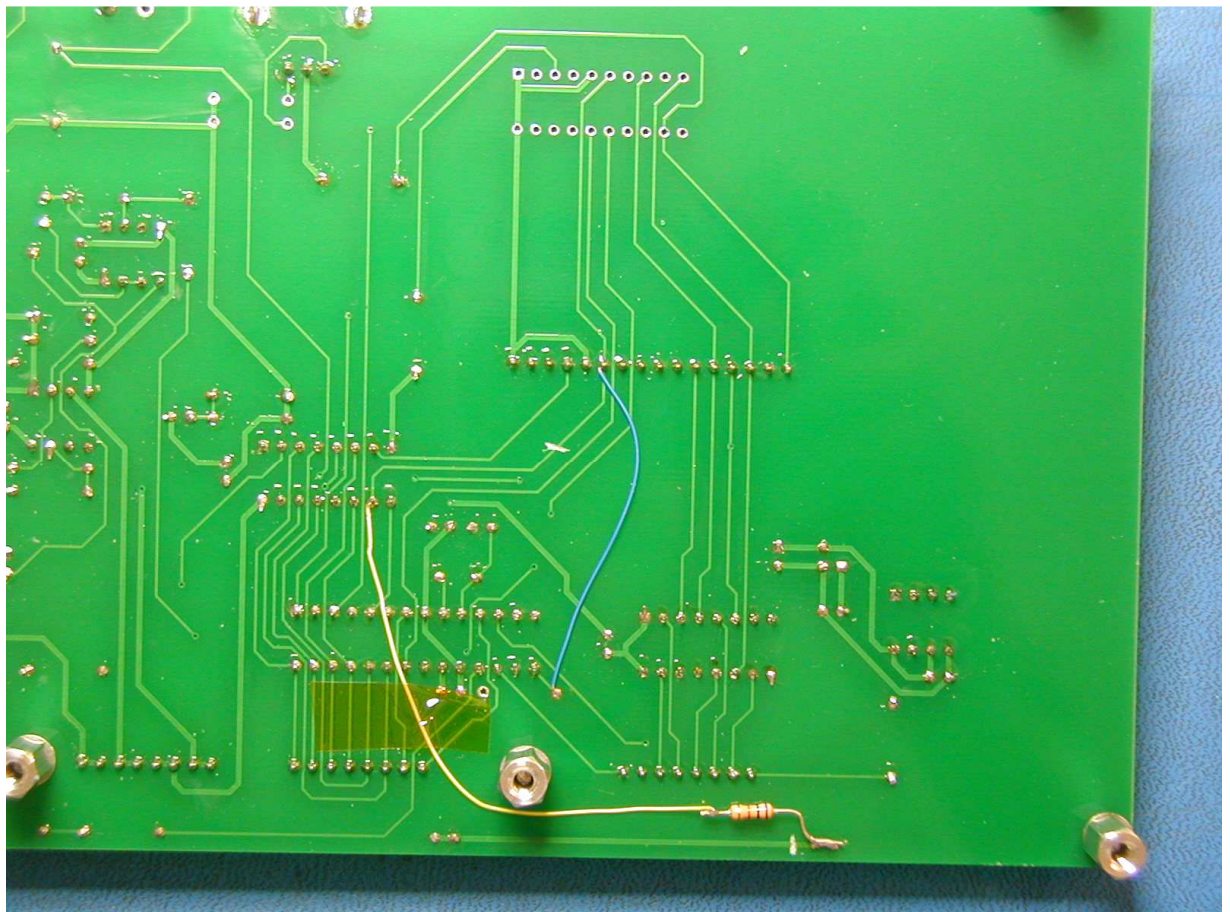


There are a few things that need mentioning concerning the pc board:

- 1) Somehow the "+" sign for the electrolytic caps got placed on a layer that is not in the silkscreen. You can see the polarity of these caps on the component placement diagram.
- 2) The holes in the pc board for the S7049 single row female connectors are smaller than they should be. You can still install the connectors, you just need to push to get them in place (carefully).
- 3) The holes in the pc board for the headers that connect to the voice board are also smaller than they should be. Again, you can still install the headers, by pushing on them.
- 4) The holes in the pcb for the 7805A voltage regulator are too small for some versions of this part. The lead size varies with the manufacturer. I soldered mine on top of the board, while another person trimmed his leads with cutters so that they would fit.
- 5) The U5 voltage regulator pin numbering on the schematic does not match the part, and the pin numbering on the pcb footprint has pin 1 on the right. These two offset each other, and if you install this regulator as shown in the component placement diagram or on the silkscreen, it will be correct.

There are two cuts and two jumpers which you will need to add to your board before using it:

- 1) U2 pin 10 is no longer used for the LCDE signal. Instead, LCDE is driven by U1 pin 15.
- 2) The TP3 test point, which was connected to ground, needs to be isolated from ground, so that it can be used as an oscilloscope trigger output. Cut the trace leading to it, and connect it with a 1K resistor in series to U2 pin 10. Add a label on the board to indicate that this is the trigger output and not ground. Here is a photo showing the cuts and jumpers on the bottom of the board:



Please install IC sockets for all chips, especially the PIC chip, in case there are firmware updates in the future.

JP1 (next to C5) must be shorted with a blob of solder for the circuitry to work. Install a jumper wire for R24. I used trimmed resistor leads for the Gnd and Vout test points, bent in the shape of a U, sticking up about 1/8" above the board. This gives something to clip an alligator clip or scope probe onto.

The pushbutton switches that the board was designed for were not usable, IMO, so I changed to better ones. Only two of the four holes for each switch are electrically connected, so make sure you install your switches between the correct two holes.

The circuit consisting of U8, D1-3 and associated components was intended for measuring the amplitude of the audio signal, for VCA testing, but is not currently used by the firmware. I suggest that you install these components anyway, in case a test is added in the future that needs them. (I currently, have no plans to add more tests, BTW, but maybe someone else will.)

Resistor R19 controls the LCD contrast. A value of 2.2K seems to work well with the LCD in the BOM. Resistor R20 controls the LCD backlight current/intensity. The 47 ohm value seems fine to me.

The display called out in the parts list has two rows of holes, one along the top edge, and one along the bottom edge. These are connected in parallel on the module, so that either one can be used. I implemented both rows of holes on the pcb, so that the connectors could be used to support the LCD. Only the top row of holes on the pcb are connected. There are other LCD's from the same vendor with different color schemes, but the one I specified seems to be the only one that has both rows of holes. So probably one of the other ones would also work, but you would need to support the LCD in some other way. Connector H1 was added to the pcb to support mounting the switches and LCD remotely, in case someone wanted to do that.