VOLTAGE CONTROLLED AMPLIFIER (VCA)

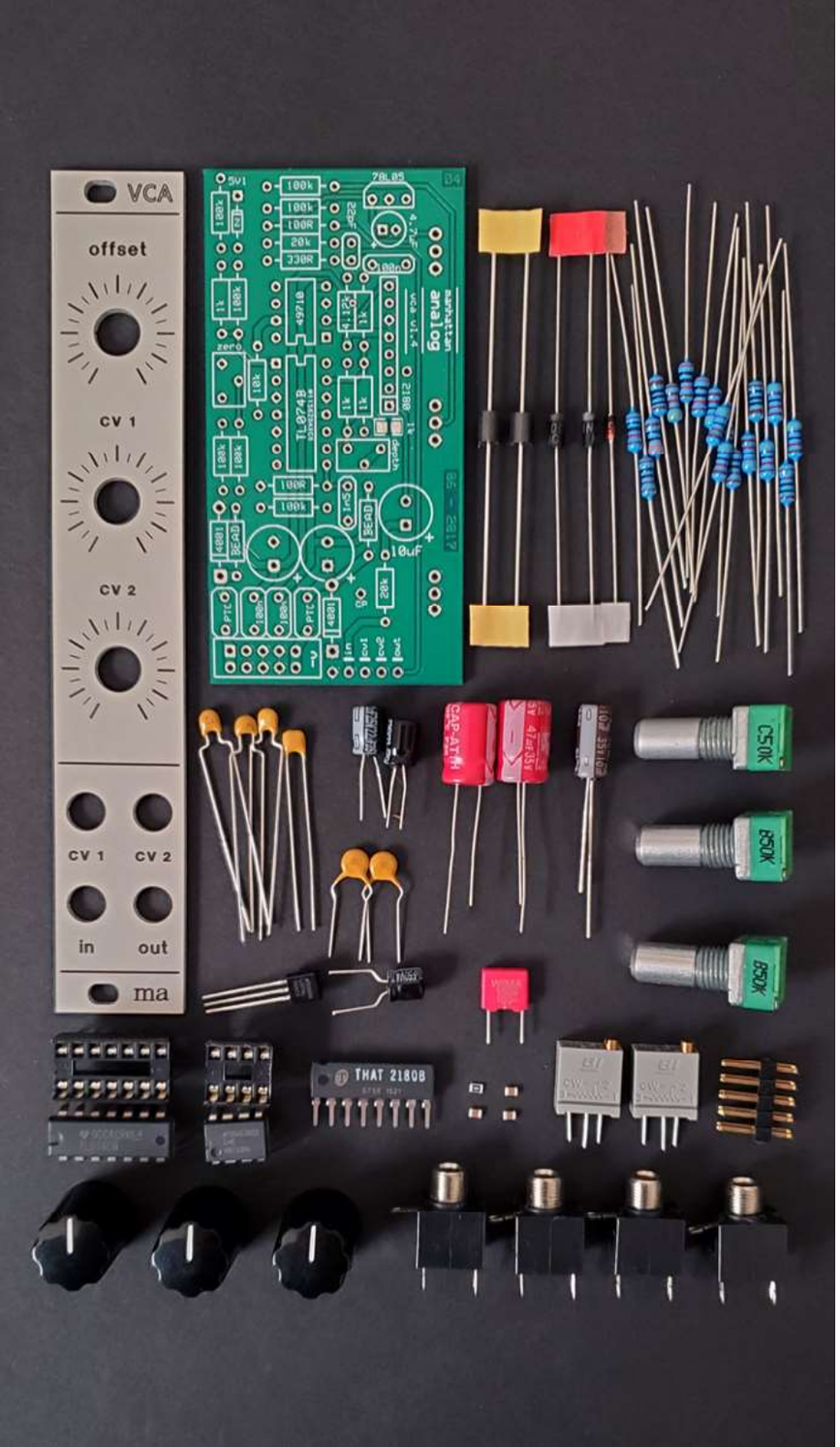
Module Documentation & DIY Build Guide, *PCB v1.4*

The Manhattan Analog VCA is a Eurorack format VCA module based around a THAT2180 integrated VCA chip. It features one audio input, two CV inputs, and a manual offset control.

The characteristic feature of this design is a crystal clear audio signal path, and it's hard to do that job much better than the 2180 does. The signal input is AC coupled, so this is an audio-only design. Both the CV1 and CV2 controls have an exponential response with an input range of -5V to +5V. The 5.1V zener diode is there to enforce that limit. The THAT2180 distorts pretty quickly when you try to push it past unity gain, which you can achieve in this module if you trim it so you get some positive amplification. (Adjust the **zero** trimpot to get a small negative voltage instead of 0.000V.) The distortion character of the VCA chip is pretty crunchy, but anything can work in the right patch... give it a go if you're curious.

The manual offset control uses a reverse-audio taper pot to approximate a linear response, which gives a control feel that I personally find more comfortable to work with than an exponential one. There's absolutely nothing stopping you from using a linear 50k pot in that position if you disagree, though. (It's an easier part to source.) There's also no difference in the build or setup procedure if you decide to use a THAT2180A or C, but I do feel like the B grade part is the best choice for this application. Finally, if you want to get fancy with an audio-grade electrolytic cap in the signal path, the 10uF electro is the one that does the AC coupling at the input. I use Elna Silmic II in all of my builds. It's probably overkill, but they're also not expensive.

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

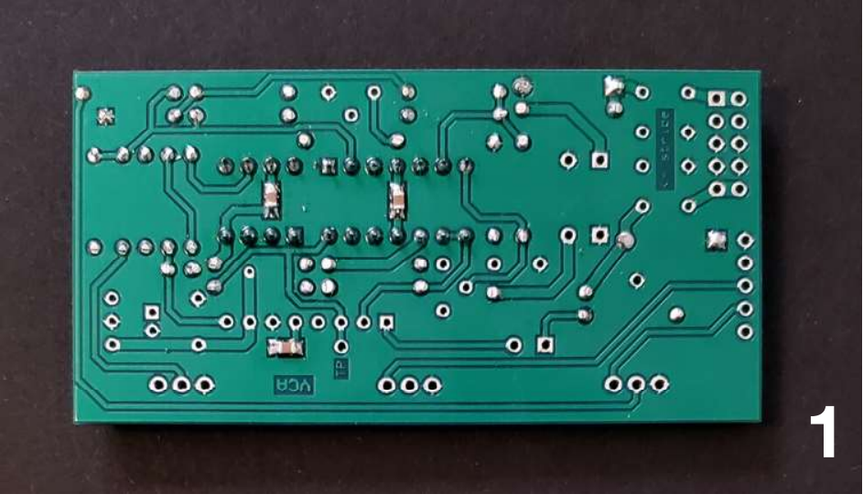
Resistors:
[7] 47k - PCB silkscreen shows 100k, and that works, but preferable to go with 47k instead for all positions
[2] 20k
[1] 10k
[1] 4.12k
[4] 1k
[1] 330R
[2] 100R
[1] 1K - 0805 SMT - Tempco preferable, but they're nearly impossible to source these days, so a regular 1k is fine.

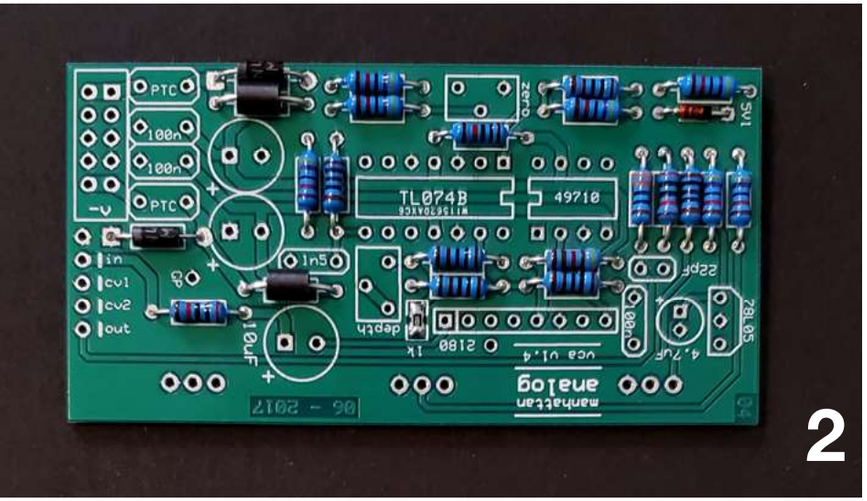
Capacitors:
[1] 10uF electrolytic - this one's in the Audio path - can step up to an Elna Silmic (pictured) or similar audio-grade cap if desired
[1] 4.7uF electrolytic
[1] 1.5nF poly film
[1] 22pF ceramic
[2] 47uF electrolytic
[3] 0.1uF ceramic
[3] 0.1uF ceramic - 0805 SMT

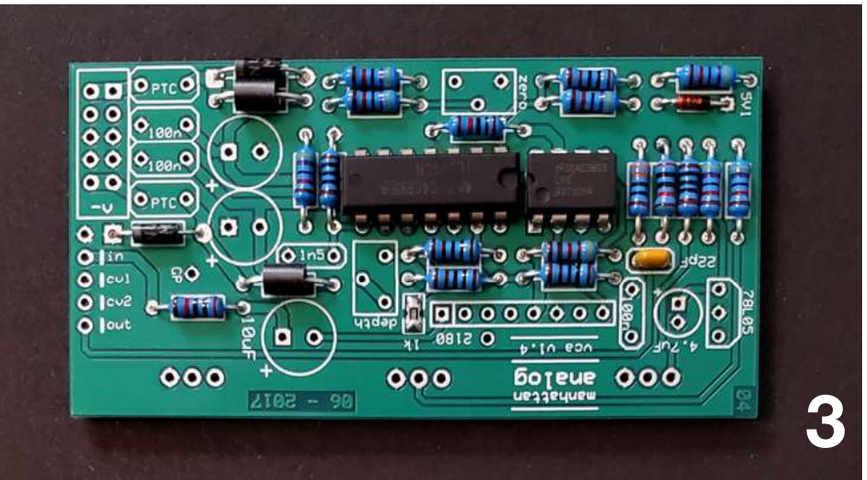
Pots:
[2] 20k multiturn trimpot, Bourns 3396 or similar
[1] 50k reverse audio, 9mm Alpha or similar
[2] 100k linear, 9mm Alpha or similar

Transistors/ICs:
[1] 5.1V Zener Diode
[1] TL074B
[1] LME49710
[1] THAT2180B - A or C grade parts will also work fine
[1] LM78L05

Miscellaneous:
[1] 10-pin, 2x5, 0.100" header, male
[1] 14 pin IC socket (optional)
[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
[3] Davies 1900H-style knob, or any other ~12.5mm-diameter knobs you prefer.







Step 1

Parts Used:

Resistors:
[7] 47k - PCB silkscreen shows 100k, and that works, but it's preferable to go with 47k instead for all positions
[2] 20k
[1] 10k
[1] 4.12k
[4] 1k
[1] 330R
[2] 100R
[1] 1K - 0805 SMT - Tempco preferable, but they're nearly impossible to source these days, so a standard 1k is fine.

Capacitors:
[3] 0.1uF ceramic - 0805 SMT
[1] 22pF ceramic

Transistors/ICs:
[1] 5.1V zener diode
[1] TL074B
[1] LME49710

Miscellaneous:
[2] Ferrite bead (can substitute 10R resistor)
[2] 1N4001 diode
[1] 14 pin IC socket (optional)
[1] 8 pin IC socket (optional)

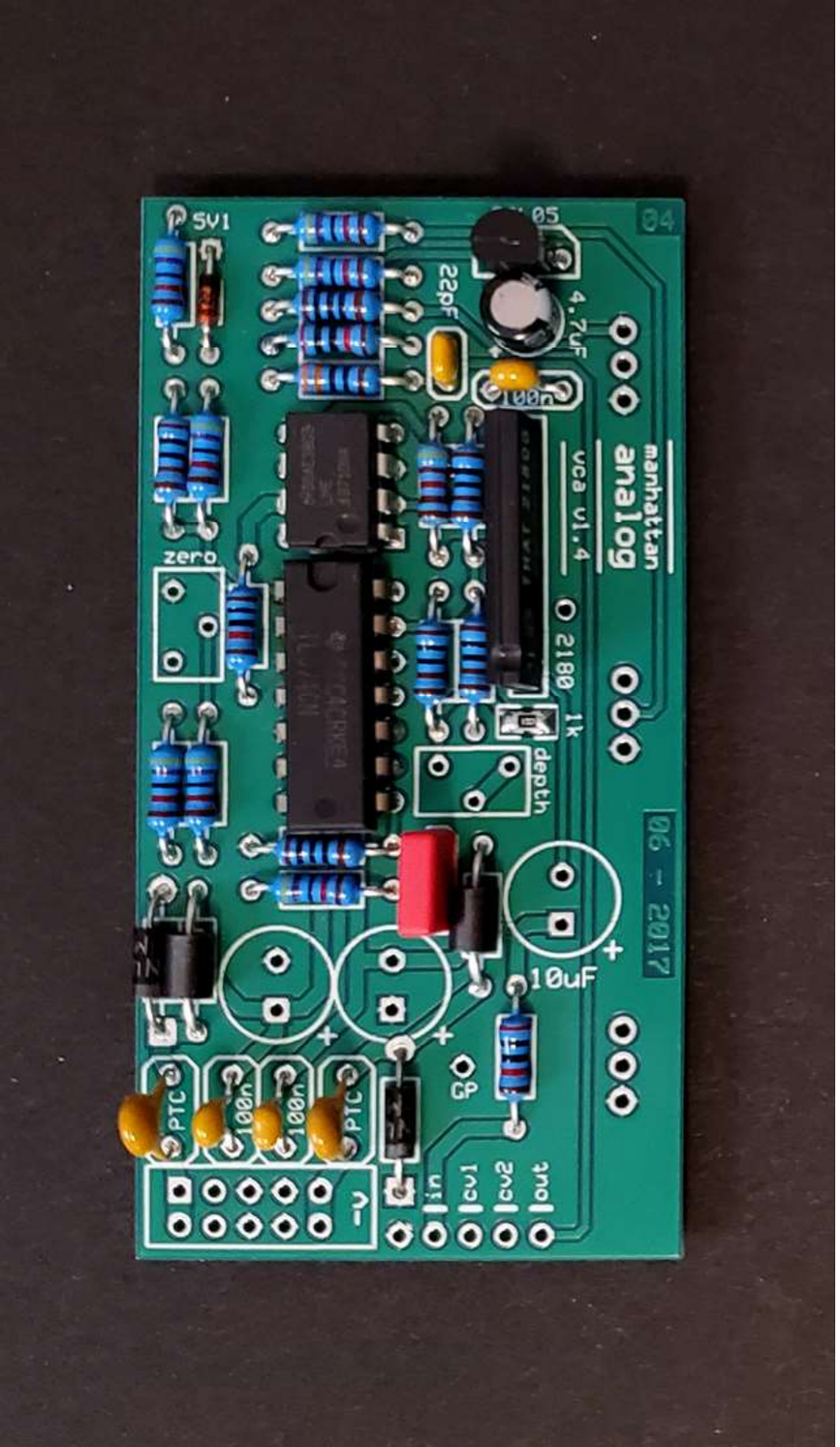
Procedure:

1) Go ahead and start with the SMT caps on the solder side of the PCB as shown in photo 1. (I know the picture shows a soldered PCB, but it's easier to do these first.) Then populate the PCB with the lowest-profile components as seen in photo 2.

2) Doublecheck resistor placement (note the stripe codes) and especially the orientation of the two 1N4001 diodes and the 5.1V zener. Solder all leads, clip them with a pair of flush-cutters, and inspect your work to check for cold joints or stray solder.

3) Pop either the ICs (pictured) or the optional IC sockets into the board, taking care to orient them correctly. As shown in photo 3, the two notches on the pin 1 ends should face one another in the center. If you're not using sockets... triplecheck things.

4) Solder one IC pin on each side of the chip/socket, then check that they're seated flush with the PCB. Adjust them if you need to by re-flowing the pins you tacked them down with and applying some downward pressure. 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:

Resistors:
[1] 1K - 0805 SMT - Tempco preferable, but they're nearly impossible to source these days, so a regular 1k is fine.

Capacitors:
[1] 10uF electrolytic - this one's in the Audio path - can step up to an Elna Silmic (pictured) or similar audio-grade cap if desired
[1] 4.7uF electrolytic
[1] 1.5nF poly film
[2] 47uF electrolytic
[3] 0.1uF ceramic

Transistors/ICs:
[1] THAT2180B - A or C grade parts will also work fine
[1] LM78L05

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

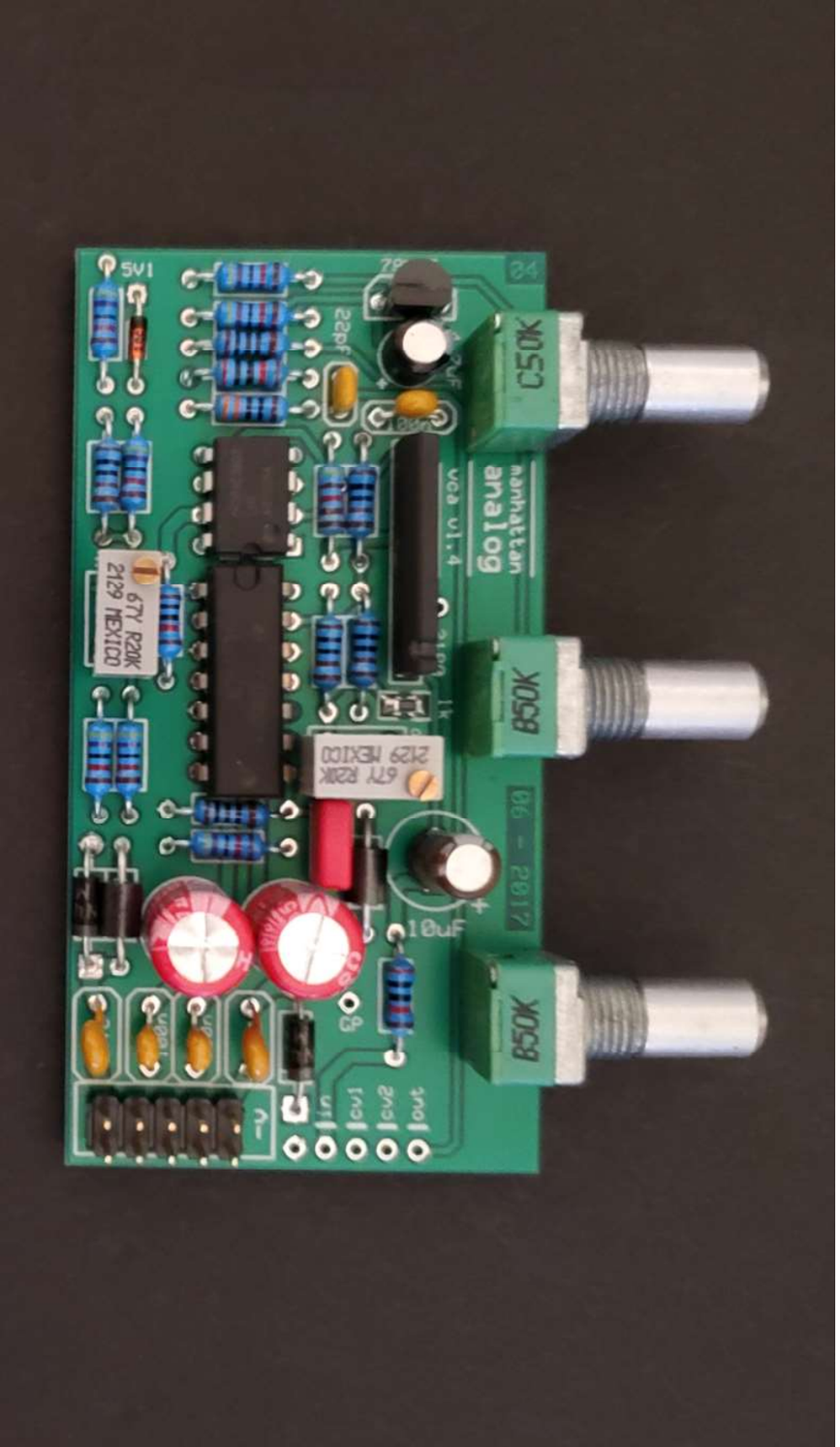
Procedure:

1) Same as in Step 1, do the 1k 0805 resistor before the other components. It's easier to access that area with your iron before the 2180 and 1.5nF cap go in.

2) 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.

3) Doublecheck the THAT2180 and LM78L05 to make sure they're oriented correctly. Take extra care with the 2180, that's an expensive one to get wrong. Doublecheck the orientation of the electrolytic caps to be sure their polarity is correct.

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:
[2] 20k multiturn trimpot, Bourns 3396 or similar
[1] 50k reverse audio, 9mm Alpha or similar
[2] 100k linear, 9mm Alpha or similar

Miscellaneous:
[1] 10-pin .100 header, male

Procedure:

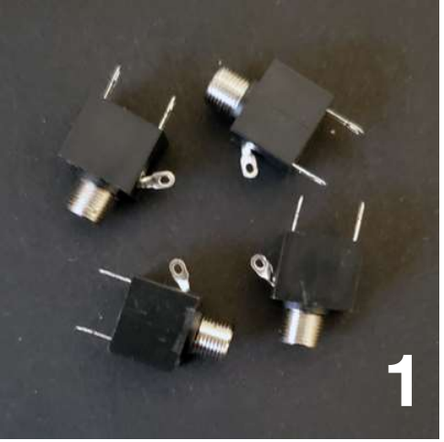
1) Populate the PCB with the tall components and the panel pots. Don't forget your tall caps from Step 2 if you had any left over.

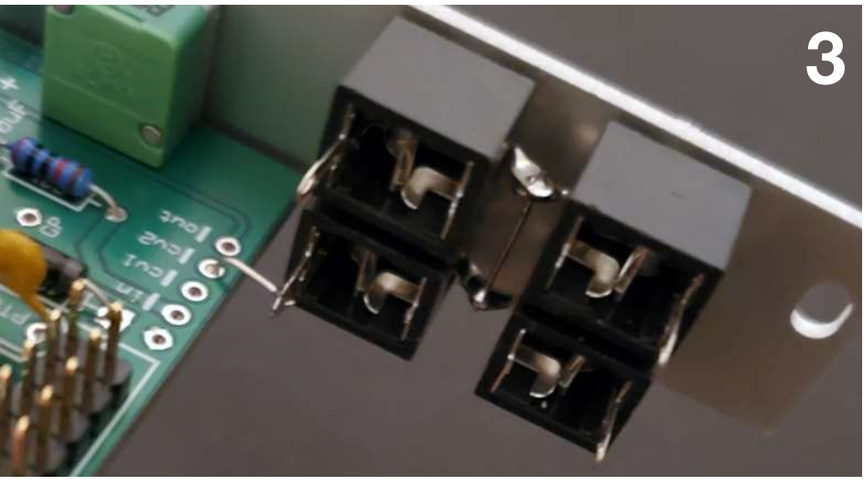
2) Tack each pot down for now by just the center pin. Do the same with the power header - tack it down using just one of the end pins.

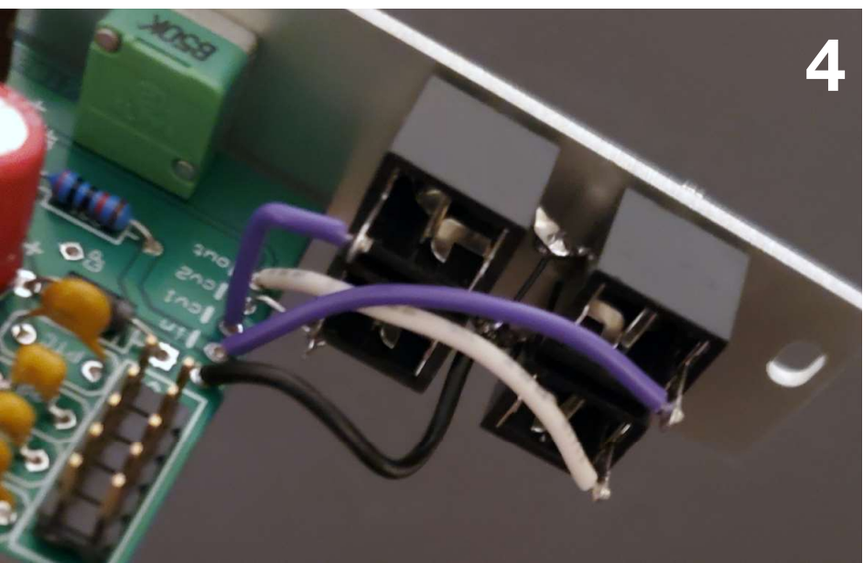
3) Solder all component leads, clip them with a pair of flush-cutters, and inspect your work to check for cold joints or stray solder.

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 three 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, with the pins perpendicular to the PCB. This will also be a mechanical connection (to the power cable) so it's equally important to get this one right. Plus, it keeps your power cable from installing at 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
[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.

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.)

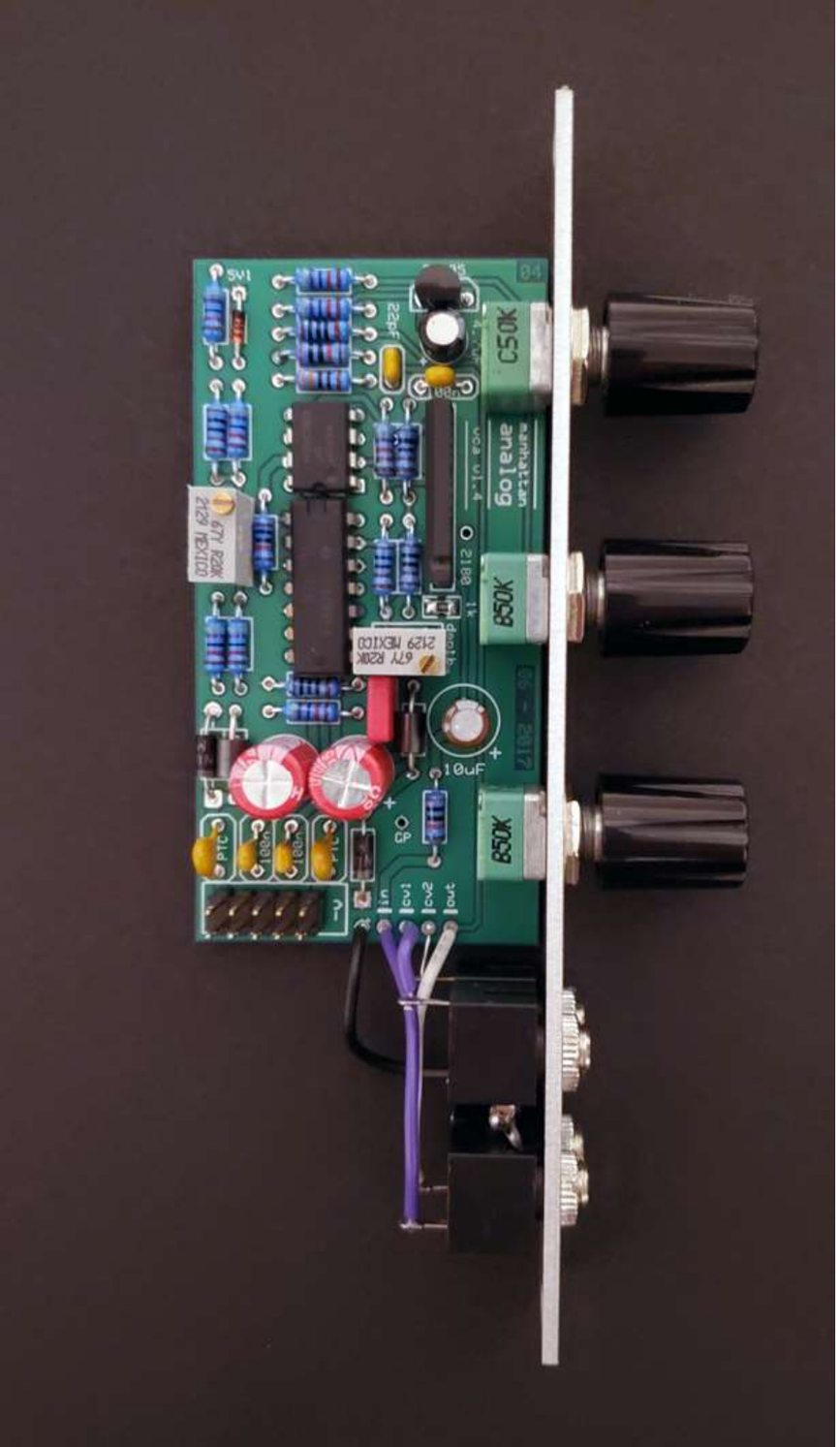
4) Install the completed PCB assembly from Step 3 onto the panel, taking care not to scratch the metal as you tighten down the nuts on the pots. Now is when you finally finish soldering the CW and CCW pins, but first, reflow each of the middle pins one more time - this will relieve any mechanical stress 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 6 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 **cv 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 (~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 the **in, out, and cv1** jacks to their corresponding pads on the PCB.

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.

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

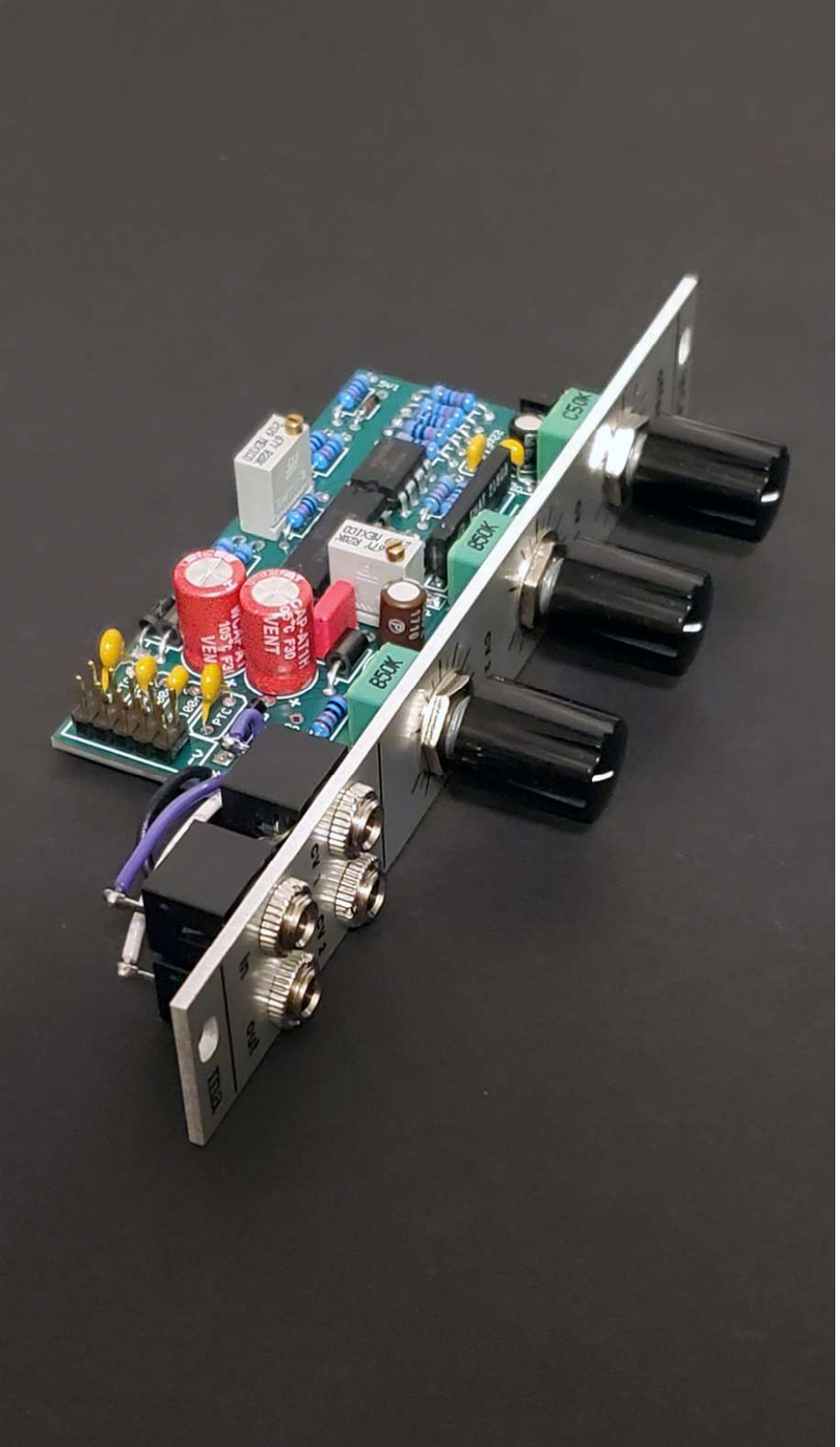
4) Set the two CV input pots to full CCW and the offset pot to full CW. Using your multimeter, probe the test point (the empty pad near pin 3 of the THAT2180) with your red lead. There's another pad near the 20k input resistor marked "GP" (*Ground Point*) for your black lead. Adjust the trimpot marked **zero** until you achieve 0.000V. Precision is important here.

5) Turn the offset pot to full CCW and probe the test point again. Now adjust the trimpot marked **depth** until you see 305-310mV. Precision is less important here, but do be sure you're in that range.

6) Finally, test the CV and audio. Set both CV pots to full CCW and the output pot to full CW. Plug an audio signal into the input and listen to the output. Turn the offset pot clockwise. You should hear your input signal come in. If you do, return the offset control to full CCW and plug a CV signal of your choice into the **cv 1** jack (an LFO works well.) Now turn the **cv 1** panel control clockwise. You should hear your audio signal come in again, this time modulated by your CV signal. If so, repeat this process with the **cv 2** input.

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:

Most of what happens in this thing happens in the ICs, 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 a chip backward, so try not to do that. The other thing to watch for is mis-wiring the jacks.

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.