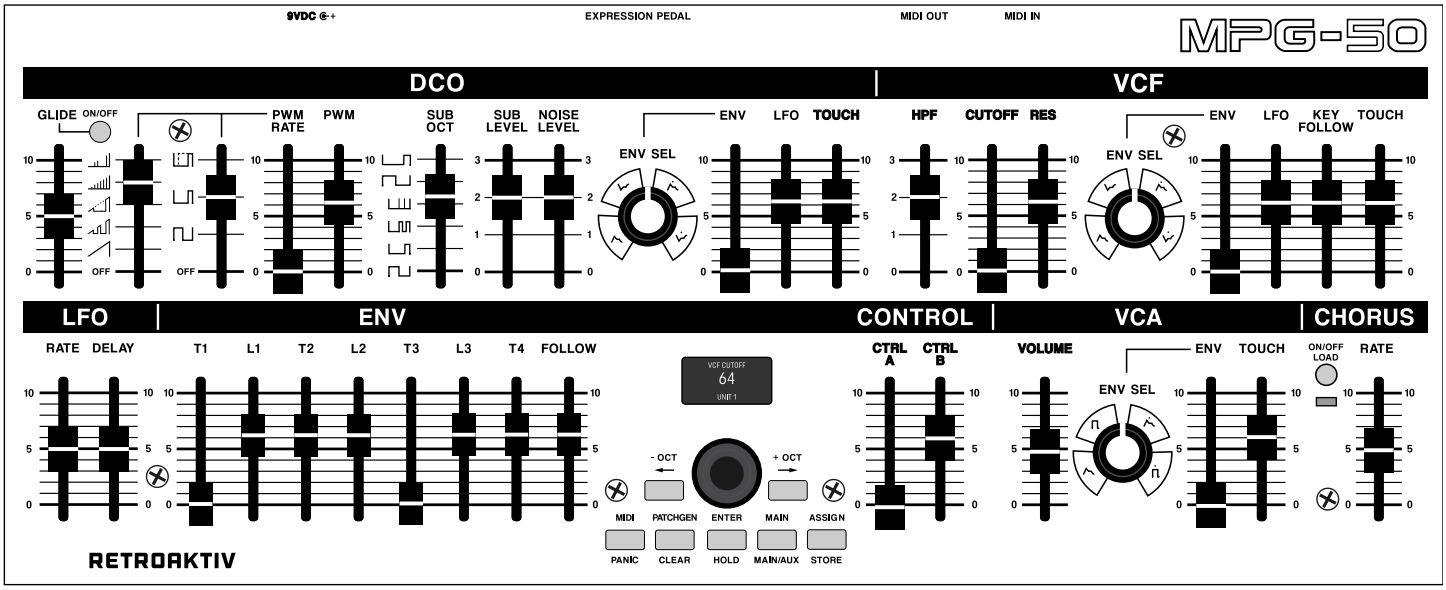
MPG-50 DIY KIT BUILD GUIDE



VERSION 1.0

This guide is intended to help guide builders through the process of assembling, programming, and testing the Retroaktiv MPG-50 controller DIY kit. This kit required soldering skills, and the ability to read schematics, a bill of materials, and identify SMD parts. If you do not have experience with these things, please seek the assistance of a person who does.

The following items are needed to assemble this kit:

- Solder
- Flux
- Multimeter or oscilloscope
- ISP programmer (USBASP for example) if you have a bare PCB DIY kit
- Philips head screwdriver
- SMD tweezers
-

When you receive your kit, the enclosure, rear plate, and front panel will be screwed together, and the PCB, and all other parts will be inside of the enclosure.

The faceplates should be washed with a soft cloth and some water, to remove any residue from offgassing of the enamel. This may not be necessary, but it is recommended. Once cleaned, the panel and enclosure parts can be set aside.

PCB Assembly:

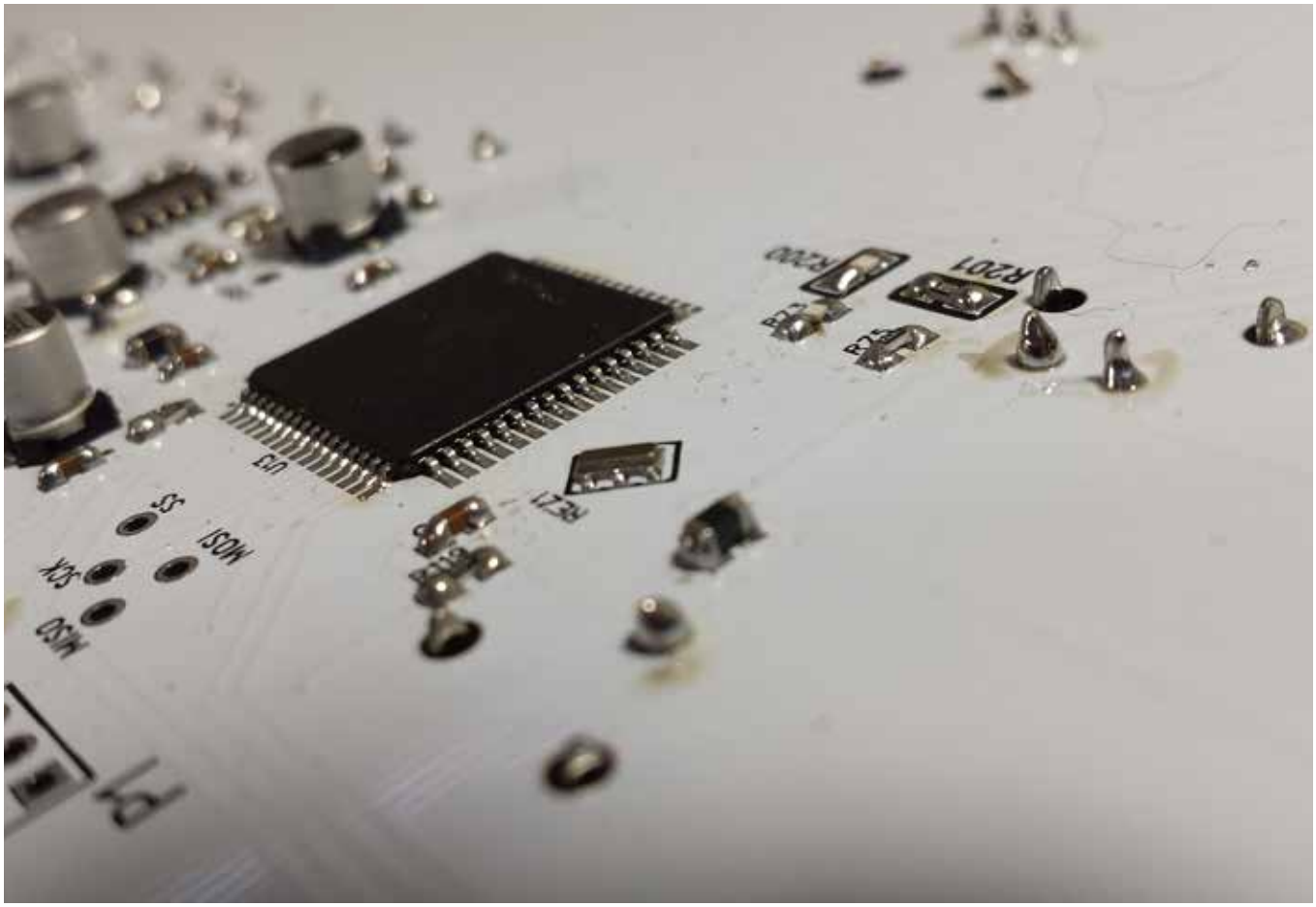
Read this section if you have a bare PCB DIY kit. (Bare PCB + parts)

For the easiest assembly, build your PCB in the order laid out in this guide. Since some parts are close together, such as in the power suppy, assembly will be easiest if you assemble in a particular order. It is recommended that you clean your work of flux as you go, as this will guarantee that no tiny pools of flux are overlooked. Flux can cause erroneous readings in the sensitive multiplexer circuitry, so it is critical that your work be clean.

The first component to solder to the PCB is the Atmega128 processor. While this component looks daunting, applying it to the PCB can be quick and easy if the proper soldering tip is used. For more information about proper tips, and videos of these tips in action, please look at this page at the Hakko website.

The ATMEGA128 PCB footprint.

The Atmega128 has 2, sometimes 3 dots imprinted on the top of the casing. The smallest dot indicates pin 1, and should align with the white dot on the PCB as shown above. Take care to inspect your work and be sure that there are not pins shorted on U13.
Once the main processor has been soldered and the flux has been cleaned, move on to soldering the ceramic resonator. Though the resonator is small, it is not difficult to solder. Apply flux to the 3 solder pads on the PCB, and using tweezers, place the resonator. Orientation does not matter. With a bead of solder on the tip of your iron, run the bead down the left and right sides of the resonator. You will see the solder adhere to the pad and the 3 gold contacts on the resonator.



Resonator soldered in place.

- STEP 2 - ICs

Next, solder the CD4067s, the CD4051, the opamps, the 74LS04, and the opto-isolator ICs. Take care not to leave flux around the pins of the ICs, especially on the CD4067s, as this will cause erroneous readings. The most common mistake when building these is to leave a mux pin unsoldered, or to have a cold joint on a mux pin. When this happens, it will cause multiple parameters to flash across the display, or the parameter associated with the unsoldered pin will flicker on the screen as you toggle other parameters. For best results, check your work using a microscope or other magnification.
The opto isolator is SMD, but the legs are bulky and they will absorb heat. This means that you should not drag solder this IC, and you should use more solder, so that you ensure a good connection. Any bad solder joint on this IC will result in MIDI IN not working.

- STEP 3 - DIODES, RESISTORS, INDUCTORS, FUSE, & CAPACITORS

Next, we move on to soldering the resistors and capacitors. Use the BOM to locate and solder all resistors and capacitors.

Diodes are marked with a bracket which indicates the direction of the cathode (black bar). A bracket such as] indicates that the bar should point to the right. A [means that the bar should point to the left.

D7 is a schottky diode and the cathode marking is tough to read. Using a magnifier, look at the printing on the diode. The side with the "1" printed on it indicates the bar. Double check your orientation when placing this part, as a mistake will mean that your power supply will fail.

The fuse is a 1206 package and is all white. Place this above the power supply.

Place the 2 small inductors (0805 package, dark grey) L1 and L3.

Place all 0603 capacitors according to the BOM. Do not place any electrolytics yet.

When you have reached this step, clean all of your work of flux.

Now place the 22uF and 10uF electrolytic capacitors. Do not place the large 680uF and the large inductor. These parts get placed last.

- STEP 4 - SWITCHING REGULATOR

This step is very important. The regulator is the key to getting stable readings and stable operation. Start by using a light abrasive and lightly clean the legs and fin of the regulator. You want to assure that the solder joints in this part are very solid. Once the metal parts that will be soldered to are clean, apply flux to the pads and set the part on the PCB. Do not drag solder this part. To hold the part in place, solder pin 1. Do the same for all 5 legs, taking particular care on pin 3, the center leg. This pin will require that you linger with the soldering tip for a bit longer, since this pin is directly connected to the heat sink fin, and the joint will absorb heat, thus taking longer for the solder to form a proper joint. To finish, apply a genereous amount of solder to the fin, making sure that solder is flowing beneath the part and forming a good ground joint. This is very important.

- STEP 5 - LARGE INDUCTOR & FILTER CAP

Finally, solder the large inductor and the 680uF filter cap in the PSU. Note that you must use a fair amount of solder to attach the large inductor to the pads. Be generous with the solder here. When happy with the inductor mounting, solder in the 680uF filter capacitor. Clean your work, and this will conclude the SMD soldering portion of the assembly.

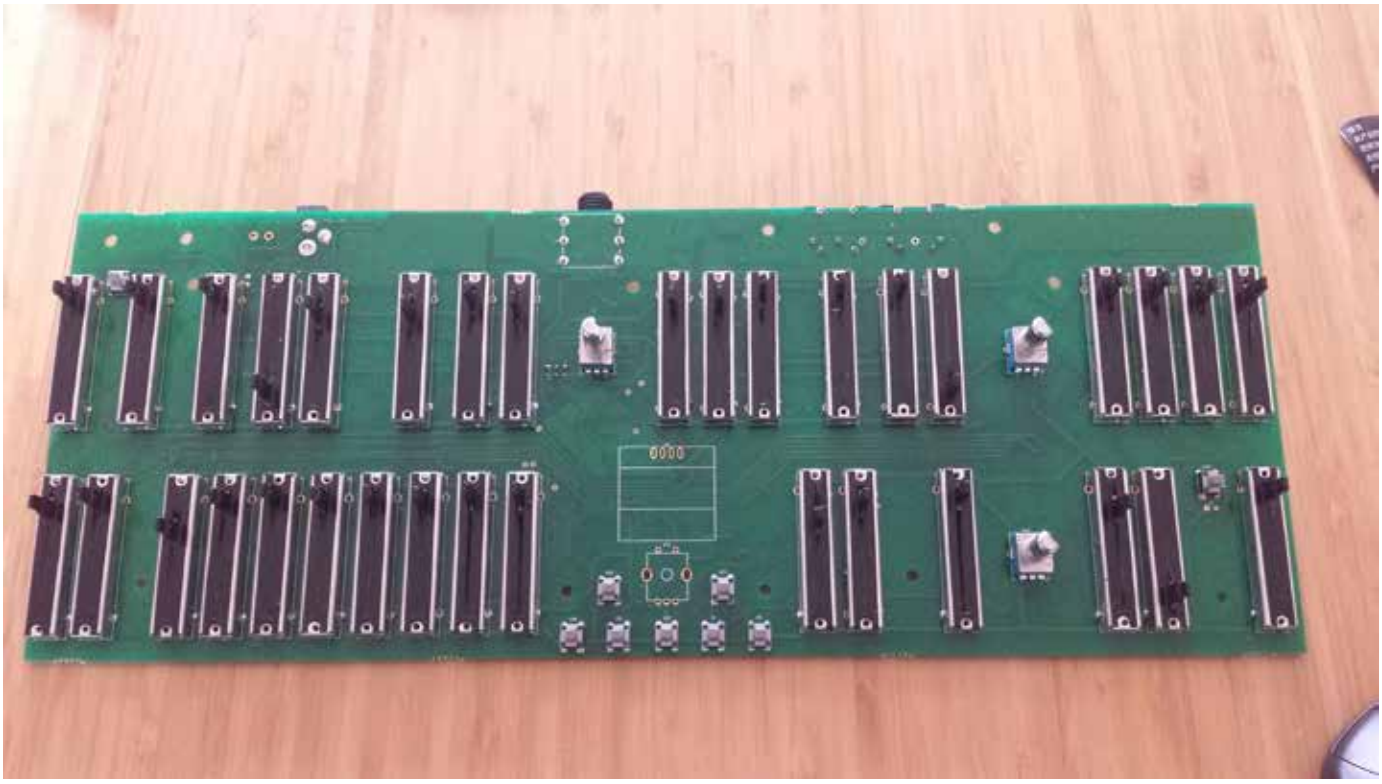
- STEP 6 - DIN JACKS, PEDAL JACK, & ISP HEADER

Next solder in the pedal jack, the 5P DIN jacks, and the 6P DIN jacks. These all go on the bottom side of the PCB (The side with the SMD components and the processor). Notice that the pedal jack will snap into place because of the kinked legs.
The ISP header should also be soldered to the bottom side of the PCB. It is important to solder these parts now, because once the PCB is attached to the panel, accessing the top side of the board will be impossible.

- STEP 7 - SLIDERS AND ROTARY POTS

In this step, we must place all of the sliders and all rotary pots EXCEPT THE SCREEN, LED, AND ENCODER.
When all pots and sliders are placed, use your soldering iron from the top side of the PCB and solder one of the stabilizing legs (The pins that are part of the slider chassis, not the 3 electronic pins) of each slider. This is done to hold the sliders in place when we flip the PCB to solder the pins of the pots and sliders to the board. There's no need to do this on the rotary pots. Those will stay in place when you flip the board without being soldered.

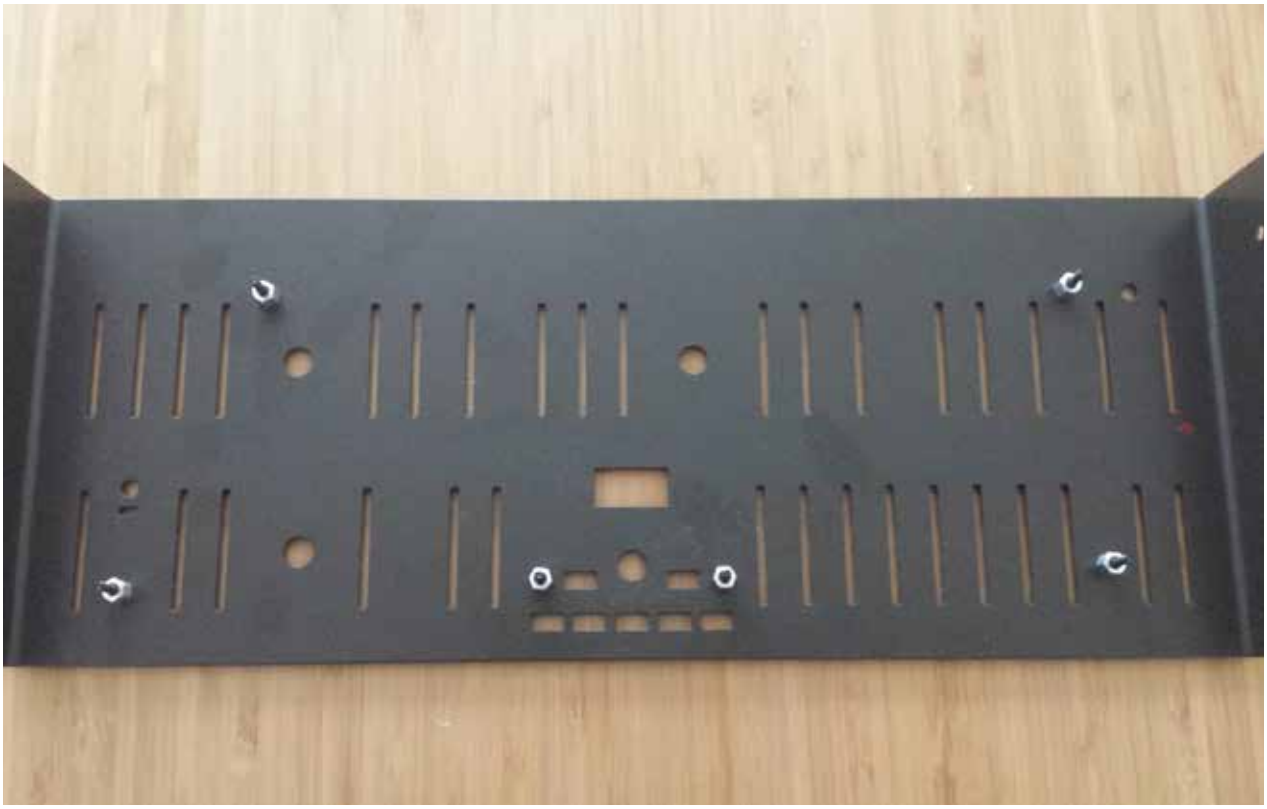
Now flip the board. Make sure that the sliders and switches are straight when you solder them in. Do the same for the rotary pots. When this step is finished, your board will have all parts populated except the LED, screen, and the encoder, as shown on the next page.



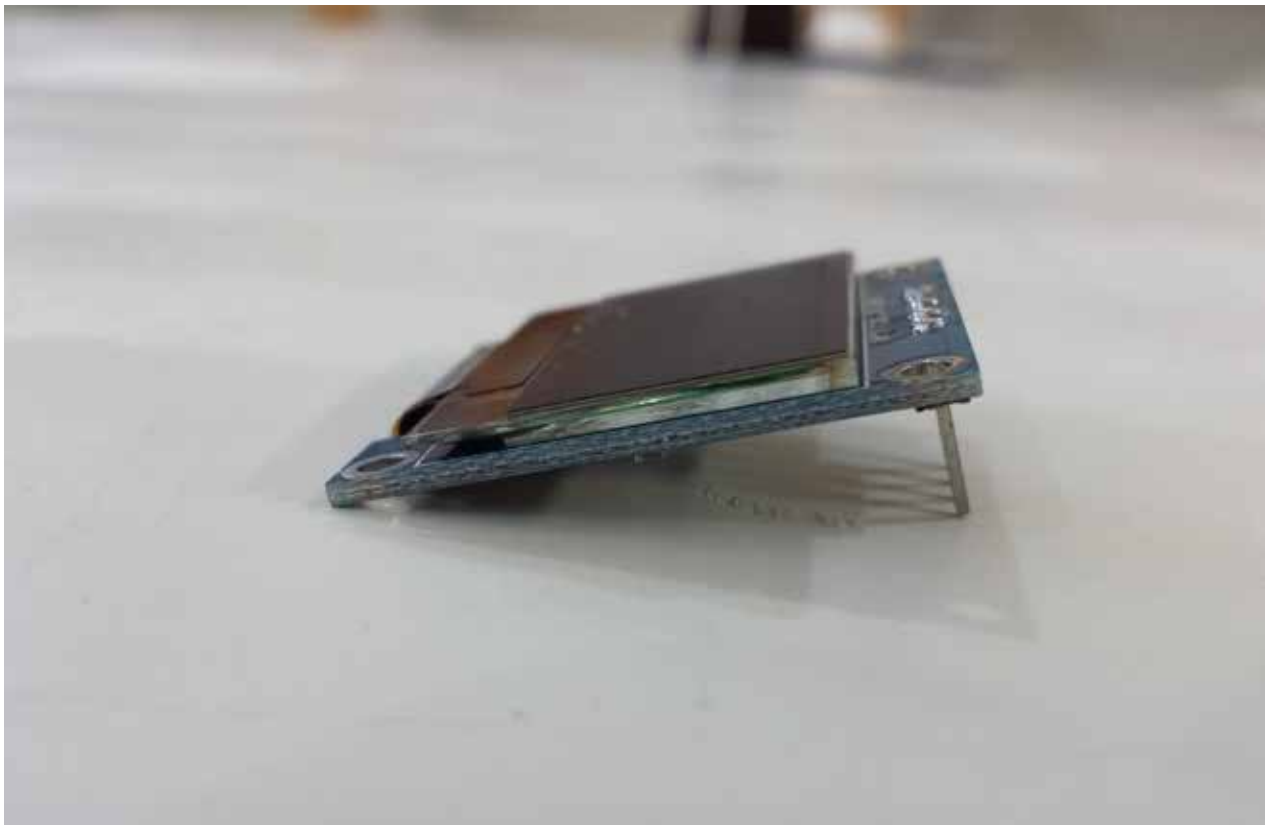
All pots and sliders installed.

- STEP 8 - ATTACHING PANEL

Next we must prepare the front panel for attachment to the PCB. Begin by gathering the long screws, the threaded standoffs, and the nylon nuts. screw the standoffs onto the screws as shown on the following page.



Now it is time to prepare the screen to be placed. The screen has 4 pins that must be trimmed to be flush with the PCB. This is so the pins do not make contact with the metal front panel. The pins should be cut using angle cutters and should be flush as shown below.



Now place the screen, the encoder, and the LED. (The short lead of the LED goes into the square hole) Do not solder yet. Just fit them in their respective holes. Attach the switch caps to their respective switches.

Use the double sided adehesive to stick the screen to the front panel. To see the visible area of the screen, look at the screen under a light from a side angle. You will see that the visible area has a blue tint. Align the screen and stick it to the front panel.

Now lower the front panel onto the PCB, taking care to gently help and button caps or OLED pins into their holes. Make sure that the screen isn't getting misalinged. When the panel alls into place, use the nylon nuts to hold the PCB to the protruding 5 screws under the board. Solder one pin of the OLED to hold it in place, then using the light and side angle view, make sure that the screen is where you want it. When satisfied with the screen positioning, solder the screen in place.

Now adjust the LED so that it's poking through the rectangular hole and flush with the panel. Solder in place when you are satisfied with the fit.

The rotary encoder is the last thing to solder into place. It is recommended that you screw the encoder into the front panel firmly THEN solder the 5 encoder pins and the 2 stabilizing pins. This will give you the best feeling encoder.

Test all of the buttons, and when happy with the fit, tighten all screws.

To complete the build, the operating system and EEPROM data files must be loaded into the CPU. Once this is done, all future updates can be sone using MIDI SYSEX.

Use the Zip file containing the MPG software files. For instructions, read the document called Flashing MPG50 included in the zip file. Read the README file as well.
Once this step is done and the unit is tested and working, the enclosure may be assembled and the build is complete.

- TROUBLESHOOTING

If you are experiencing technical issues with your build, this section can help to identify common failure modes:

Flurry of parameters flashing on the screen: Indicates an open circuit on either one of the pot/slider legs or a bad joint on one of the 4067 multiplexers. A pool of flux on a 4067 pin can also cause this. If the power supply is not stable, then you will also see this behavior.

Buttons don't work: check the soldering of the CD4051. Check that no diodes are backwards. Check address pins and enable pins of 4051 and trace them back to the processor. Look for bridges and pools of flux. If there is a diode that was in backwards, chances are it destroyed your 4051. Replace that and the diode and try again.

Unit freezes but screen still on: Check the resonator soldering.

One parameter always flickers on the screen, even when toggling other functions: Check the mux pin associated with that parameter. The MPG is telling you that there's a bad connection on that line. It could be the mux input pin, or it could be the pot.

If you have any other questions, issues, etc, please email them to retroaktivmodular@gmail.com. Thanks for supporting these products!