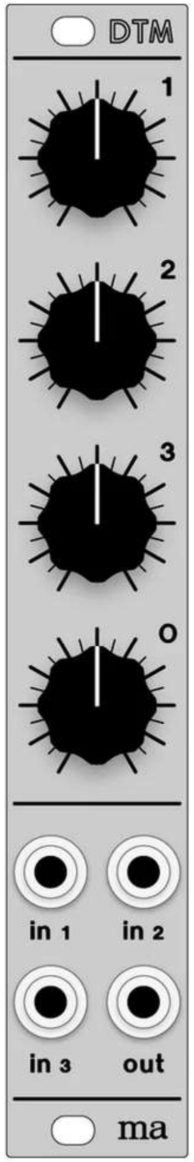
DISCRETE TRANSISTOR MIXER (DTM)



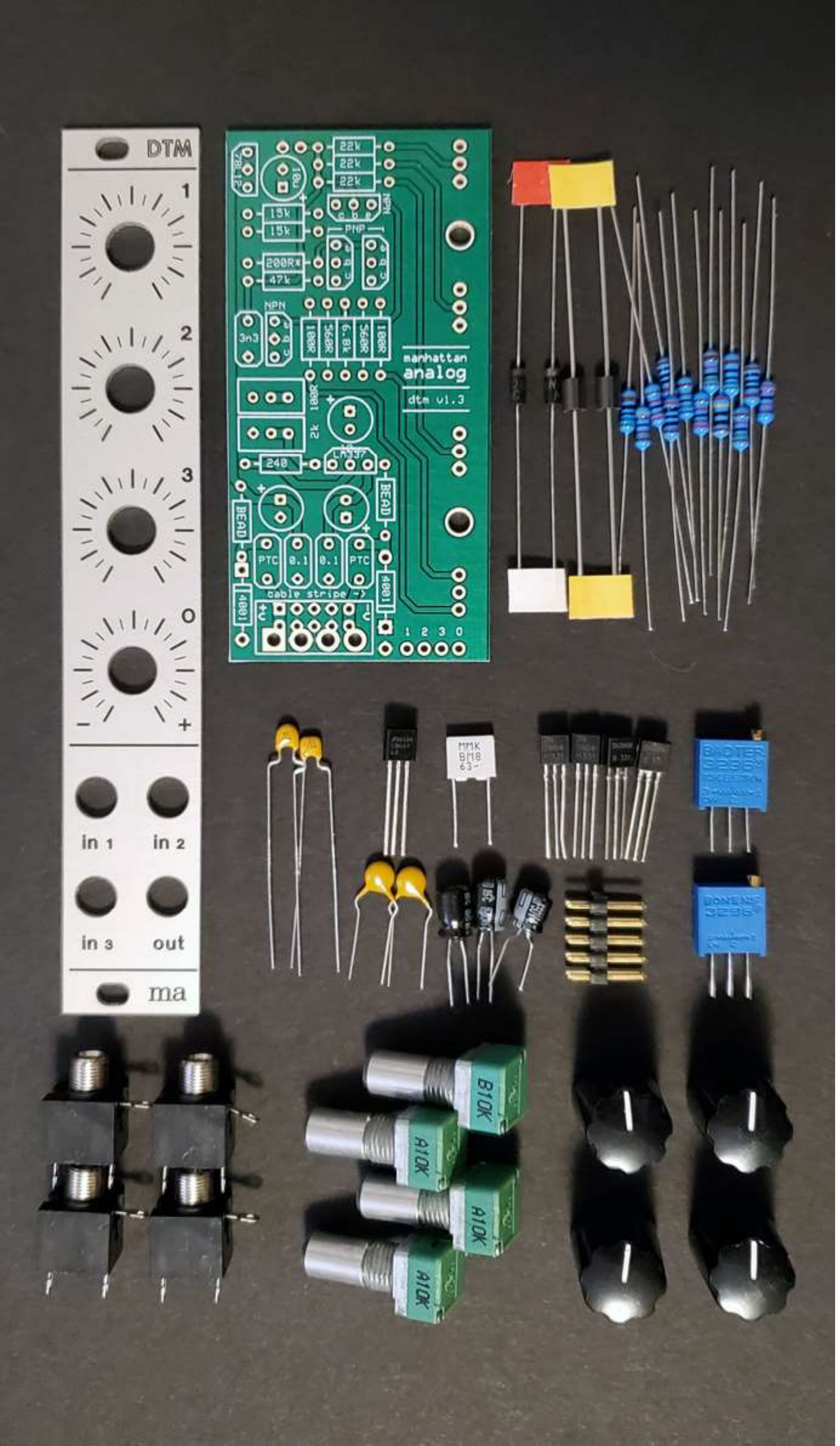
Module Documentation & DIY Build Guide, *PCB v1.3*

The Manhattan Analog Discrete Transistor Mixer (DTM) is a Eurorack format, 3-input audio and control voltage mixer based on one of the classic modular synth mixers of the 1960s, the Moog CP3.

The characteristic feature of this design is a warm, asymmetrical overdrive/distortion. This is accessed by input amplitude, so either dial back the input attenuators to keep your signals clean - which the DTM does quite well - or push them into saturation and clipping at the clockwise end. The output panel control is bipolar with the zero point at 12:00. The positive and negative phase signals coming out of the mixer aren't just identical (albeit inverse) copies of one another, you'll find a slightly different saturation character between the two swings. The original carried these signals on separate jacks, but providing both in the 4hp form factor dictated this solution instead.

Builders, please note that the character of the saturation effect will change depending on the particular transistors you use. I've marked the PCB footprints with the Base, Collector, and Emitter pins so you can readily employ any alternative parts you prefer, even if you have to do some leg-bending tricks to get there. The original is shown as using 2N3392 and 2N4048, and I've used those for years in the retail version, but I've built them with several different small signal NPN/PNP sets and never had one fail to sound good. You may find that you need to adjust the 200R* resistor's value in order to trim the bias correctly for the parts you choose. (The 3392/4058 sets, for example, need a 120-150R in that position.)

For the bill of materials, build instructions, and testing & trimming procedures, please refer to the rest of this document. For those of you not building this board for use in the Eurorack format, any necessary changes relevant to your power situation are covered in the instructions where appropriate. Take your time, be safe, and enjoy the build!



Bill of Materials

Resistors:
[2] 100R
[1] 200R*
[1] 240R
[2] 560R
[1] 6.8k
[2] 15k
[3] 22k
[1] 47k

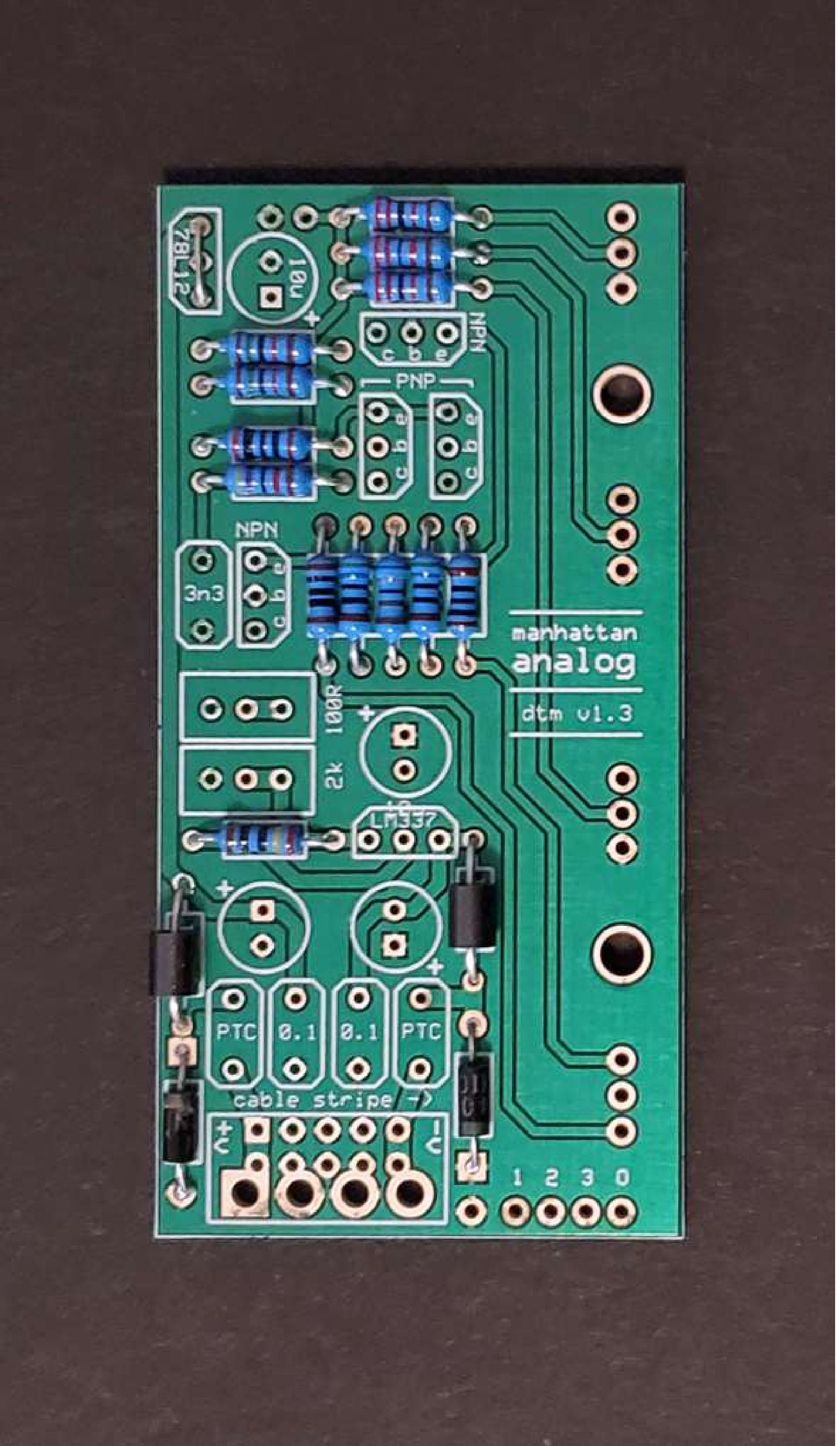
Capacitors:
[2] 10/22uF electrolytic
[2] 0.1uF ceramic
[1] 3.3nF poly film
[1] 10uF electrolytic
[1] *4.7/10uF electrolytic [Optional, only install if you use the +12V regulator]*

Pots:
[3] 10-25k audio, 9mm Alpha or similar
[1] 10-25k linear, 9mm Alpha or similar
[1] 2k multiturn trimpot, Bourns 3396 or similar
[1] 100R multiturn trimpot, Bourns 3396 or similar

Transistors/ICs:
[1] LM337LZ - be sure to get the TO-92 package
[2] 2N3904 (or similar NPN) - Matched pair (optional)
[2] 2N3906 (or similar PNP) - Matched pair (optional)
[1] *LM78L12 [Optional, only install for +/-15V supply voltage operation.]*

Miscellaneous:
[1] 10-pin, 2x5, 0.100" header, male (Eurorack power) **OR** 4-pin MTA-156 header (MOTM power)
[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.

* - If you choose different transistor pairs you may need to adjust the value of this resistor in order to properly trim the bias.



Step 1

Parts Used:

Resistors:
[2] 100R
[1] 200R*
[1] 240R
[2] 560R
[1] 6.8k
[2] 15k
[3] 22k
[1] 47k

Miscellaneous:
[2] Ferrite bead (can substitute 10R resistor)
[2] 1N4001 diode

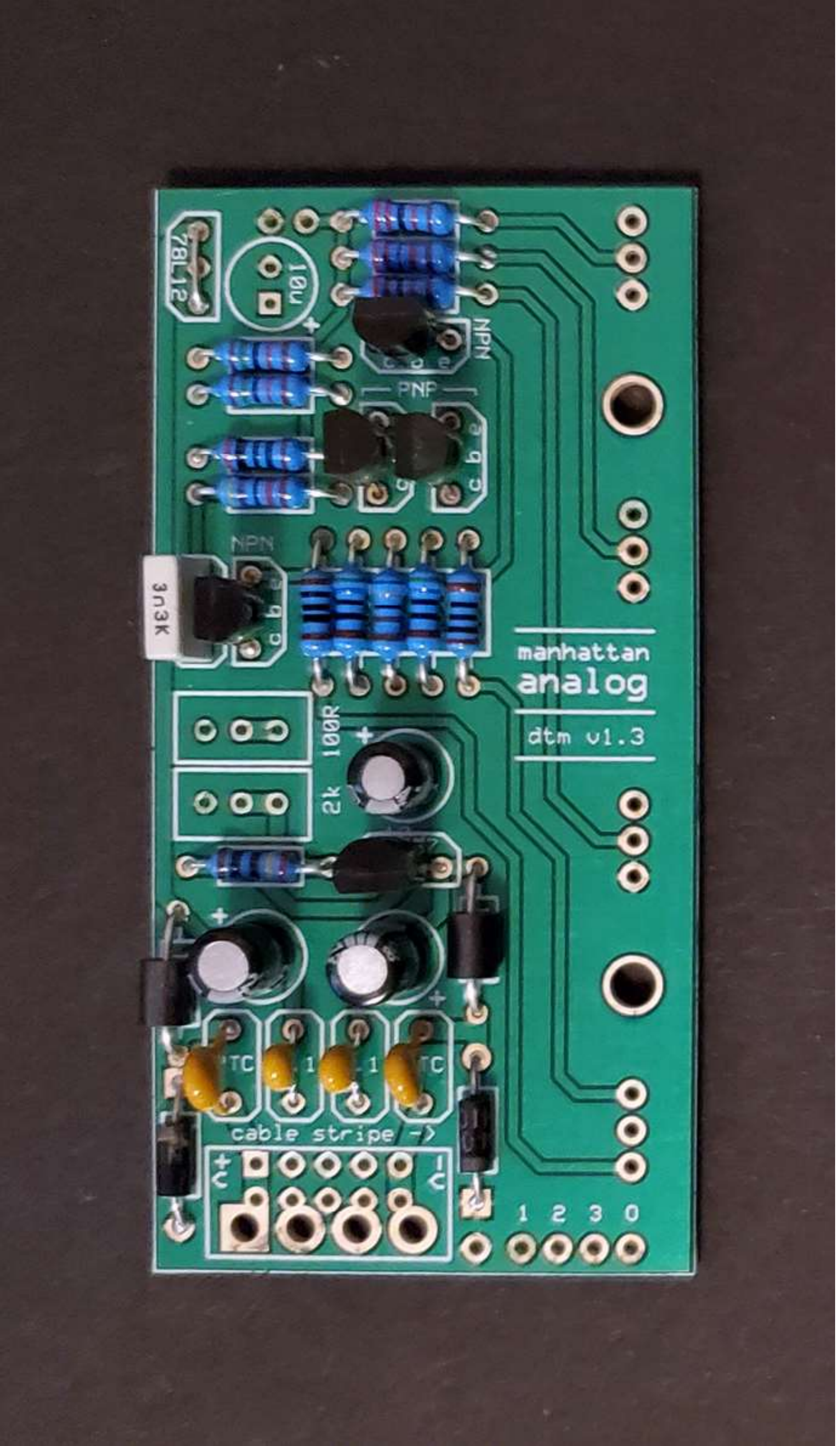
Procedure:

1) If you're building this for use with a +/-12V power supply, install a wire jumper across the outer pads of the LM78L12 footprint, as shown in the photo, to bypass the regulator. If you're building this for a use with a +/-15V power supply, do not install the jumper and pay attention to the extra instructions in Step 2.

2) Populate the PCB with the lowest-profile components - the resistors, diodes, and ferrite beads. You may find it easy to do them all at once or it may be easier to do them in a couple of batches.

3) Doublecheck resistor placement (note the stripe codes) and especially the orientation of the two 1N4001 diodes. If you know you're going to need to use a different value for the 200R* resistor, make that change now. If you're not sure about it yet, just pop in the 200R but be ready to revisit this in the future.

4) Solder all leads, clip them with a pair of flush-cutters, inspect your work to check for cold joints or stray solder, and when you're satisfied, proceed to Step 2.



Step 2

Parts Used:

Capacitors:
[2] 10/22uF electrolytic
[2] 0.1uF ceramic
[1] 3.3nF poly film
[1] 10uF electrolytic
[1] *4.7/10uF electrolytic [Optional, only install if you use the +12V regulator]*

Transistors/ICs:
[1] LM337LZ - be sure to get the TO-92 package
[2] 2N3904 (or similar NPN) - Matched pair (optional)
[2] 2N3906 (or similar PNP) - Matched pair (optional)
[1] *LM78L12 [Optional, only install for +/-15V supply voltage operation.]*

Miscellaneous:
[2] PTC resettable fuse (.050 hold, .100 trip)

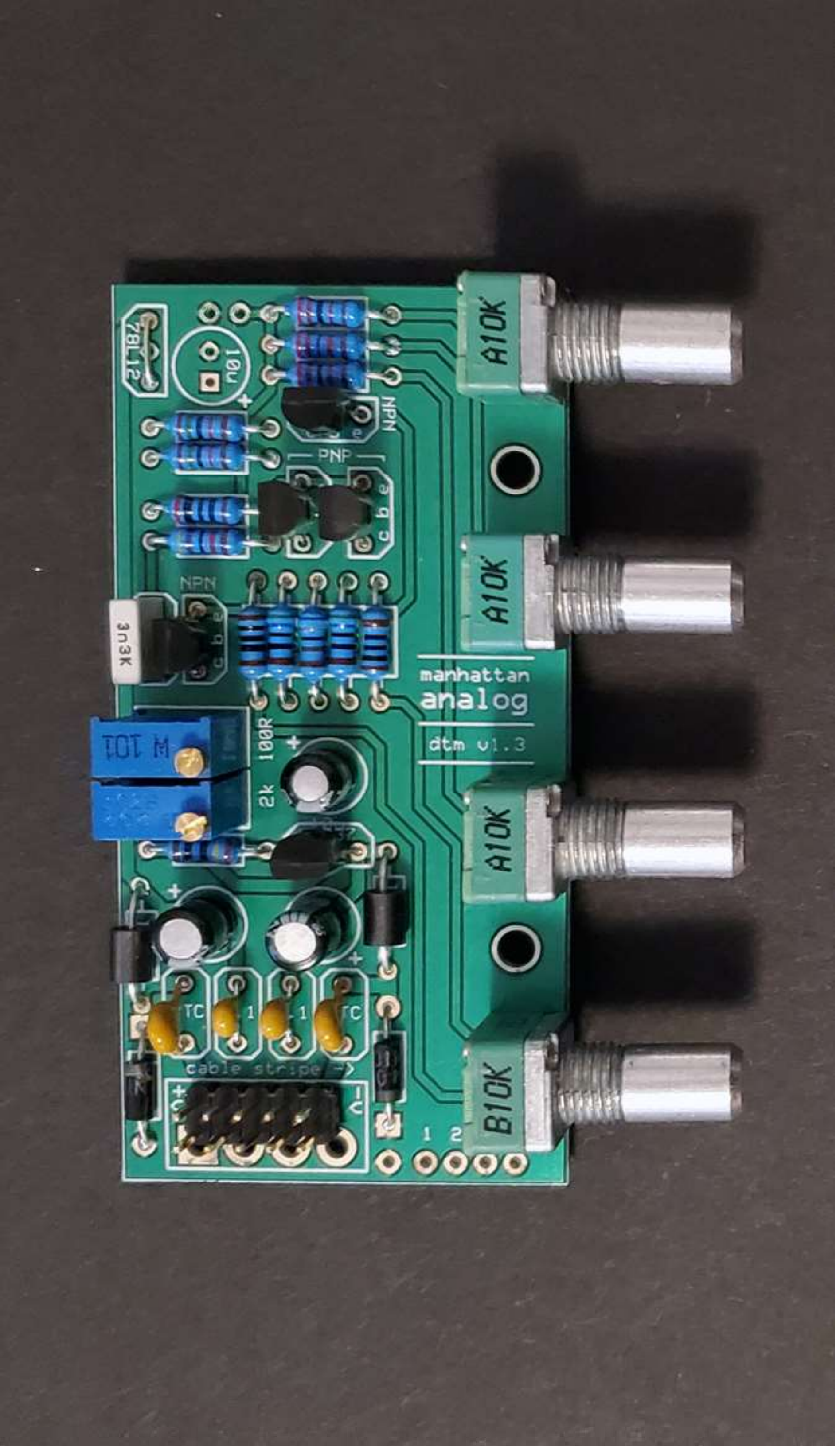
Procedure:

1) Populate the PCB with the mid-height components. 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) If you're building this for use in a system with a +/-15V power supply, install the optional LM78L12 regulator and its associated decoupling capacitor now. *(Not pictured.)*

3) Doublecheck the transistors and regulator(s) to make sure they're oriented correctly and in the correct spots. Doublecheck the orientation of the electrolytic caps to be sure their polarity is correct. If you've used alternative transistor pairs with different pinouts, make sure you've done all your leg twisting/bending correctly and the C, B, and E pins are where they need to be with respect to the silkscreen on the PCB.

3) Solder all leads, clip them with a pair of flush-cutters, inspect your work to check for cold joints or stray solder, and when you're satisfied, proceed to Step 3.



Step 3

Parts Used:

Pots:
[3] 10-25k audio, 9mm Alpha or similar
[1] 10-25k linear, 9mm Alpha or similar
[1] 2k multiturn trimpot, Bourns 3396 or similar
[1] 100R multiturn trimpot, Bourns 3396 or similar

Miscellaneous:
[1] 10-pin .100 header, male (Euro power) **OR** 4-pin MTA-156 header (MOTM power)

Procedure:

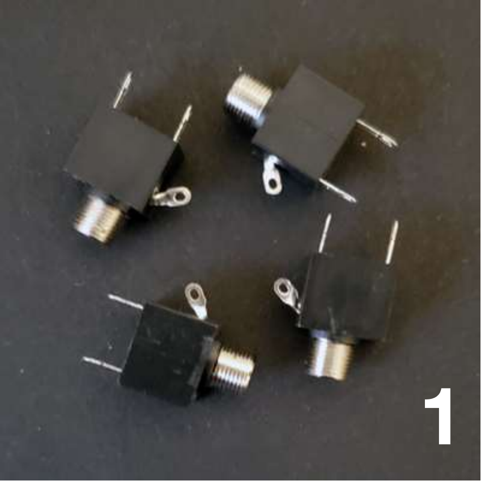
1) Populate the PCB with the tall components and the panel pots. Don't forget your tall caps from Step 2 if you had any left over.

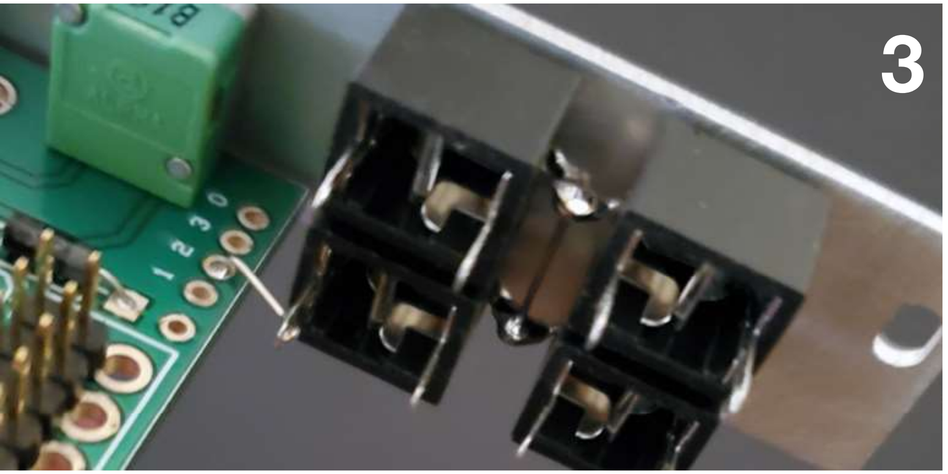
2) Tack each pot down for now by just the center pin. Do the same with the power header - tack it down using just one of the end pins.

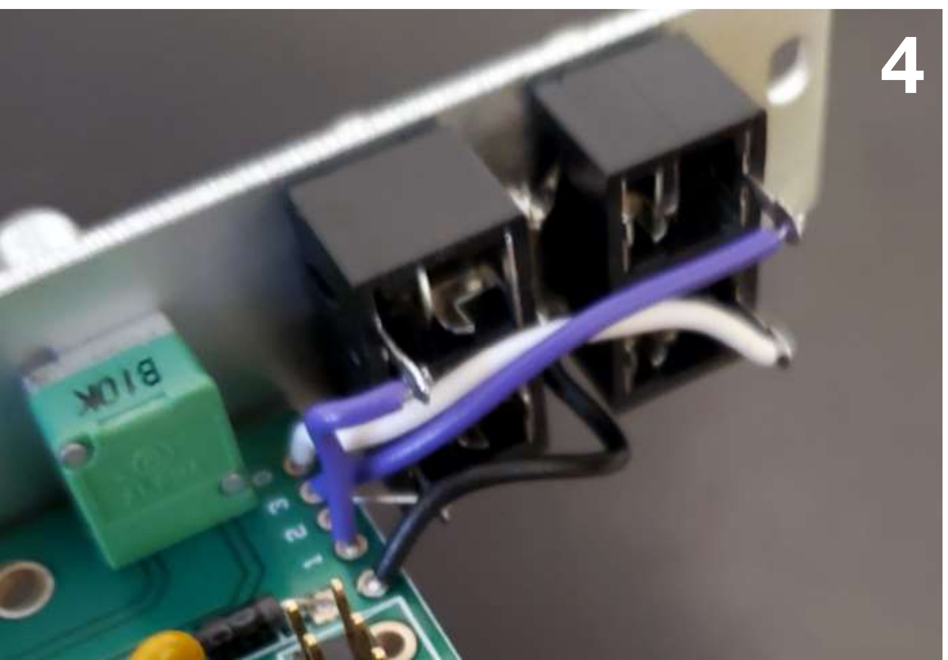
3) Solder all component leads, clip them with a pair of flush-cutters, and inspect your work to check for cold joints or stray solder.

4) 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 moderate 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 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.

5) Do the same thing with the power header - make sure it's seated securely, flush, and perpendicular to the PCB. This will also be a mechanical connection (to the power cable) so it's important to get this one right. Plus, it keeps your power cable from installing a wonky angle and looking weird. **Again, mind your fingers near the iron.** Once you're satisfied, solder the remaining 9 pins to complete the power header and proceed to Step 4.







Step 4

Parts Used:

Miscellaneous:
[4] 3.5mm mono N/C jack, Kobiconn 16PJ138 or Thonkiconn style
[4] 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.

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.)

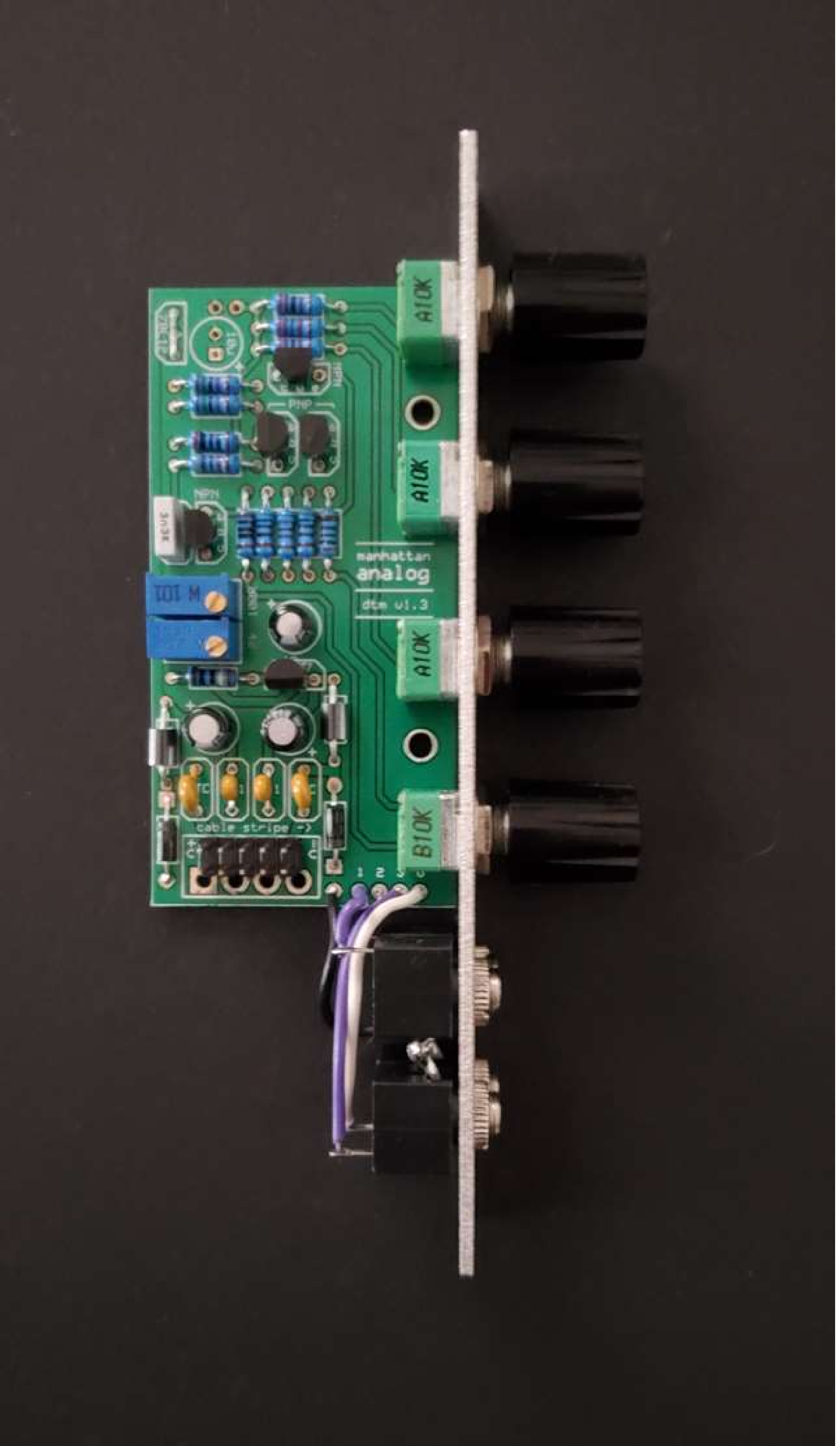
4) Install the completed PCB assembly from Step 3 onto the panel, taking care not to scratch the metal as you tighten down the nuts on the pots. Now is when you finally finish soldering the CW and CCW pins, but first, reflow each of the middle pins one more time - this will relieve any mechanical stress 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 8mm length on one side, and use that to connect the **IN 2** jack to the pad marked **2** 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 (~30-35mm) from the common ground wire you just installed in the jacks to the unmarked pad next to the **1** on the PCB. Then run wires from **IN 1** to pad **1**, **IN 3** to pad **3**, and **OUT** to pad **O** to complete the panel I/O connections. Again, see photo 4.

6) Install the knobs onto the pot shafts.

7) If you've used solder with water-soluble flux, or if you want to use a solvent to clean up rosin flux, please inspect your work one more time for faults or stray solder and then do your cleanup. Give the board 12-24 hours to dry completely (don't rush it - powering up a wet module is counterproductive) and then proceed to Step 5 for testing and trimming.

8) If you've used solder with a no-clean flux of some kind, just inspect your work for faults or stray solder one last time and then proceed to Step 5 for testing and trimming.



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 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.)

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 trimming.

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 one more time and/or check again for stray solder bridges. Once this issue is identified and resolved, proceed to trimming.

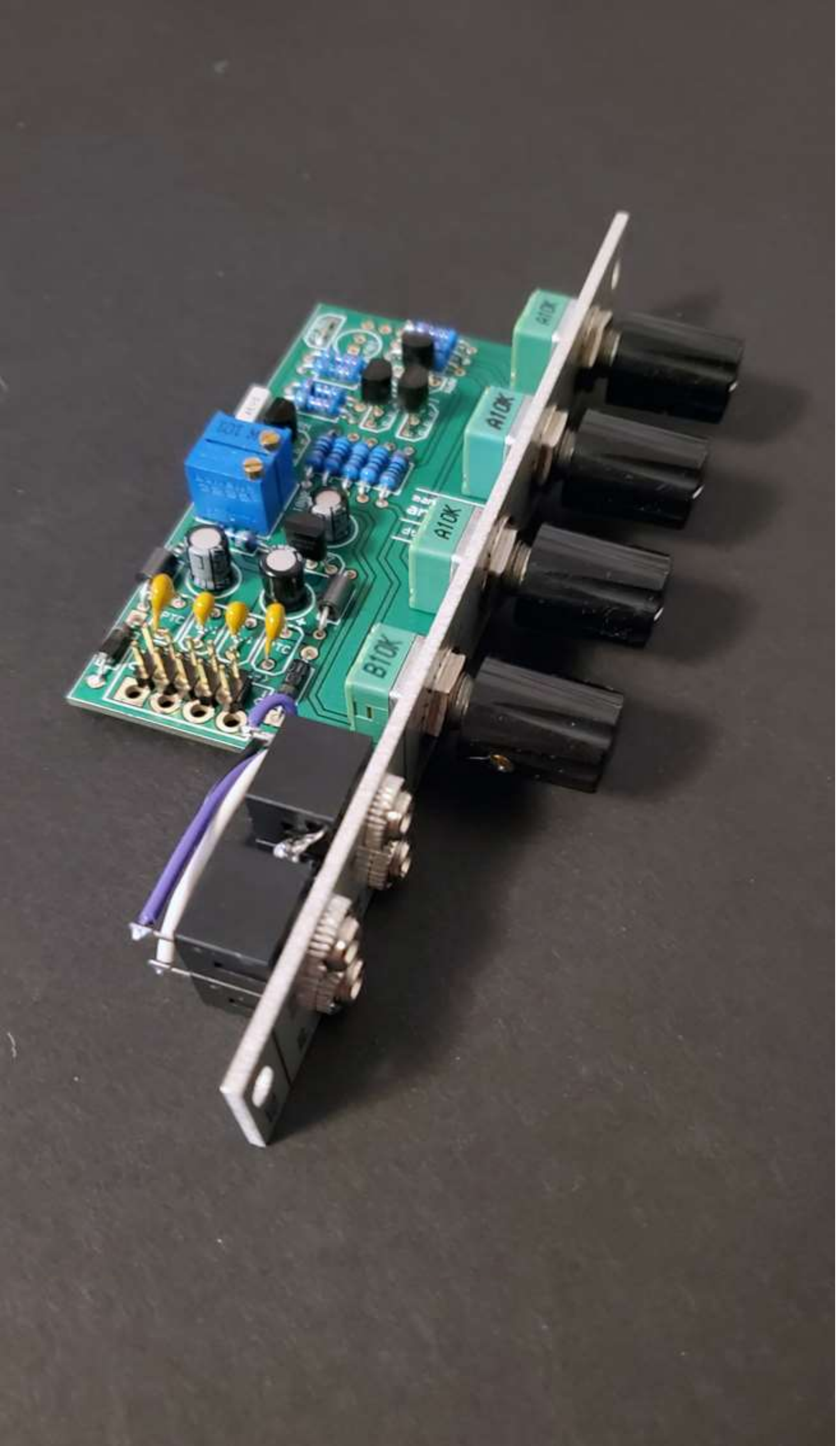
4) First, set up the -6V power rail. With your multimeter, probe the center pin of the LM337 with the red lead. (Black lead to any ground point.) Adjust the 2k trimpot to get -6.00V.

5) Next you need to trim the output DC offset voltage. With no inputs in use, set the O control pot to 12:00 and probe the tip of the output jack with the red lead. (Black lead to ground.) Adjust the 100R trimpot until you get 0.00V, or as close as you're comfortable with. (It may drift a bit. Don't drive yourself crazy.) If you cannot achieve 0V within the range of the trimmer, you probably need to change the value of the 200R* resistor to suit your transistors. Please revisit Step 1.

6) Finally, test the audio. Set all input pots to full-CCW and the output pot to full CW. Plug a VCO or some other audio signal into input 1 and turn the channel 1 input pot clockwise. You should hear your signal. Repeat this process for the other two inputs. With audio still playing, turn the output pot to 12:00 - this should silence, or nearly silence, the output. Then turn it fully CCW, which should bring the audio back in. (With its phase reversed, but you won't "hear" that. If you have an oscilloscope of some kind you can verify it there.)

If all of these things happen as described, congratulations, the module works and you're done!

If you have exhausted your troubleshooting leads and your module still does not work, please proceed to Step 6 for a few hints on troubleshooting a non-working build.



Step 6

Troubleshooting tips:

This is a very simple design with few components, so there aren't too many things that can go wrong. This is helpful when it comes to troubleshooting a nonworking build. Here are a few thoughts based on the years of experience I have building these:

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.

Any signal issue that affects only one input channel has to be jack/pot/input resistor/wiring/PCB-related as the three signals are summed together after that point.

The most common mistake I make involves incorrect transistor installation. If you're not getting any sound out of the mixer, double- and then triple-check your transistor pinouts and make sure you have them installed correctly. This is by far the most common mistake for people who choose alternative transistors. Also be sure you didn't get the NPN pair where the PNP goes and vice-versa, or that you didn't mismatch the pairs. (This is what I absentmindedly do sometimes.)

If you are unable to fully trim the offset voltage it's likely due to the 200R* resistor being out of range for your transistor pairs, as covered earlier in this guide. Please refer back to Step 1 for sorting that out.

The Manhattan Analog "Working Module" Commitment:

Every once in a while you'll have a build that looks good, tests good, 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 contact 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 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.