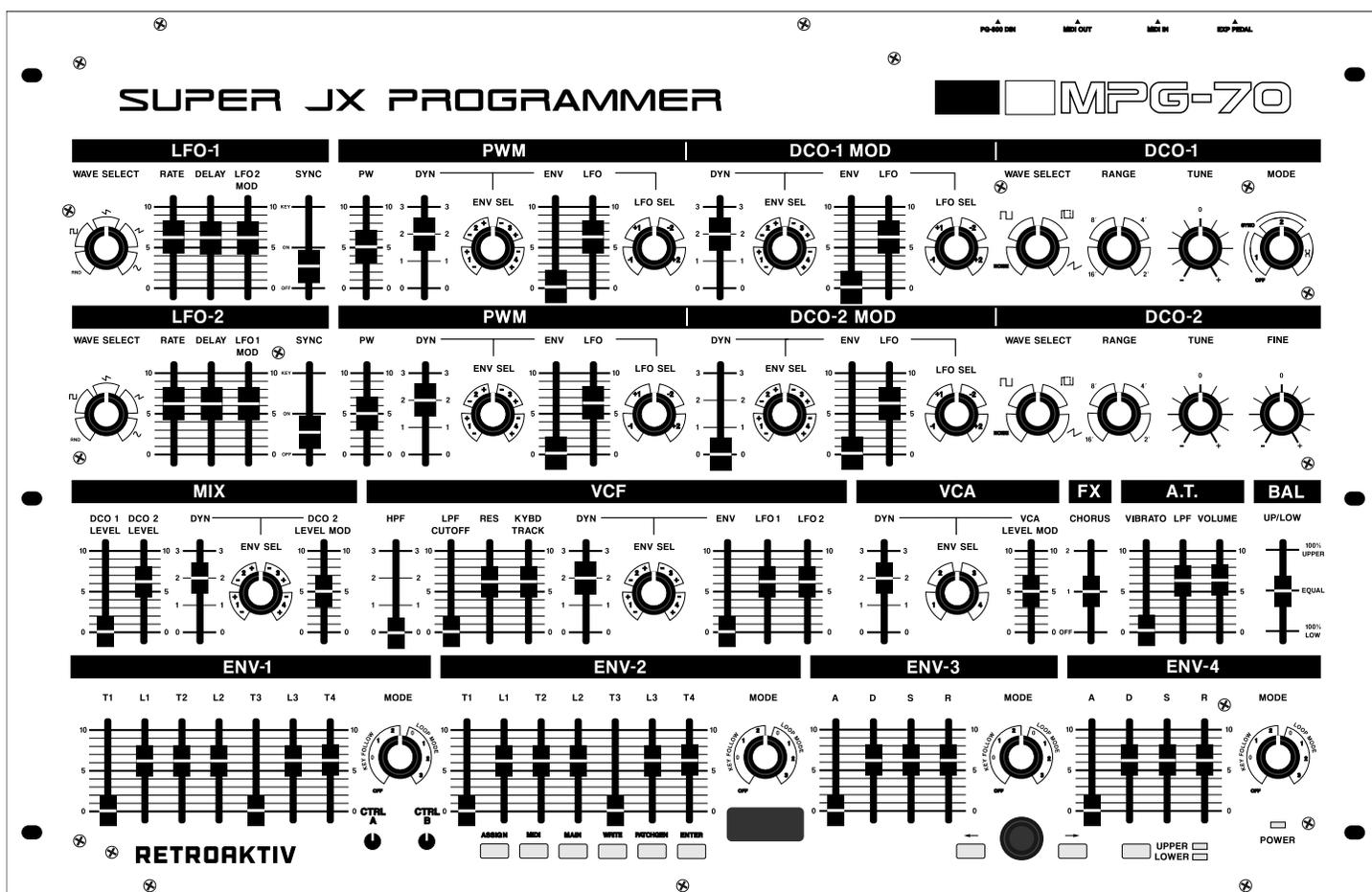


# MPG-70 DIY KIT BUILD GUIDE



VERSION 1.0

This guide is intended to help guide builders through the process of assembling, programming, and testing the Retroaktiv MPG-70 Super JX 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 (USBTiny, for example)
- Philips head screwdriver
- SMD tweezers
- 

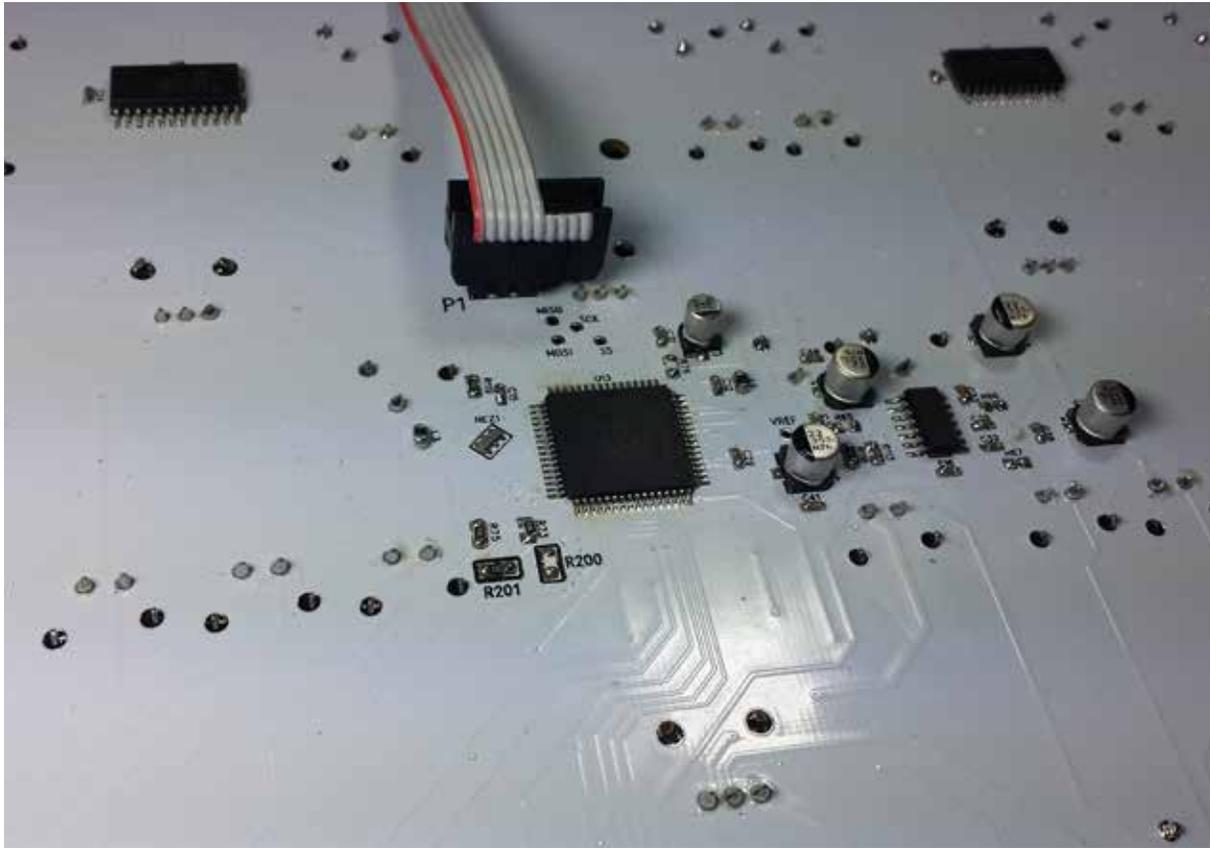
When you receive your kit, the enclosure, rear plate, and front panel will be screwed together, and the PCB, and all other parts will be inside of the enclosure.

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:

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 supply, 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.

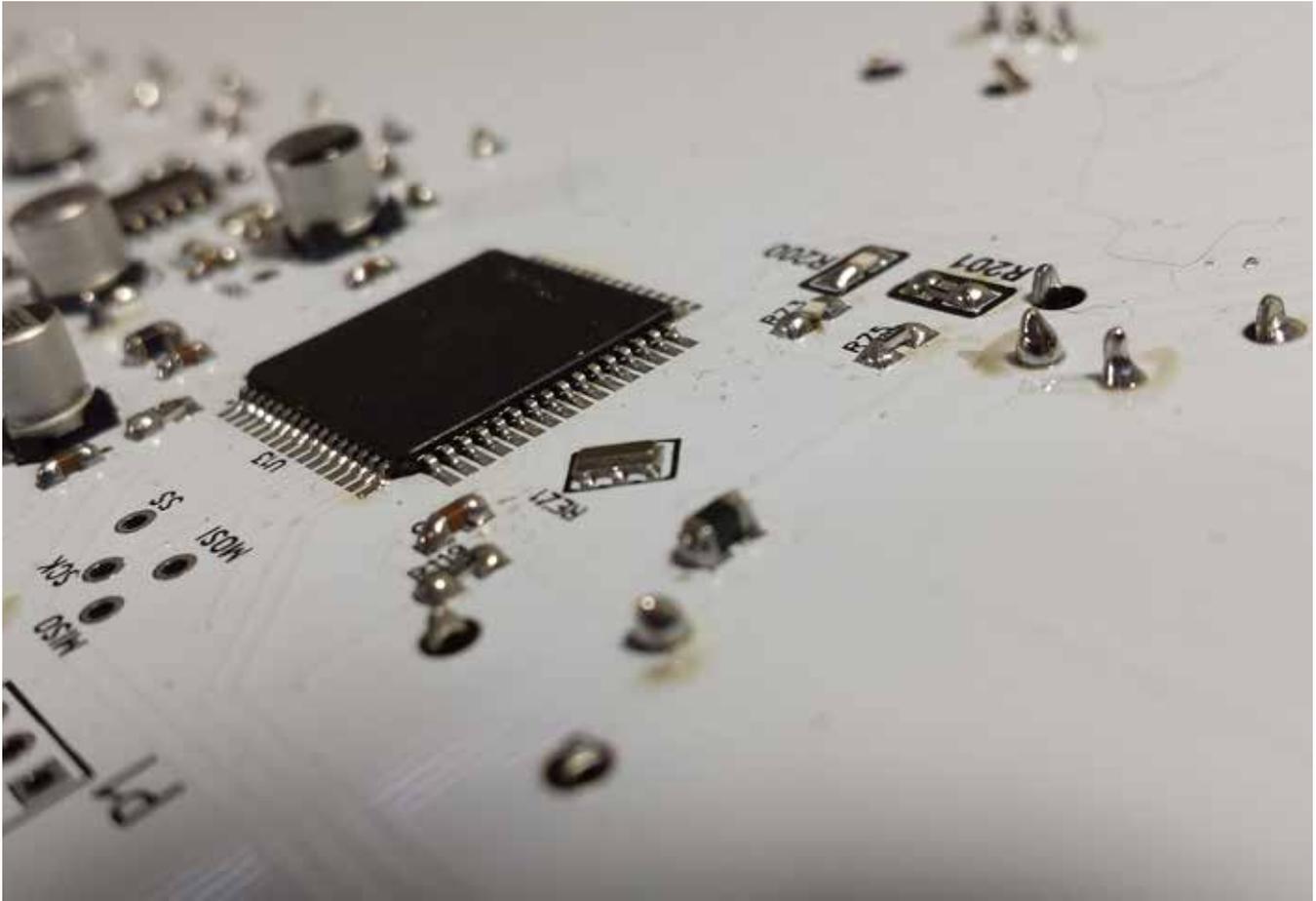
The first component to solder to the PCB is the Atmega128 processor. While this component looks daunting, applying it to the PCB can be quick and easy if the proper soldering tip is used. For more information about proper tips, and videos of these tips in action, please look at this page at the Hakko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



The ATMEGA128 PCB footprint.

The Atmega128 has 3 dots imprinted on the top of the casing. These dots form a triangle that points to pin 1, which is indicated by the center dot. The center dot on the processor should match with the pin 1 indicator dot as shown above. (Dot to the upper right of U13) 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 6 CD4067s, the CD4051, the two OPA opamps, the 74LS04, and the op-to-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. Note that R100 and R112 (the resistors that set the regulator output voltage) are 0805 resistors. Be sure to use the 0805 resistors here.

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 generous 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.

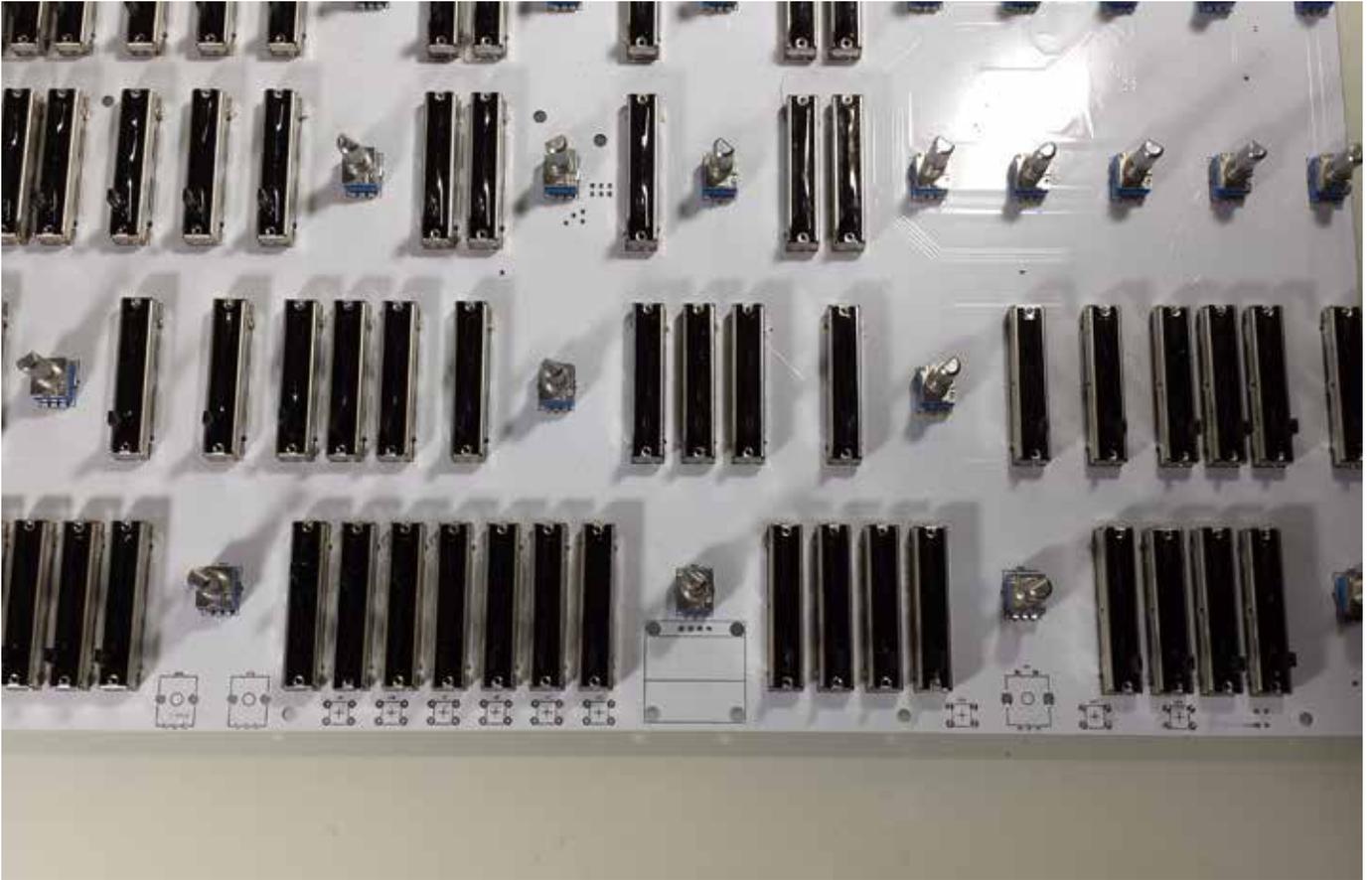
The ISP header should also be soldered to the bottom side of the PCB. It is important to solder these parts now, because once the PCB is attached to the panel, accessing the top side of the board will be impossible.

- STEP 7 - SLIDERS AND ROTARY POTS

In this step, we must place all of the sliders and all rotary pots EXCEPT THE 2 TALL TRIMMERS AND ENCODER. Note that we are NOT placing the 9 switches, the two tall trimmers, the encoder, and the 3 red LEDs yet. This comes later.

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.

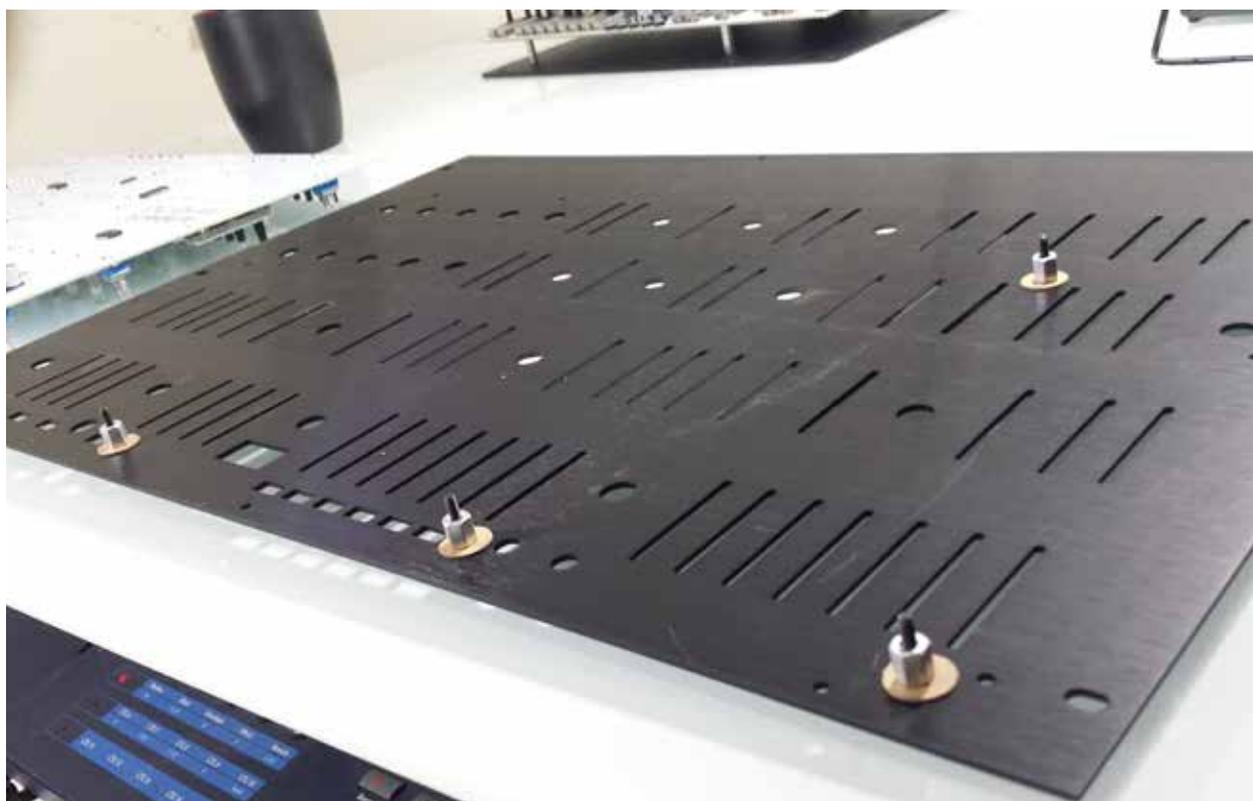
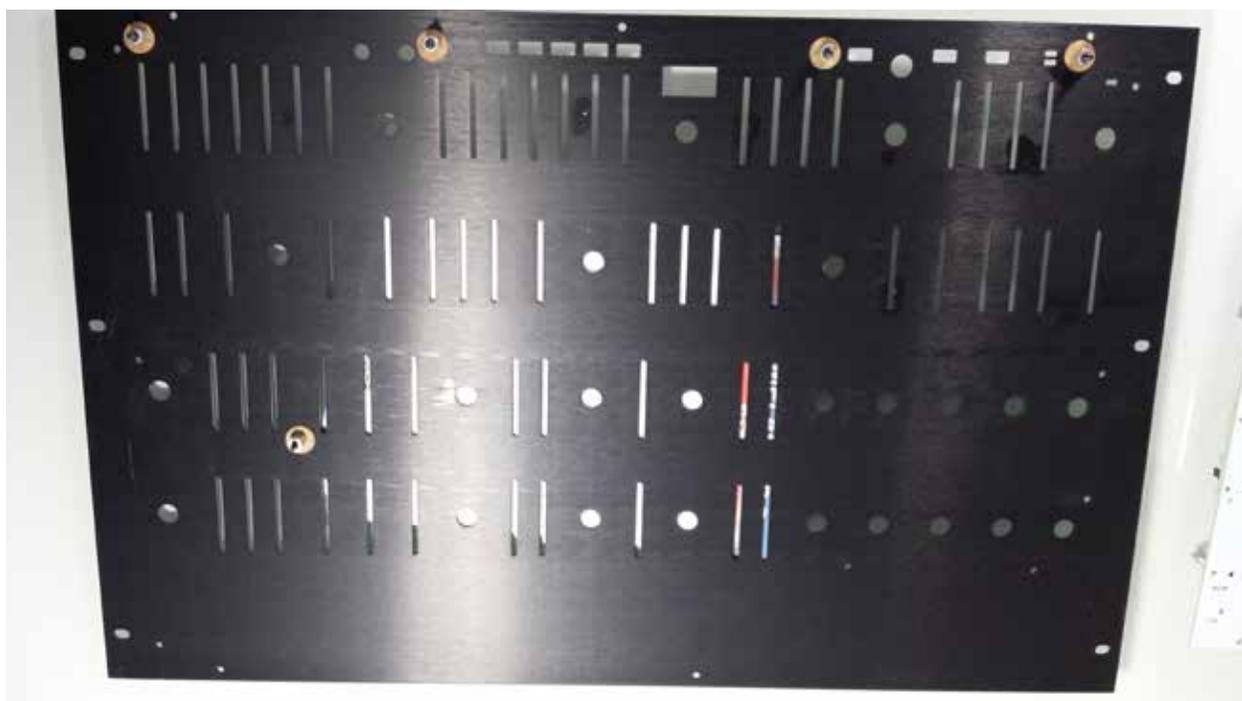
Now flip the board. Make sure that the sliders are straight when you solder them in. Do the same for the rotary pots. When this step is finished, your board will have all parts populated except the switches, LEDs, tall trimmers, screen, and the encoder.



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 5 long screws, the 5 threaded standoffs, and the paper spacers. Depending on the kit, you have either blue paper spacers or brown cardboard spacers. If you have blue spacers, use 4 spacers under each standoff and thread through the panel as shown below.



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.



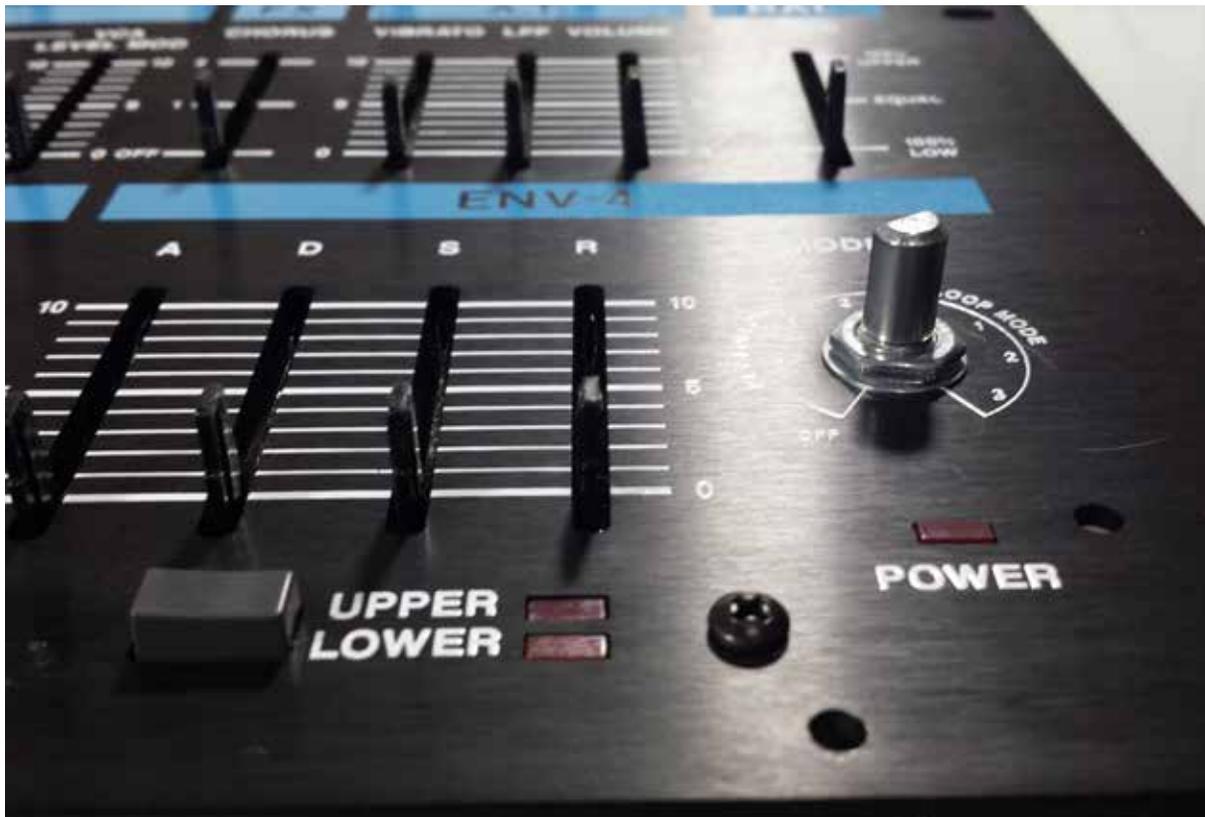
Now place the screen, the encoder, the tall trimmers, the switches, and the 3 LEDs. Do not solder yet. Just fit them in their respective holes.

Once all parts are placed, place the switch caps on the switches. Be sure that the switches are placed flush with the board and NOT AT AN ANGLE. Soldering them in at an angle will mean that your PCB will not fit through the panel, or that your switches will stick when pressed.



Now lower the front panel onto the PCB. Take care to make sure that the switches and pots that aren't yet soldered do not get pulled from their holes. When the panel falls into place, use the nylon nuts to hold the PCB to the protruding 5 screws under the board.

Now adjust the 3 LEDs so they're poking through the rectangular holes. Don't solder yet. Be sure that the PCB is sitting firmly on the standoffs and is not catching on anything. Now go down the row of switches and test each one, ensuring that the fit is good and the switch clicks up and down freely. Solder in place when you are happy with each switch. Take care not to let the screen fall out when you flip the assembly. Then adjust the LEDs so they are flush with the face of the panel as shown.



Solder in place and clip the leads. Note that LEDs should be placed with the long leg in the round hole and the short leg in the square hole. It is recommended that you solder one leg of each LED first, then make your height adjustment, and solder the remaining pin when the LED is placed correctly.

To solder the screen, place the panel face down and allow the screen to sit flush with the panel. The glass of the screen should be flush with the rear side of the front panel.



Finally, test the movement of the 2 tall trimmers. They should be easy to move. If they are not, adjust their positioning so they are not chafing the panel and they move freely. Solder into place.



Now be sure that the encoder is sitting flush and solder it into place.

This concludes the soldering portion of the assembly. Now apply washers and nuts to all 25 rotary pots. Tighten all nuts and then apply knobs to all pot shafts, and caps to all sliders. The panel and PCB assembly is now finished. Now it is time to power up, program the processor and test.

- STEP 9 - DIN CABLE

If you have a wall wart adapter, you may connect the board to power using this. Otherwise, use the included 6P DIN cable. The kits comes with 2 DIN plugs and a length of 5 conductor cable. Begin by disassembling the DIN plugs. There are 4 pieces in each plug assembly: A rubber sleeve, the male plug, and two halves of the clamshell plug assembly. Look at the rear of the male plug, and you will see 6 hollow, concave pins, each with an imprinted number, 1-6 next to it. Pin 6 is the center pin. Pin 1 is the top-right pin. Moving clockwise, the pin numbering goes from 1 to 5. Take not of these pin numbers.

Next, strip 1" of the grey insulation off of the length of multi-conductor wire to expose the 5 wires inside. This is a good time to thread the rubber sleeves of the DIN plug onto the grey cable. **DO NOT FORGET TO DO THIS BEFORE YOU SOLDER THE WIRES.** If you do,

then you will need to rebuild the cable.

Strip 2mm of insulation off of each end of the 5 colored wires. Tin the ends to the stranded wire will not fray. Use a drop of flux on pins 1, 3, 4, 5, and 6 (omit pin 2) of the male plug and load each pin with solder. With the plug held in place with a clamp, etc place the pre-tinned wires into the pre-tinned holes while heating the posts with a soldering iron. Each wire should slide in and the solder will bond. Start with center pin (pin 6) and then move clockwise. Use the chart below:

Color scheme is as follows:

Pin 1 = Red wire

Pin 2 = NO CONNECTION

Pin 3 = Green wire

Pin 4 = Brown wire

Pin 5 = White wire

Pin 6 = Black wire

- STEP 10 - POWER UP

Using your wall adapter or DIN cable, it is time to apply power to the board. Note that there are test points for GROUND and VCC above and below the 680uF power filter cap. Turn the unit on and you should see the power indicator light up. Use a scope or meter to see the voltage between ground and VCC. It should be about 4.7VDC. Observe the power for a bit and make certain that it is stable. A noisy or unstable supply can cause the programming of the processor to have issues, or worse, make it impossible for you to FLASH the program.

- STEP 11 - PROGRAM CPU

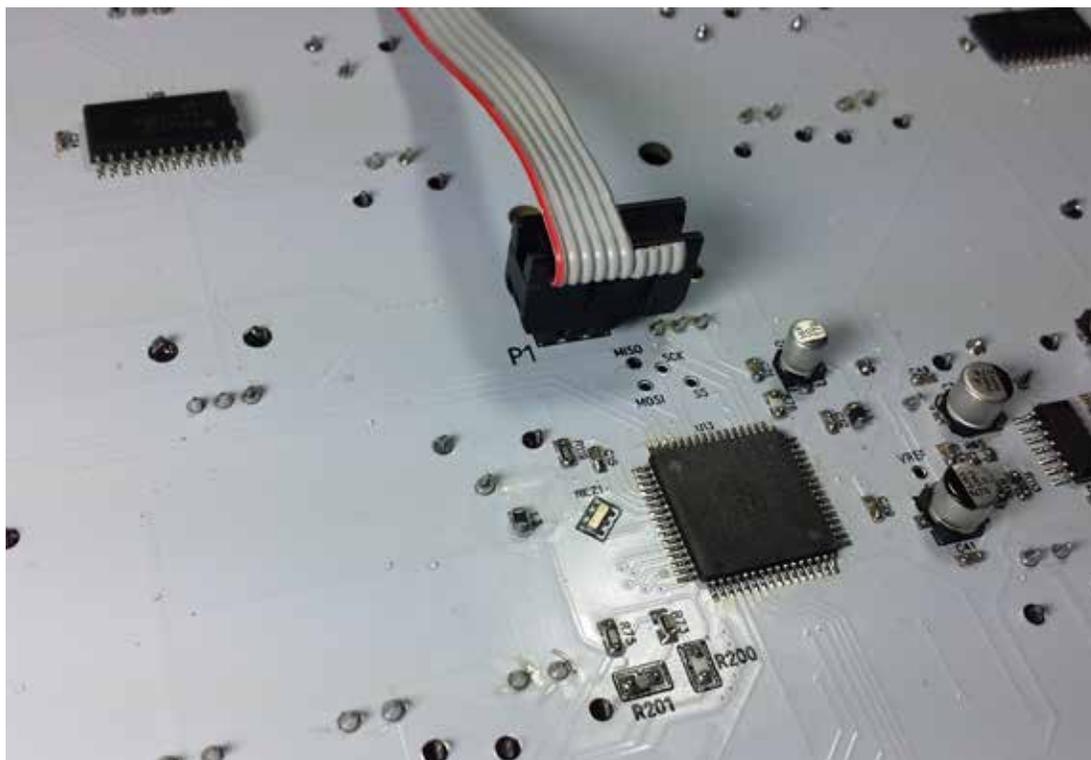
The CPU must be programmed using an ISP programmer such as USBTiny. The USB tiny is a common ISP programmer that can be purchased for less than \$10. This is the programmer we will be using in all of the programming examples.

The easiest program to flash the processor with is AVRDude. You can use any software that you are comfortable with, but in these examples, we use AVRDude. Download AVRDude and install.

The default folder that AVRDude looks in when it flashes a file is C:/USERS/MAIN USER, in my case, since my name is Rob, I would look in C:/USERS/Rob.

People who purchased kits are emailed 2 files: an MPG70 EEPROM data file, and an MPG70 program file. Put these 2 .BIN files in the aforementioned folder.

Next, connect your ISP connector to the MPG70 board, oriented so that the red strip (pin 1) lines up with the PIN1 indicator dot on the MPG70 board as shown below. When connected, apply power.



Open command prompt on your PC. You are now ready to begin programming the processor. We must first set the processor fuses to the correct configuration needed to run the MPG70 hardware. **DO NOT CONFIGURE YOUR ISP DEVICE TO POWER THE MPG70 BOARD!! TURN THIS OPTION OFF OR YOU RISK DAMAGING STUFF.**

In the command prompt, use the following line of code, and press ENTER.

```
avrdude.exe -p m128 -c usbtiny -U lfuse:w:0x0:m -U hfuse:w:0xd9:m -U efuse:w:0xff:m
```

This line of code sets the Atmega128 clock source to use the external resonator, and it sets up the chip to the configuration used in the MPG70.

Next, we need to tell the chip to allow us to program the EEPROM. The EEPROM contains tables of characters and data that the main program uses to create sysex strings and character strings for the slider/pot parameters.

```
avrdude.exe -p m128 -c usbtiny -U hfuse:w:0x99:m
```

Next, we erase the content of EEPROM.

```
avrdude.exe -p m128 -c usbtiny -e
```

Now we must load the new EPROM data. Replace the MPG70EPROM1V2.bin in the next line of code with the exact (case sensitive) name of the EPROM file you received. Hit enter to load EPROM. Wait at least 10 seconds for the program to run before moving to the next step.

```
avrdude.exe -p m128 -c usbtiny -U flash:w:MPG70EPROM1V2.bin
```

Now we must disable EPROM erase. This protects the contents of EPROM when we load the main program.

```
avrdude.exe -p m128 -c usbtiny -U hfuse:w:0x91:m
```

Finally, load the main program file. As with the EPROM file, you must replace the file name with the exact name of the firmware file you received.

```
avrdude.exe -p m128 -c usbtiny -U flash:w:MPG70R1V37.bin
```

Do not interrupt the ISP programming when the program is reading writing. Doing so may disable the Atmega128's ability to be flashed, which will require a special tool to resurrect. Avoid doing this at all costs.

NOTE: As of the time of this writing, the OS revision is 1.37. The bin filenames in the commands above must match the file names of the EEPROM and OS files you are trying to flash.

If you receive an error message when trying to program, be sure that your connections are correct. 99% of the time, these error messages are displayed because you have connected the programmer backwards, or the MPG70 is not powered up.

After programming the final bin file, you should see the MPG-70 splash screen and the version of the OS you're running. Now test all of the functions and you should be ready to start using your MPG70.

### • TROUBLESHOOTING

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

**Jibberish on screen:** Repeat the 5 step programming procedure. If you see odd characters being displayed, then chances are that your EEPROM file didn't program correctly. Try all 5 steps again.

**Screen dimly lit, no power LED:** Reflow all solder joints on voltage regulator, main filter cap, fuse, 22uF ceramics, the 5.6nF cap and 2 0805 ¼ watt resistors in the power supply. This behavior is almost always indicative of a cold joint in the power supply.

**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.

**LED's flickering:** Check your power supply connections, especially the 2 resistors and the pins on the regulator.

**Unit freezes:** Check for cold solder joints on the power supply. Excessive ripple can cause the main processor to freeze. This is almost always indicative of a bad solder joint on one of the PSU components.

**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](mailto:retroaktivmodular@gmail.com). Thanks for supporting these products!