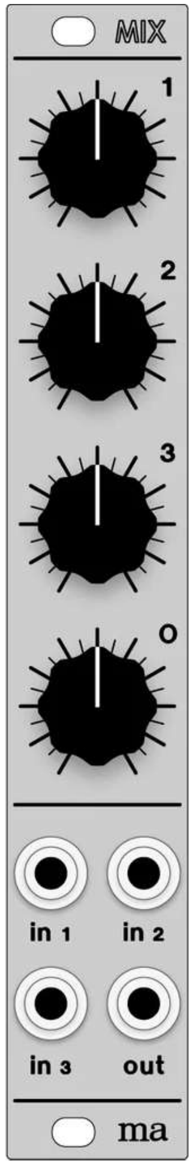
THREE CHANNEL MIXER (MIX)



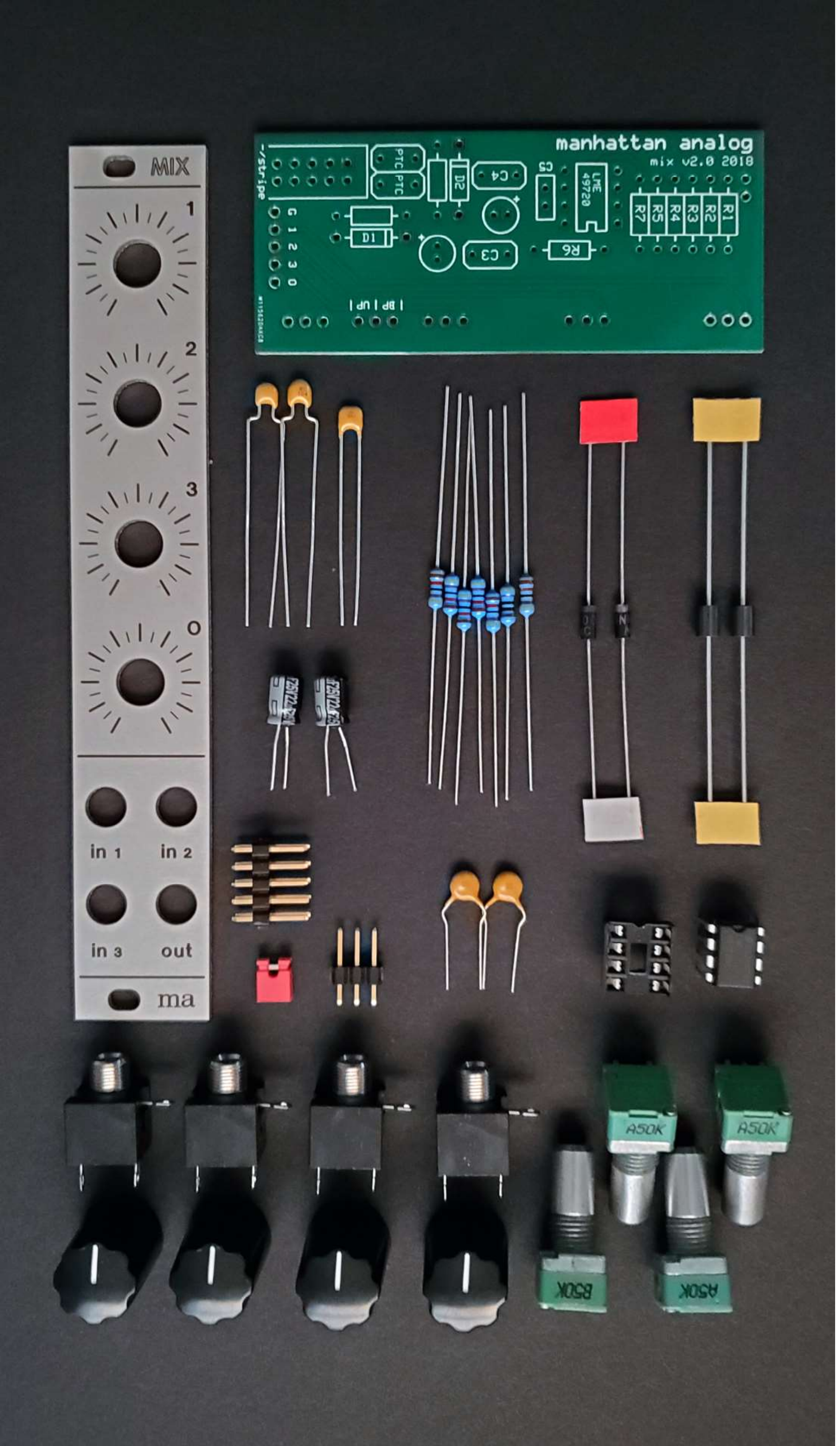
Module Documentation & DIY Build Guide, *PCB v2.0*

The Manhattan Analog *mix* is a Eurorack format mixer module using a very basic dual inverting stage opamp design. It features three inputs and a unity gain output which can be either bipolar or unipolar in response, selected by an onboard jumper.

Being such a basic design, this one can be built out in a number of ways depending on how you want to use it. It’s intended to be a high quality audio mixer with the LME49720 opamp, but it can just as easily be built as a low power consumption CV mixer using a TL062 or as a basic utility mixer using the ubiquitous TL072. The 47k resistors can be upgraded to 0.1% tolerance parts if you need to use it with pitch CVs and you can tweak the gain of each inverting stage by changing a couple of resistor values if you require amplification.

You can change the input pots over to linear taper if you prefer the way the controls feel that way, but I do recommend leaving the output pot as a linear taper so it will work with either jumper setting. (I mean, it’ll *work* with an audio taper pot in bipolar mode, but your zero point won’t be at 12:00 anymore and that can be annoying.)

For the full bill of materials, build instructions, and testing & trimming procedures, please refer to the rest of this document. Take your time, be safe, and enjoy the build!



Bill of Materials

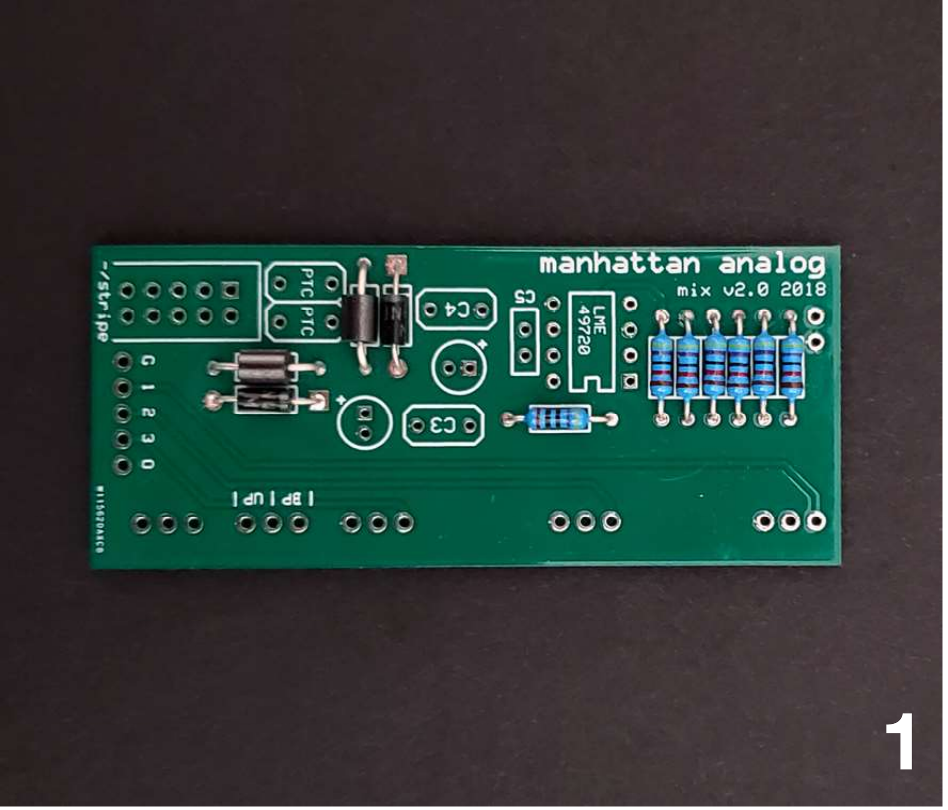
Resistors:
[6] 47k 1% - 0.1% optional
[1] 470R

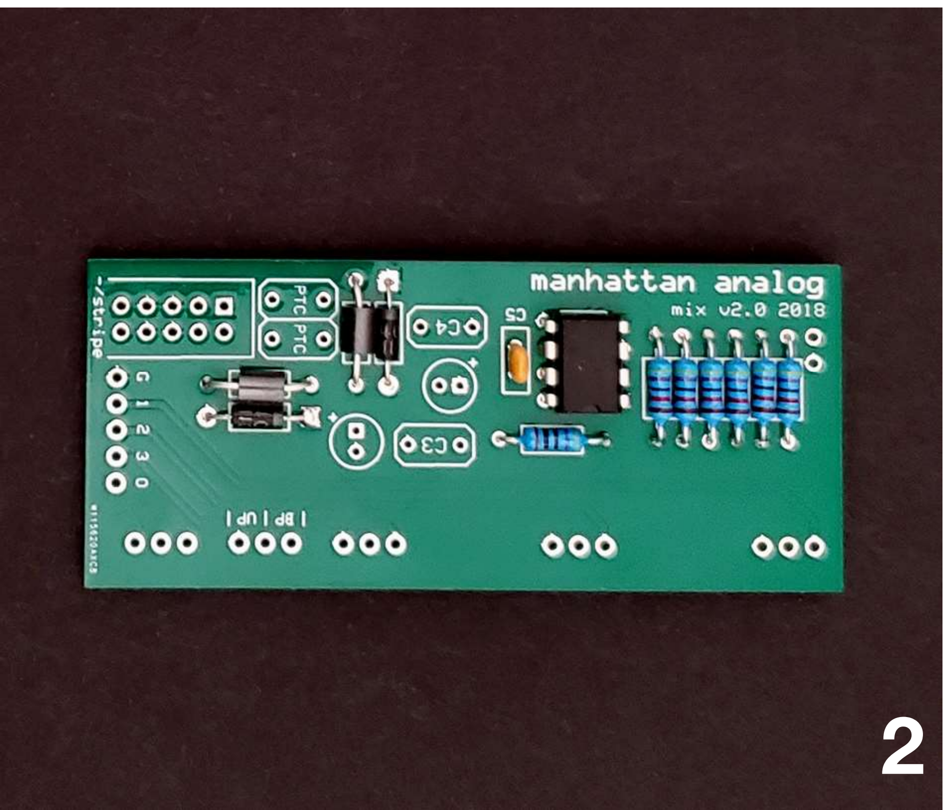
Capacitors:
[1] 22pF ceramic
[2] 22uF electrolytic
[2] 0.1uF ceramic

Pots:
[3] 50k audio, 9mm Alpha or similar
[1] 50k linear, 9mm Alpha or similar

Transistors/ICs:
[1] LME49720 - any pin-compatible opamp should work: TL062, TL072, OPA2134, etc.

Miscellaneous:
[1] 10-pin, 2x5, 0.100” header, male
[1] 3-pin, 1x3, 0.100” header, male
[1] 0.100” jumper
[1] 8 pin IC socket (optional)
[2] Ferrite bead (can substitute 10R resistor)
[2] PTC resettable fuse (.050 hold, .100 trip) - can replace with wire link but will lose some reverse power protection features
[2] 1N4001 diode
[4] 3.5mm jack, 16PJ138 or Erthenvar/Thonk style
[4] Davies 1900H-style knob, or any other ~12.5mm-diameter knobs you prefer.





Step 1

Parts Used:

Resistors:
[6] 47k
[1] 470R

Capacitors:
[1] 22pF ceramic

Transistors/ICs:
[1] LME49720

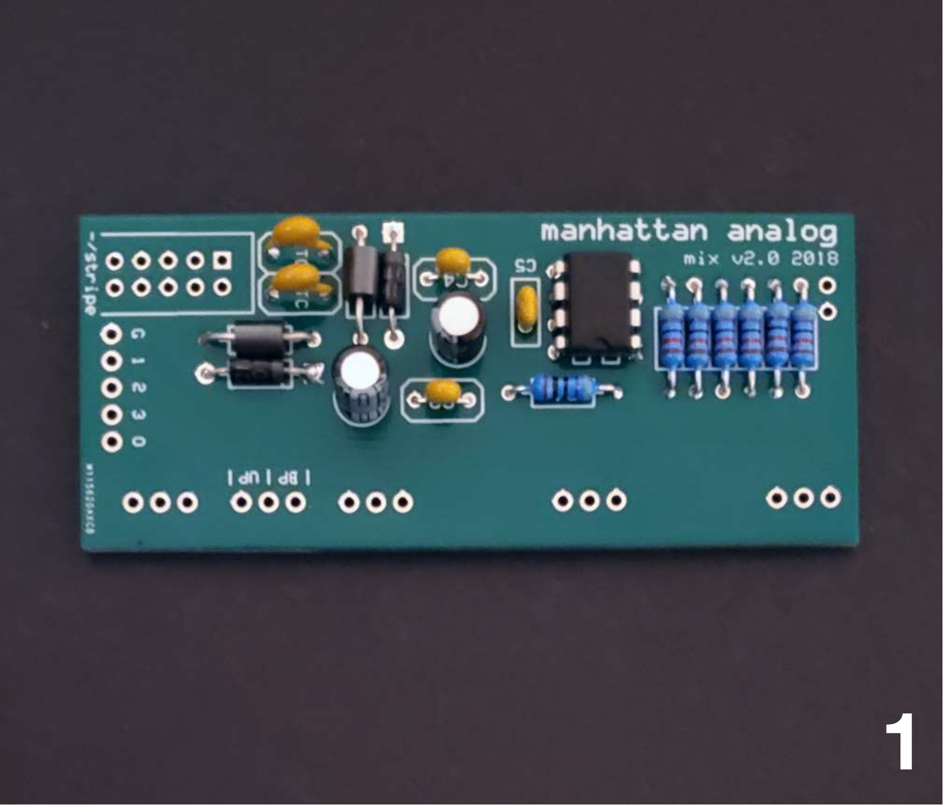
Miscellaneous:
[2] Ferrite bead (can substitute 10R resistor)
[2] 1N4001 diode
[1] 8 pin IC socket (optional)

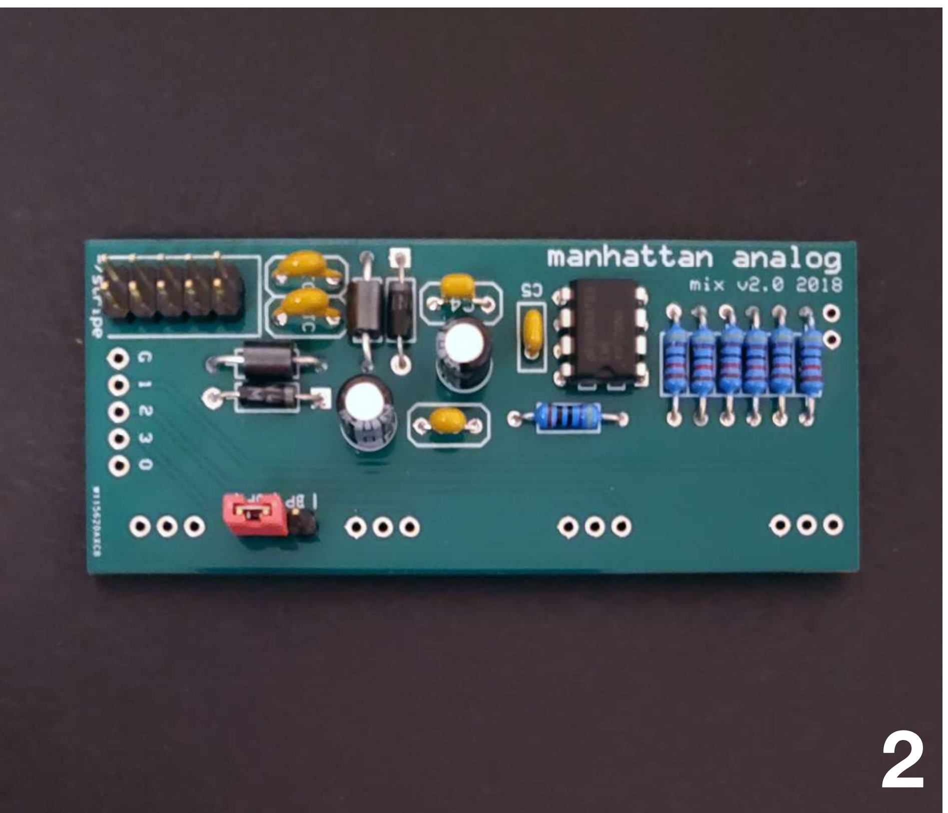
Procedure:

1) Start by populating the PCB with the lowest-profile components: the resistors, diodes, and ferrite beads. Solder all leads and clip them with a pair of flush-cutters.

2) Drop in the opamp (or the socket - not pictured) along with the 22pF cap. Take a moment to doublecheck the orientation of the IC at this point - that's pretty much the only critical mistake there is to make in this build, and LME49720s aren't *extremely* cheap... then solder the capacitor in place and clip the leads so you have better access to the IC pins.

3) Solder one pin on each side of the chip/socket, then flip the board over to check that it's seated flush with the PCB. Adjust it if you need to by reflowing the pin(s) you tacked it down with and applying some downward pressure on the body of the IC. Mind your fingers, the pin you're reflowing will be hot. When you're satisfied, solder the rest of the pins, inspect your work to check for cold joints or stray solder, and proceed to Step 2.





Step 2

Parts Used:

Capacitors:
[2] 22uF electrolytic
[2] 0.1uF ceramic

Miscellaneous:
[2] PTC resettable fuse (.050 hold, .100 trip)
[1] 10-pin, 2x5, 0.100” header, male
[1] 3-pin, 1x3, 0.100” header, male
[1] 0.100” jumper

Procedure:

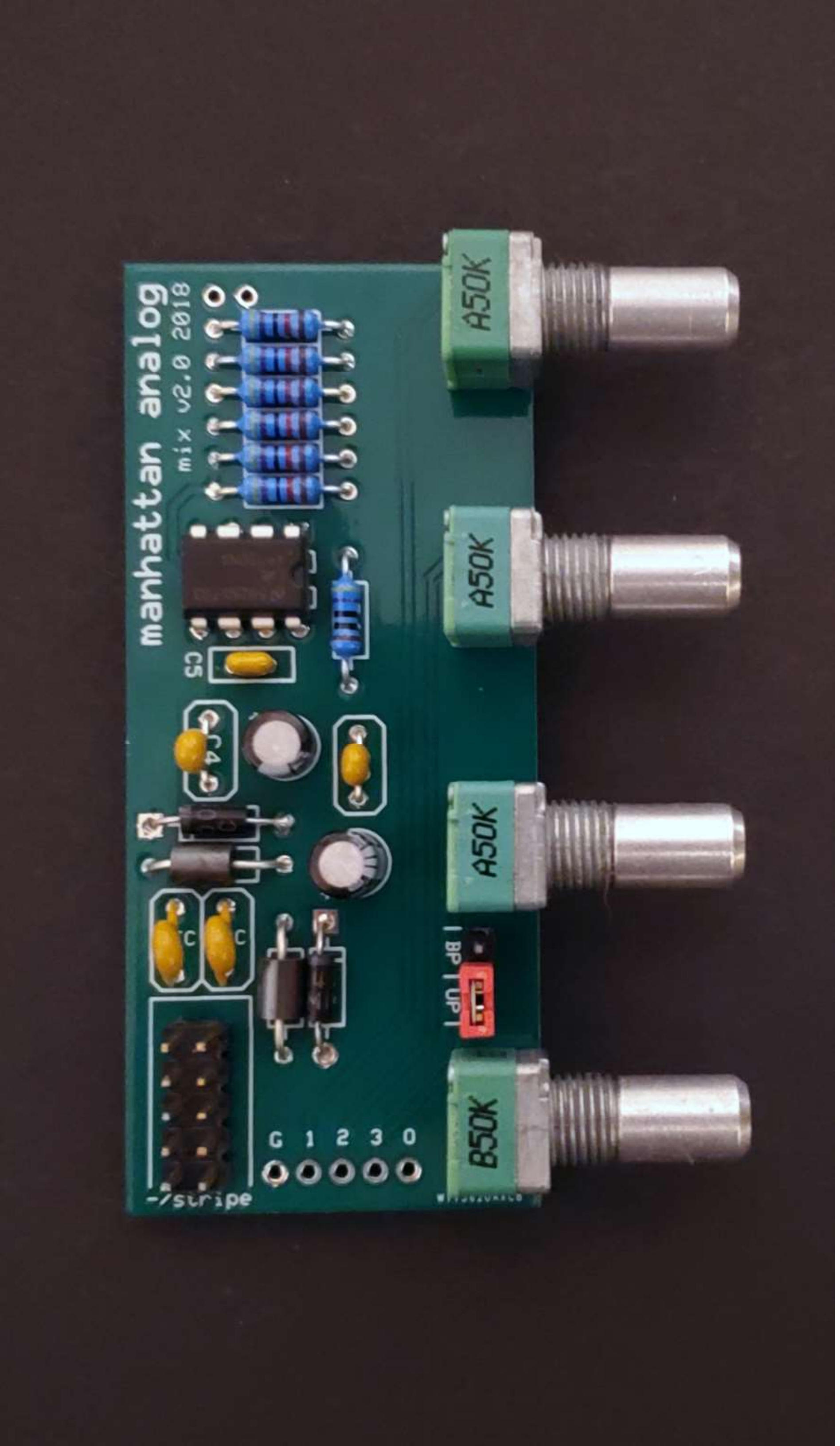
1) Populate the PCB with the mid-height components, then solder everything and clip the leads. I find it easier to place and solder the caps & fuses first and then do the headers, as shown in the two respective photos. If you happen to have especially tall electrolytic caps, no problem, but you may find it easier to reserve those for installation with the other tall components in Step 3.

2) Tack the 3-pin header down for now by just the center pin. Do the same with the power header - tack it down using just one of the end pins.

3) Now you need to make sure the pin headers are seated securely, flush, with the pins perpendicular to the PCB. If either one is sitting at an angle, reflow the pin you tacked it down with and apply downward pressure with your finger to seat the header flush on the board. **Mind your fingers near the iron, the pin you’re reflowing will be hot.** Once you’re satisfied, solder the remaining pins.

Don’t skip this step, especially with the power header. It makes a mechanical connection (to the power cable) and sitting flush on the PCB will make that connection stronger. Plus, you won’t have a power cable coming off the board at a wonky angle.

4) Install the jumper on the 3-pin header - select UP for unipolar (normal) mode or BP for a bipolar output. Your zero point on the panel control will be at 12:00 in BP mode. Proceed to Step 3.



Step 3

Parts Used:

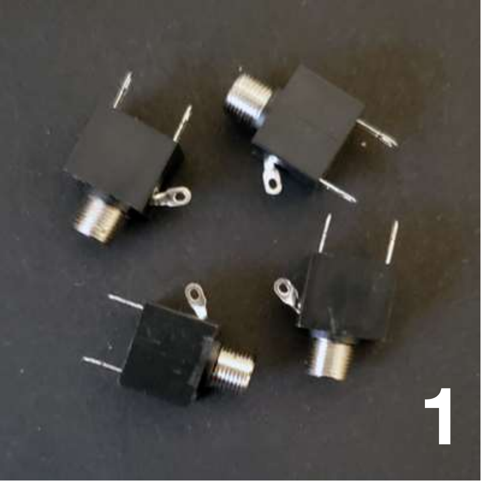
Pots:
[3] 50k audio, 9mm Alpha or similar
[1] 50k linear, 9mm Alpha or similar

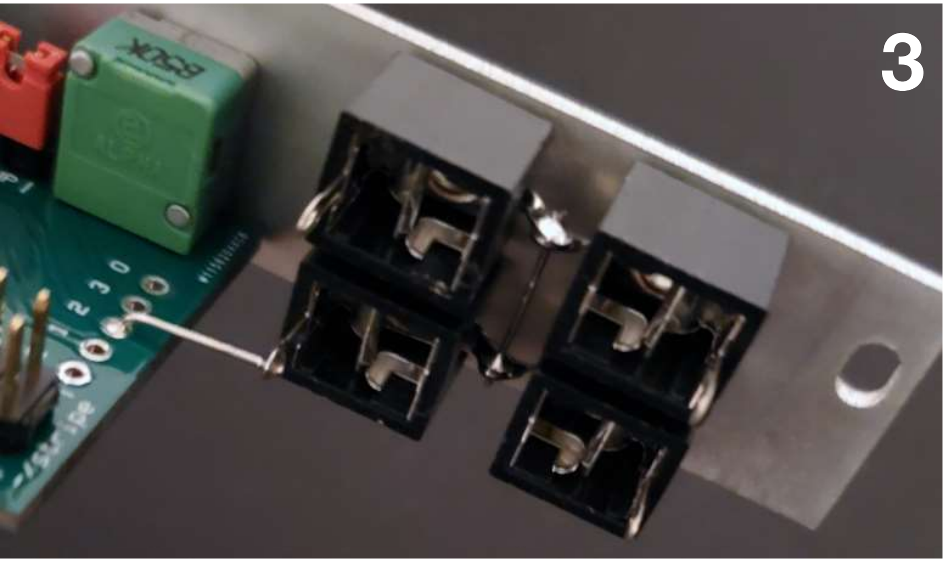
Procedure:

1) Populate the PCB with the panel pots and your tall caps from Step 2 if you had any left over.

2) Tack each pot down for now by just the center pin. Solder your caps if necessary, clip the leads with a pair of flush-cutters, and inspect your work to check for cold joints or stray solder.

3) Now it's time to seat the pots flush with the PCB. One at a time, reheat the solder on the middle pin while applying downward pressure with your finger to seat the body of the pot flush and tight with the PCB. This will form the mechanical connection to the panel in the finished module, so it's important for long-term reliability to get this right. **Mind your fingers near the iron.** When all four pots are sitting flush, don't solder the other pins of the pots just yet - we'll take care of that after we've mated the board to the panel in Step 4.





Step 4

Parts Used:

Miscellaneous:
[4] 3.5mm mono N/C jack, Kobiconn 16PJ138 or Thonkiconn style
[3] Davies 1900H-style knob, or any other ~12.5mm-diameter knobs you prefer.

Procedure:

1) If your panel has a protective plastic film on it, peel that off. (They haven't had this for several years, but in case you get older stock.)

2) Bend the ground tabs of your jacks 90 degrees and angle them slightly as shown in photo 1, then install all four onto the panel so that the tabs line up. You're going to pass a single piece of wire through all of them in the next operation, as shown in photo 2, so now's the time to arrange them to make that possible. See Thonk build notes below***

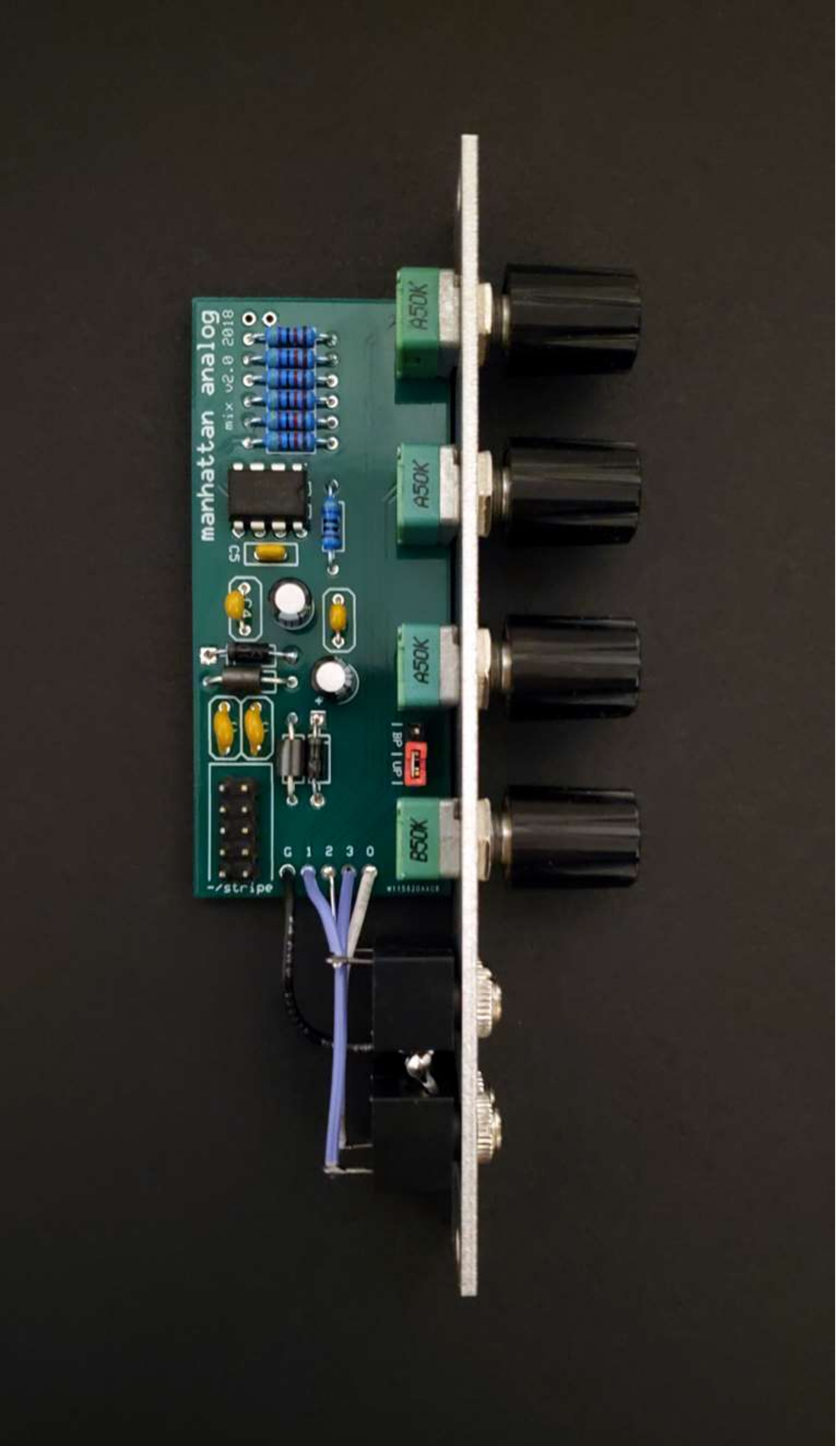
3) Pass that wire through all 4 ground tabs of your jacks to tie them together as shown in photo 2. (I use a clipped resistor lead, you probably should too. You'll have plenty by now and they're free.) Make sure you have good solder contact on all four ground tabs, they can be a little tricky if they don't meet up quite right. Mind your angles and try not to melt the plastic jack bodies with your iron. Clip the spare wire from the outsides of the ground tabs as best you can. (See photo 3.)

4) Install the completed PCB assembly from Step 3 onto the panel, taking care not to scratch the metal as you tighten down the nuts on the pots. Now is when you finally finish soldering the CW and CCW pins, but first, reflow each of the middle pins one more time - this will relieve any mechanical stress that has been induced by installation onto the panel. This is important for long-term reliability, so don't skip this step. Once all four pots have been reflowed and allowed to settle into place, finish the remaining 8 unsoldered pins. You may wish to clip these short with your flush cutters. This is optional, but I've taken to doing it.

5) Take another clipped resistor lead, bend it at 90 degrees with about 12mm length on one side, and use that to connect the **in 2** jack to the corresponding pad on the PCB as seen in photo 3. Solder this in place and trim the excess wire. While you're at it, clip the inner jack tabs with your flush cutters so you have more room to work as you wire the rest of the jacks. Please see photo 4 for clarification, this is one you don't want to get wrong. Run a short length of insulated wire (~35mm) from the common ground wire you just installed in the jacks to the pad marked **G** on the PCB. Then run wires from the **in 1, in 3,** and **out** jacks to the corresponding **1, 2,** and **O** pads.

***The jack sockets included in Thonk kits have longer ground tabs than the type described in this guide. Bend the long ground tabs on each jacks to 90 degrees, and then trim them down slightly so that they can be placed in the configuration of image 2.

The ground pins which face each other can overlap and be soldered together for stability. Then place a resistor leg over them as shown and solder the leg to each pair of ground pins to connect the ground of all four jacks.



Step 5

Testing & Trimming Procedure:

1) Doublecheck the orientation of your power cable and then plug the module into your power supply. This design features robust reverse-power protection and therefore shouldn't suffer any damage if you get this wrong, but it won't *work* on reversed power and that will make the rest of this procedure difficult to complete. (If you do get this wrong the PTC fuses will get very hot, so please mind your fingers once again when you unplug the module.)

2) Once it powers up, the first test is always "Fire/No Fire." This is a very simple test: if there is smoke or fire present upon powering up, it's a fail. If there is not, it's a pass and you may proceed to the second test.

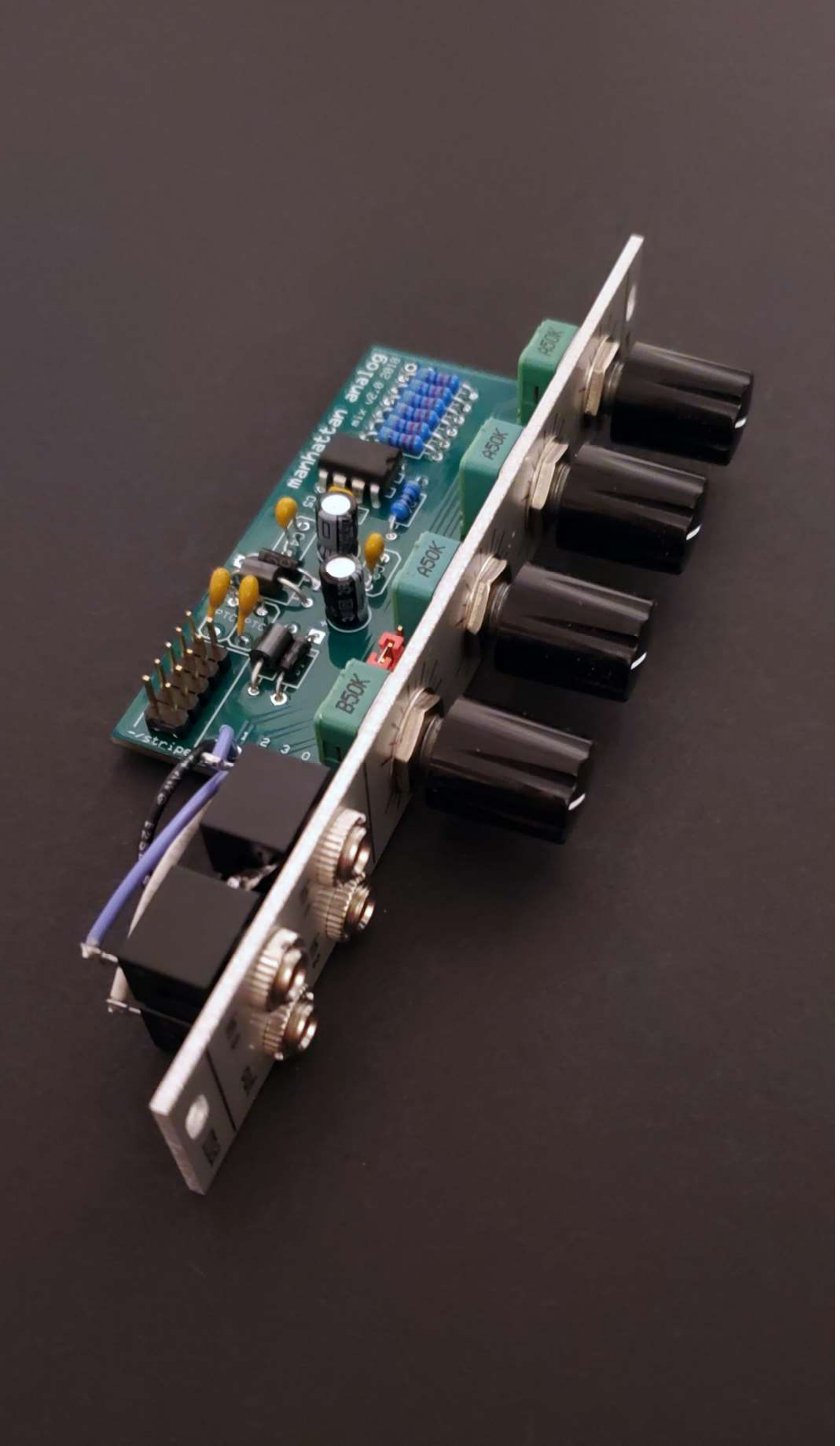
If there *is* smoke (fire is admittedly unlikely,) remove the module from the power supply immediately and let it cool down before troubleshooting the issue. This situation will almost certainly require some replacement parts and repair work before you can move forward with the build. When you do get it all sorted, move on to the second test.

3) The second test is to touch the two PTC resistors. Do so carefully - if there is a problem here they will, again, get very hot. If they're not hot, you pass and can proceed to testing.

If one or both of them is hot, that means there's likely a short somewhere on the corresponding power rail(s.) Doublecheck the orientation of the 1N4001 diodes & ICs one more time and/or check again for stray solder bridges. Once this issue is identified and resolved, proceed to trimming.

4) To test the module, first set all pots to full CCW. Plug the output into a monitor of some kind and plug an audio source into the **in 1** jack. Turn the **in 1** pot to 12:00 and then turn the output pot CW. You should hear your signal come in. Swap the audio signal to the **in 2** jack and turn the **in 2** pot CW. You should again hear your signal come in. Repeat this once more for channel 3. If everything goes as planned, congratulations, you're done! If you've chosen to test in Bipolar mode, I'll trust you to understand how the output control works and to adjust this procedure accordingly.

If your build does not work at this point, proceed to Step 6 for some troubleshooting tips, although in this module there aren't many.



Step 6

Troubleshooting tips:

Most of what happens in this thing happens in the opamp, so if you place all your parts correctly and don’t make any soldering mistakes, things will rarely go too far wrong. I don’t have any of these in my “bin of shame,” which says something. Every once in a while I’ll install the chip backward, so try not to do that. The other thing to watch for is mis-wiring the jacks. And make sure you haven’t put the 470R among the 47ks.

Note that any issue which only occurs on one input channel must be somewhere in the jack, wiring, pot, or input resistor. Once the signals hit the opamp they are summed together and a problem further downstream will occur on all channels equally.

Please check over all of your soldering and reflow any joints that seem even remotely suspect. Err on the side of caution, it doesn’t hurt anything to reflow a solder joint. You may need to use fresh flux, which may necessitate another cleaning and drying cycle, but do make sure everything is soldered cleanly. This is a common beginner mistake and a common source of faults in the customer builds I’ve had to fix in-house. It’s also the easiest problem to fix.

The Manhattan Analog “Working Module” Commitment:

Every once in a while you’ll have a build that looks good, is built well, uses known-good parts and... still just won’t work. Don’t panic. It’s uncommon, but PCB manufacturing faults are real and most likely that’s what you’re dealing with. It happens to me too, every couple hundred builds or so. In the event you simply cannot find the issue with your build, I’m here to help. Please get in touch with me through the contact form at www.manhattananalog.com. (And remember to check your spam filter for the reply if you don’t hear from me in a couple days.) I can also be reached on the Mod Wiggler forums. DMs are welcome.

I’m happy to have a look at any DIY build and help troubleshoot/fix it, even if that means having you send it to me. If you’re sure the PCB itself is faulty, I’ll gladly send a replacement. Whatever it takes, the end result should be you having a working module, and I’ll make sure we get there.