LUNAR DELAY

v1.3 Build Guide

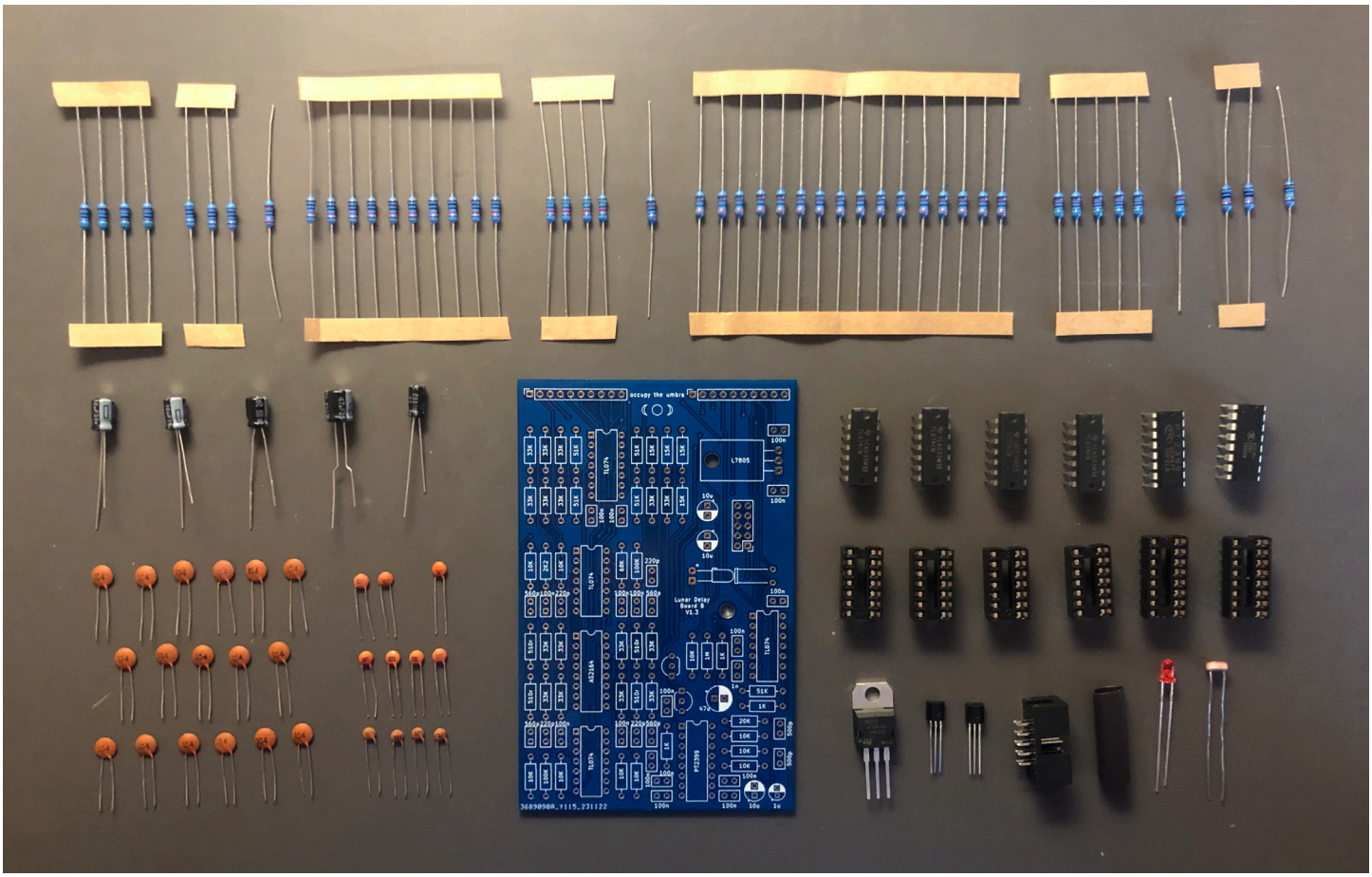
INTRODUCTION

Lunar Delay is a beginner/intermediate level build. You will need to know how to solder through hole components. There is nothing too tricky, but there are a lot of components to solder and you will need to make a vactrol. It's a good idea to read through this entire document before starting to build. The assembly should take 4 to 8 hours to complete. The following tools are required:

- Soldering Iron
- Solder
- Flush Cutters
- Needle Nose Pliers or Tweezers
- Helping Hands & Lighter (for making the vactrol)
- Multimeter

This module has two PCBs. The kit comes with all components sorted into bags according to which board they go to. Download the BOM for the complete and detailed list of parts.

BAG 1 / BOARD B



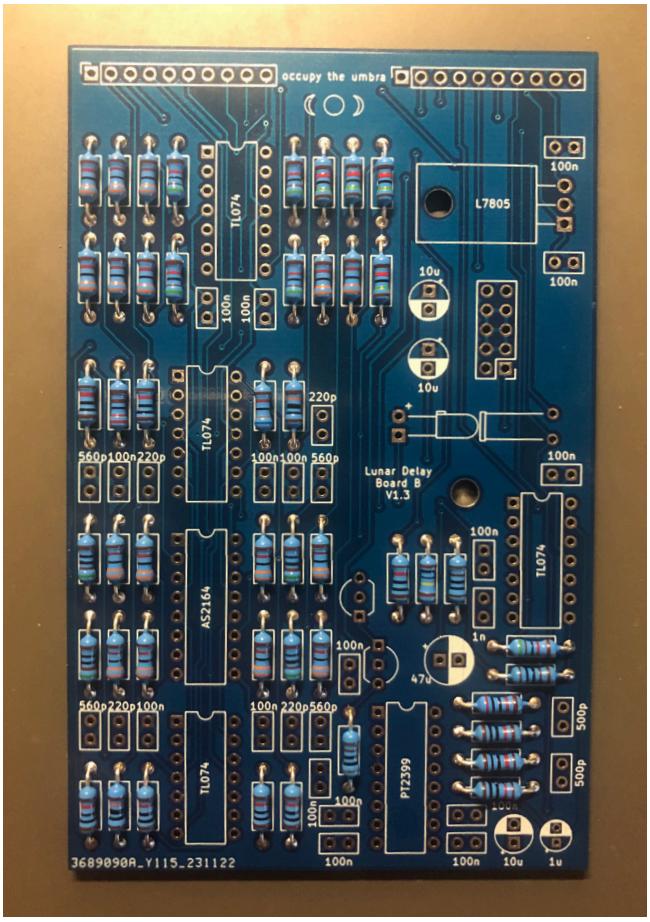
BAG 1 / BOARD B

Qty	Part	Value	Body Marking
4	Resistor	510r	Green Brown Black Black Brown
3	Resistor	1K	Brown Black Black Brown Brown
1	Resistor	2K2	Red Red Black Brown Brown
10	Resistor	10K	Brown Black Black Red Brown
4	Resistor	15K	Brown Green Black Red Brown
1	Resistor	20K	Red Black Black Red Brown
16	Resistor	33K	Orange Orange Black Red Brown
5	Resistor	51K	Green Brown Black Red Brown
1	Resistor	68K	Blue Grey Black Red Brown
2	Resistor	100K	Brown Black Black Orange Brown
1	Resistor	1M	Brown Black Black Yellow Brown
4	IC Socket	DIP 14	
2	IC Socket	DIP 16	
1	Capacitor	1n	102
4	Capacitor	220p	221
2	Capacitor	500p	501
4	Capacitor	560p	561
17	Capacitor	100n	104
2	Transistor	2N3904	
1	Voltage Regulator	L7805	
1	LED	3mm Red	
1	Light Dependant Resistor	10K on / 2M off	
1	Heat Shrink Tubing	6mm diameter, Black	
1	Shrouded Header	2x5	
1	Electrolytic Capacitor	1u	
3	Electrolytic Capacitor	10u	
1	Electrolytic Capacitor	47u	
4	IC	TL074	
1	IC	AS2164	
1	IC	PT2399	

Grab Board B, open Bag 1, and organize the components. It's a good idea to double check all resistor values with a multimeter.

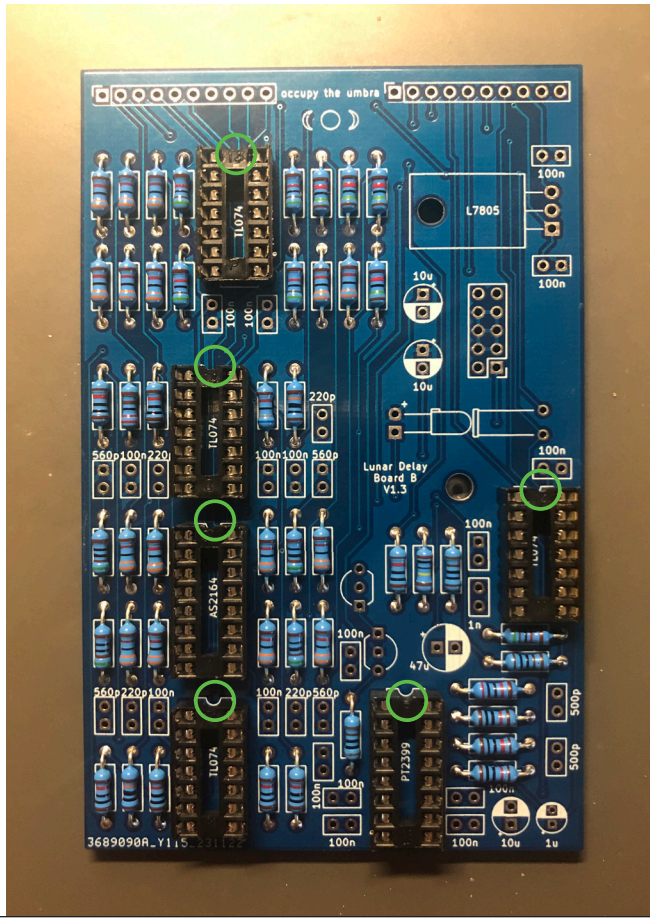
BAG 1 / BOARD B

Start by soldering the resistors. Resistors are not polarized, their orientation does not matter. There are 48 resistors in total so get comfortable, this step can take a while.



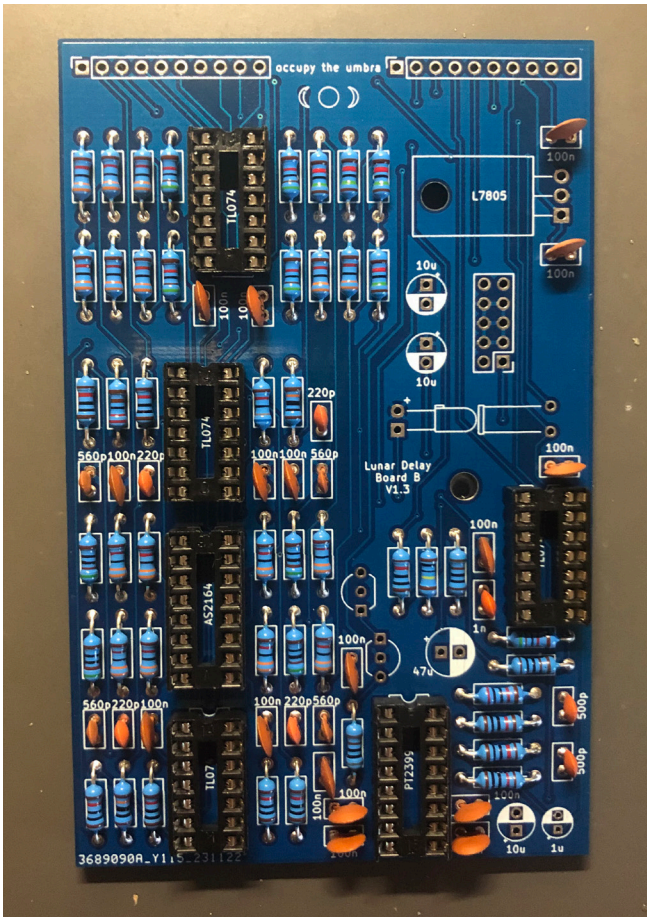
Now move on to the IC sockets. There are four 14 pin sockets and two 16 pin sockets. Each one must be placed in a specific orientation. Make sure the notched end of the socket matches the notch drawn on the PCB silkscreen. All sockets should be orientated with the notch toward the top of the PCB.

When soldering the sockets in place, first solder just one pin on each socket. Then flip the board over and check that each socket is sitting flush against the board. If it isn't flush, reflow the one pin you soldered while gently pressing the socket against the board. When each sockets is flat against the board, solder all the remaining pins.

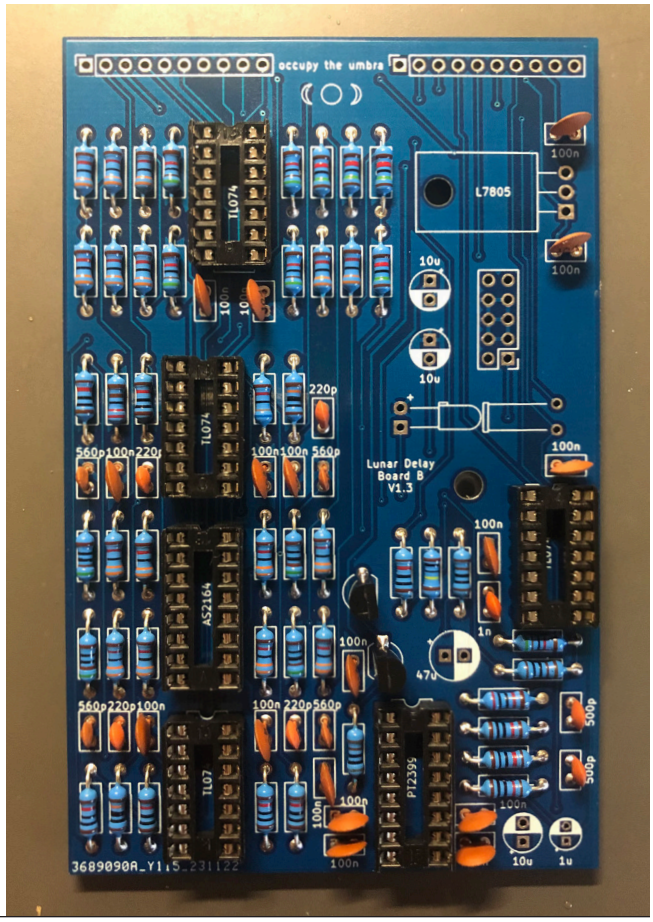


BAG 1 / BOARD B

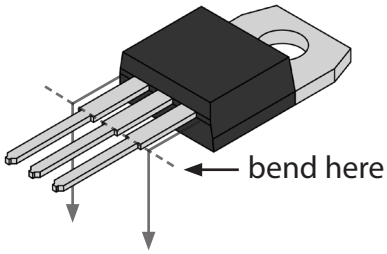
Next solder the ceramic capacitors. These are the small orange ones. They are not polarized, their orientation does not matter. There is one 1n cap, four 220p caps, two 500p caps, four 560p caps, and seventeen 100n caps.



Now do the two transistors. They are both 2N3904 NPN transistors. Make sure the orientation of each transistor matches the silkscreen drawing.



BAG 1 / BOARD B



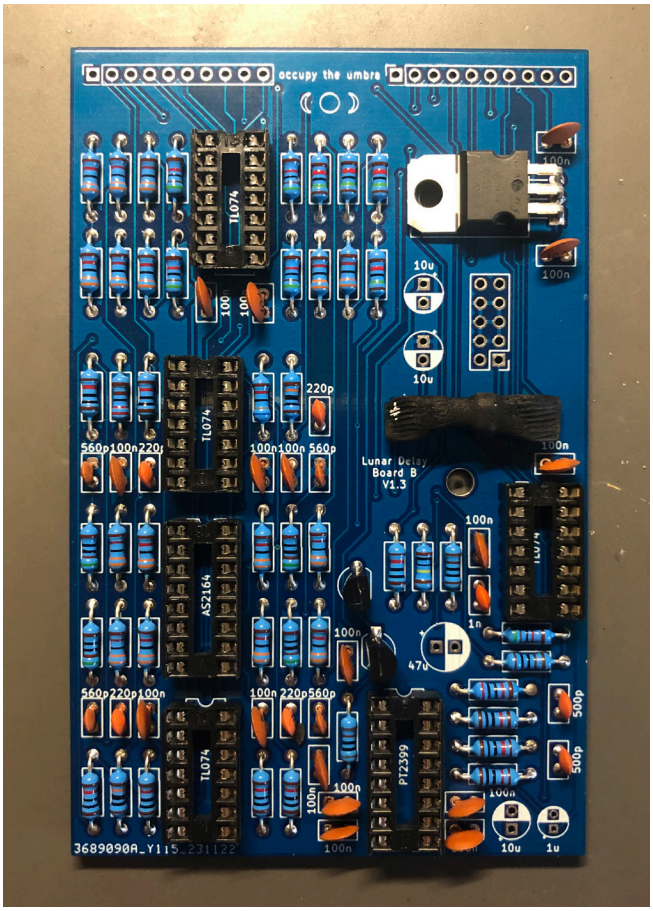
Next up is the L7805 voltage regulator. Before you place it on the board, prepare the component by bending its leads down at the point where they transition from thick to thin. A needle nose plier or pair of tweezers is handy for bending all three leads at once, which keeps them aligned with each other. Position the voltage regulator with its metal tab flat against the PCB and solder the three pins in place.

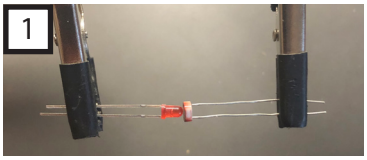
After the voltage regulator is done, we will make the vactrol. A vactrol consists of a light emitting diode and a light dependent resistor contained inside a light-proof enclosure. This kit comes with heat shrink tubing for the vactrol but you can also roll it up in electrical tape if you choose.

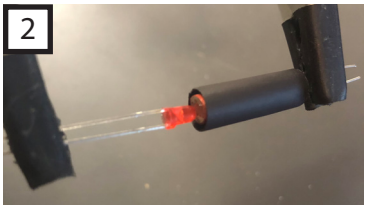
To make the vactrol with heat shrink tubing, it helps to have "helping hands," or something similar to hold everything in place. Line up the LED and LDR and then slide the heatshrink over them. Use a lighter on the heatshrink, and then clamp the ends shut to seal it up.

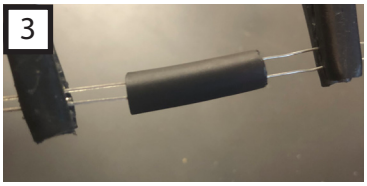
If you don't want to use heat shrink, you can simply roll the LED and LDR up in electrical tape. Just make sure then ends are closed, and no light can leak in or out.

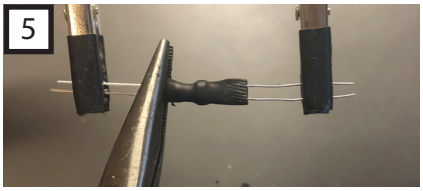
Once the vactrol is made, it's a good idea to mark the LED's anode (long lead) with a plus sign. Mount the vactrol on the PCB with its anode in the hole marked with a plus sign, and solder it in place.

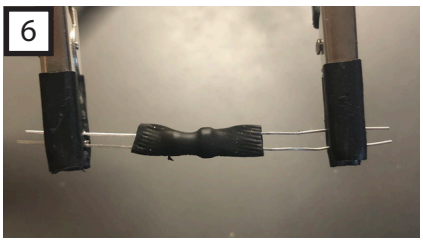


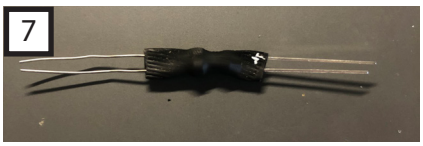






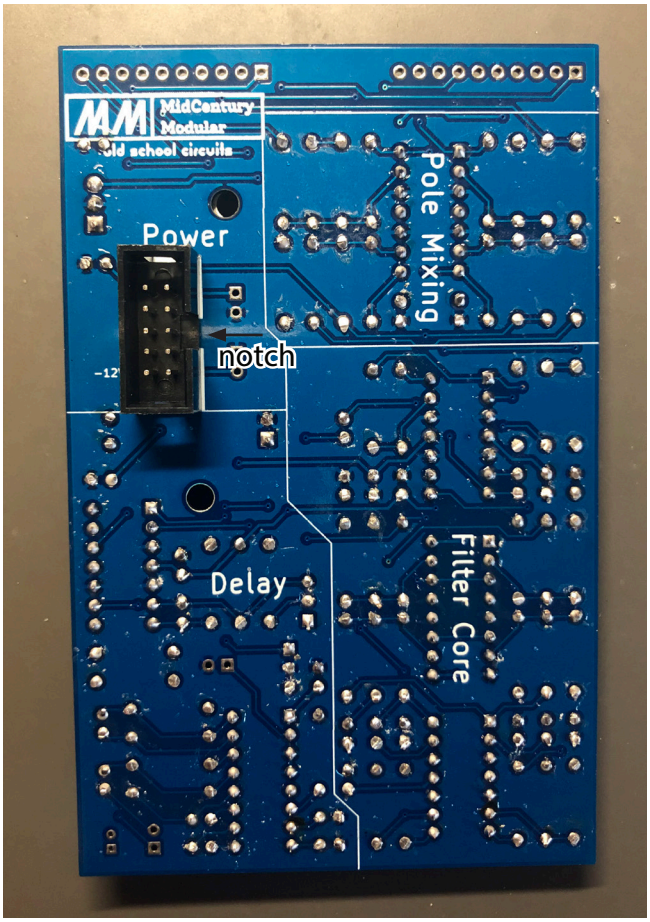






BAG 1 / BOARD B

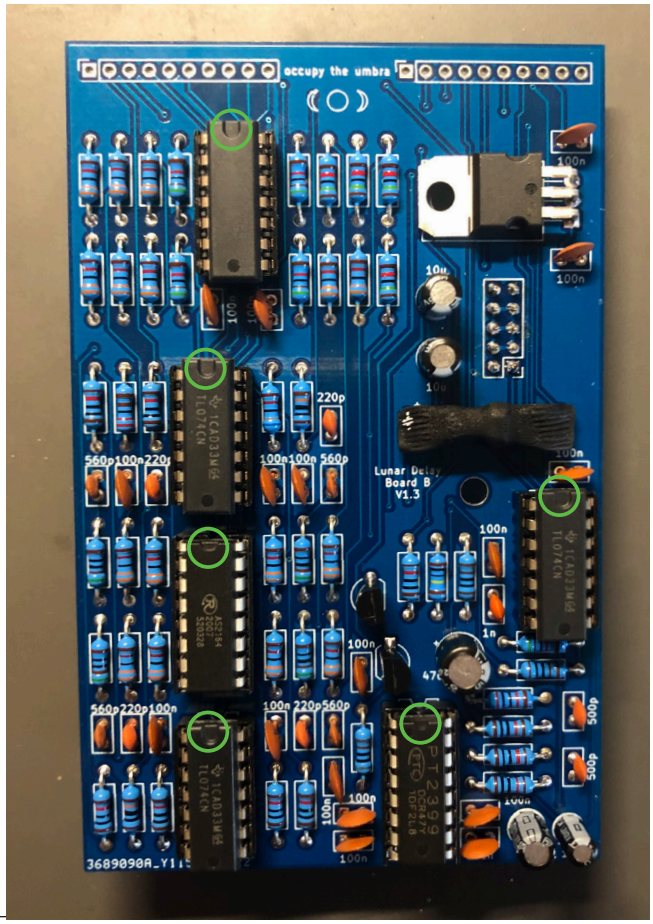
Place the 10 pin shrouded header on the back of the board. Before you solder any pins, make sure the orientation is correct. The header's notch should match the notch drawn on the silkscreen. If it is soldered in the wrong position, the module won't power on, and it will be a real pain to fix if all ten pins are soldered. Just like the IC sockets, solder one pin and then make sure the header is flat against the PCB. If it needs to be adjusted, reheat the one pin you soldered while gently pressing the shrouded header toward the board. When you are satisfied with its position, solder the other nine pins.



Finally, solder the electrolytic capacitors. These are polarized, make sure the long lead of each one goes into the hole marked with a plus sign. There is one 1u cap, three 10u caps, and one 47u caps.

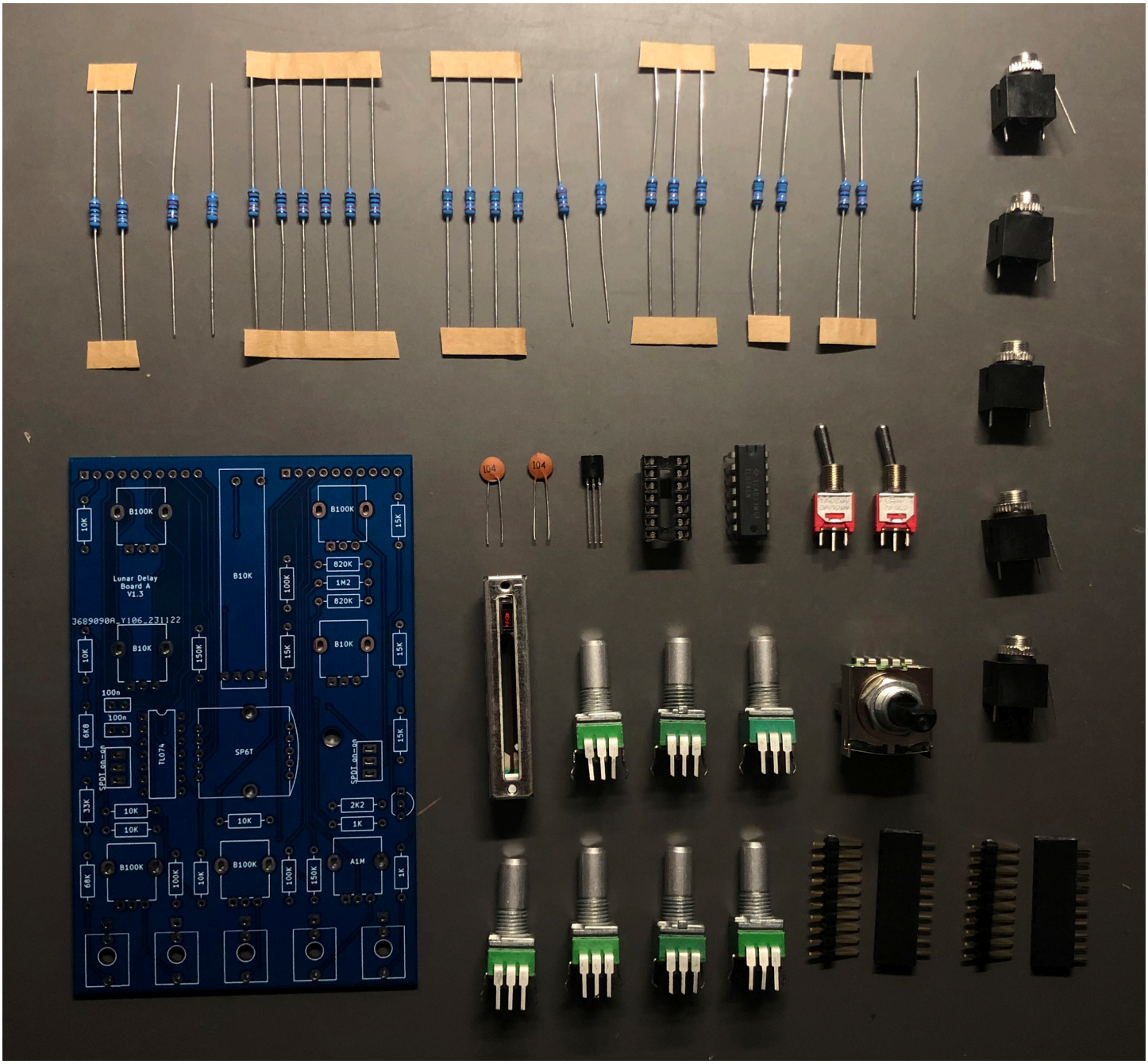
Now place the ICs in their sockets. You may need to bend pins slightly in order for them to fit. Make sure each one is orientated correctly, the notch on each IC matches the silkscreen drawing.

Once the ICs are in their sockets, you can set this board aside. This is a good opportunity to take a break, eat a snack, maybe call it a night. When you are ready, move on to Board A.



BAG 2 / BOARD A

Get Board A out, open Bag 2, and organize the components. Once again, it's a good idea to check the resistor values with your multimeter. Cut two 1x10 segments from both the male and female pin header strips. Keep in mind, when cutting female pin headers you will always lose one pin.



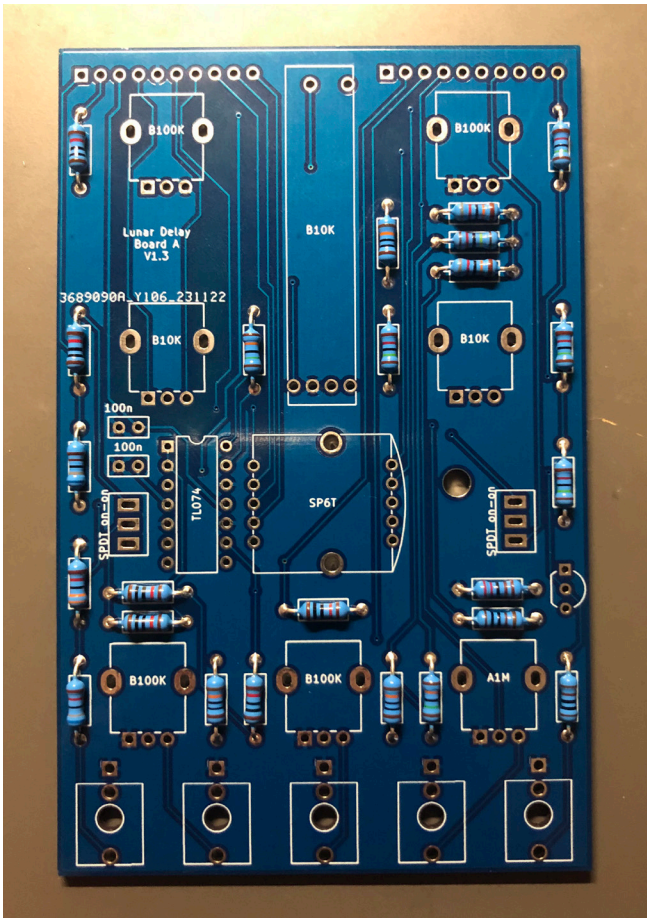
BAG 2 / BOARD A

Qty	Part	Value	Body Marking
2	Resistor	1K	Brown Black Black Brown Brown
1	Resistor	2K2	Red Red Black Brown Brown
1	Resistor	6K8	Blue Grey Black Brown Brown
6	Resistor	10K	Brown Black Black Red Brown
4	Resistor	15K	Brown Green Black Red Brown
1	Resistor	33K	Orange Orange Black Red Brown
1	Resistor	68K	Blue Grey Black Red Brown
3	Resistor	100K	Brown Black Black Orange Brown
2	Resistor	150K	Brown Green Black Orange Brown
2	Resistor	820K	Grey Red Black Orange Brown
1	Resistor	1M2	Brown Red Black Yellow Brown
1	IC Socket	DIP 14	
2	Capacitor	100n	104
1	Transistor	2N3904	
2	Pin Header Male	1x10	
1	Slide Potentiometer	B10K*	B10K or B103
2	Potentiometer	B10K	B10K or B103
4	Potentiometer	B100K	B100K or B104
1	Potentiometer	A1M	A1M or A105
1	Rotary Switch	SP6T	
2	Toggle Switch	SPDT on-on	
5	Jacks	Thonkiconn	
2	Pin Header Female	1x10	
1	IC	TL074	

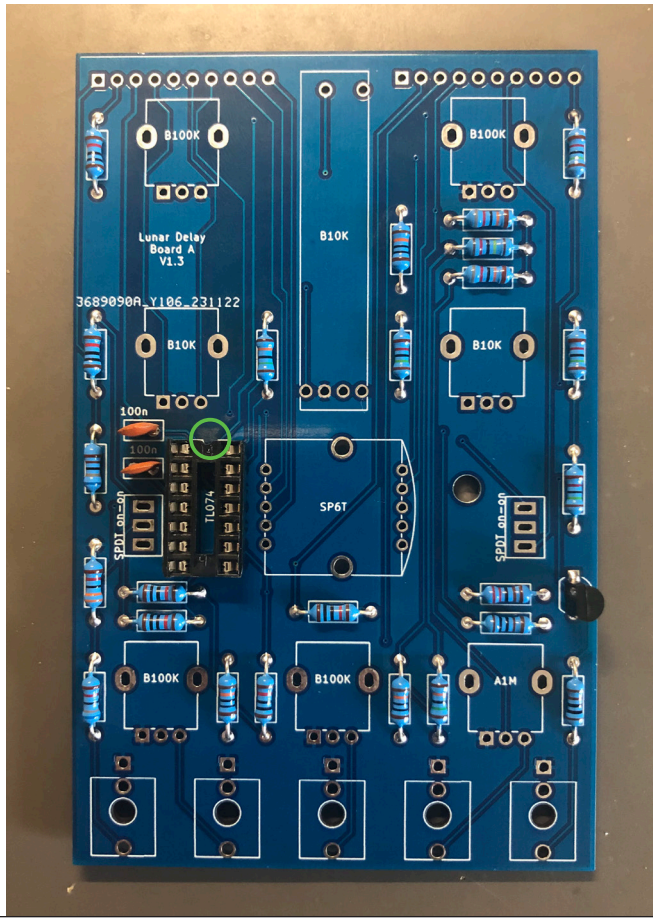
** Slide potentiometer can be substituted with B100K - the most important things are that the slider's part number begins with PTL30-15, the taper is linear, and the LED is red.*

BAG 2 / BOARD A

Begin Board A by soldering the resistors. There are two 1K, one 2K2, one 6K8, six 10K, four 15K, one 33K, one 68K, three 100K, two 150K, two 820K, and one 1M2.

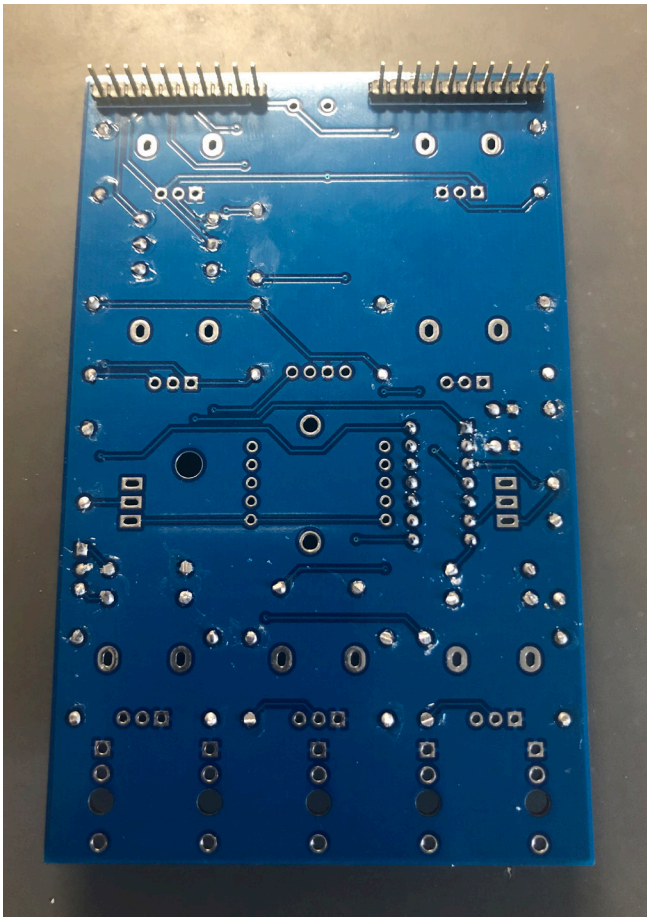


Nest, solder the two 100n ceramic caps, one transistor, and one 14 pin IC socket. The ceramic caps are not polarized, their orientation does not matter. The transistor and IC socket do need to be in the correct orientation. The flat side of the transistor should match the silkscreen drawing, and the notch on the IC socket should also match the silkscreen drawing.



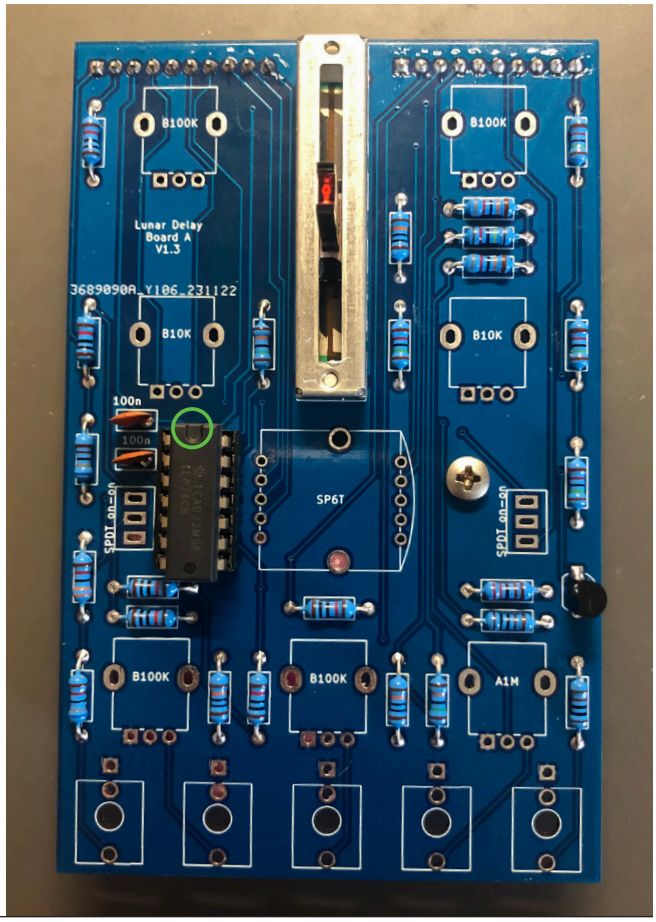
BAG 2 / BOARD A

Place the two male pin headers on the back of Board A, and solder just one pin on each strip. Make sure each header is flush against the PCB. If it isn't, reflow the one pin you soldered while gently pressing the pin header toward the board. Make sure both rows of pin headers are positioned correctly, then solder all remaining pins.



Now move on to the slide pot. This should be soldered before the rest of the front panel components, it just needs to be flush against the PCB. Just like the pin headers, mount the slide pot on the board and solder just one pin. Then reheat that pin while gently pressing the slide pot against the board. This will ensure that it sits flat against the PCB. Once the slide pot is in the correct position, solder the rest of the pins.

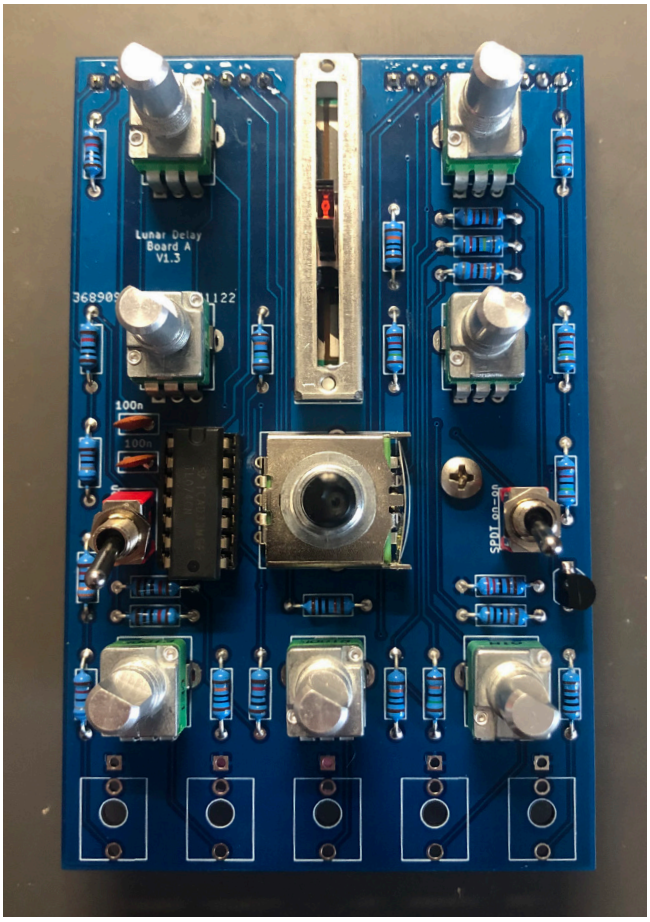
We are almost ready to do the rest of the front panel components. Before we start that though, this is a good time to place the TL074 in its socket and attach the M3 standoff to the back of the board.

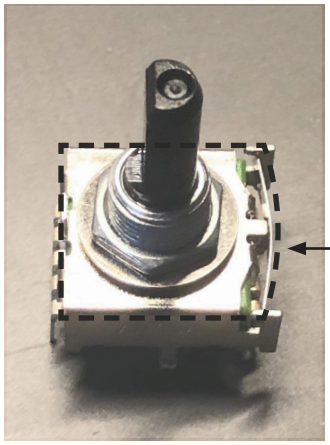


BAG 2 / BOARD A

Some of the front panel components need to be prepared before you place them on the board. Potentiometers may have anti-rotation tabs; if they do, cut those off. The rotary switch has an anti-rotation tab but **do not** cut it off. The tab is just the right height to fit behind the panel. The toggle switches each come with two hex nuts. Screw one hex nut onto each toggle switch before placing it on the PCB, and screw the nuts as far down as they will go. This will keep each switch secure behind the panel, and give them less wiggle room.

Now, place the panel mount components on the board, but **do not solder them yet**. There are five jacks, seven pots (two B10K, four B100K, and one A1M), two toggle switches, and one rotary switch. The rotary switch must be placed in a specific orientation - one side of the body is slightly rounded, this side should be facing toward the right side of the board when placed on the PCB.



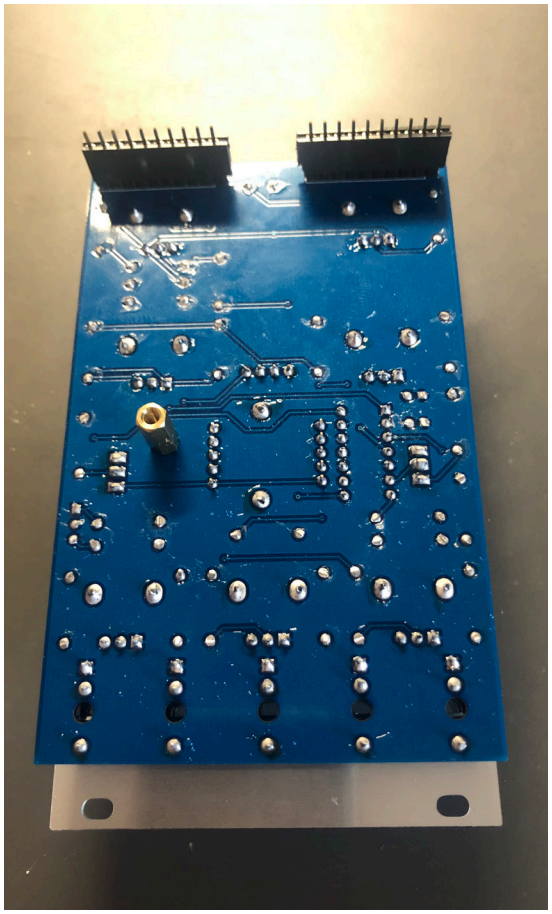


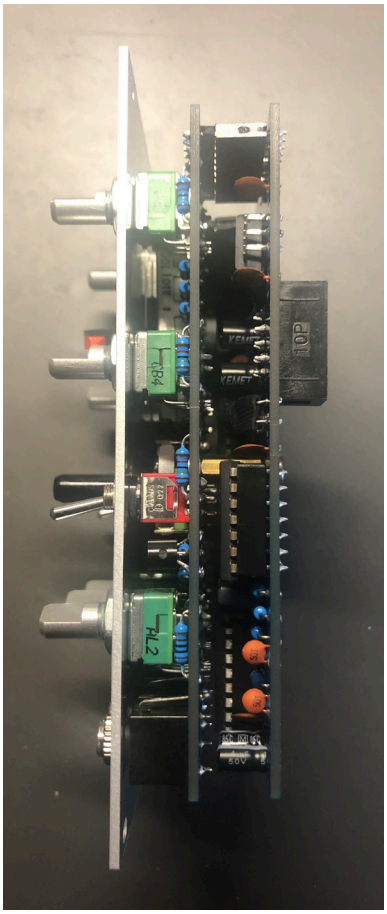
Once all the jacks, pots, and switches are in place, attach the front panel. Screw the jack nuts in place, just finger-tight, and do the same for the toggle switch nuts. For the potentiometers and rotary switch, mount the provided washers on the shafts before screwing down the hex nuts.

BAG 2 / BOARD A

With the front panel in place and all components firmly secured, now we can solder everything. Solder all jacks, potentiometers, toggle switches, and the rotary switch.

Once all components are soldered, place the female pin headers onto their corresponding male headers attached to Board A. Then connect Board B to Board A. Make sure all female pin headers are poking through the holes on Board B, then use the remaining M3 screw to secure Board B to the spacer. Once everything is lined up and in place, solder all female headers to Board B.





Now all the soldering is complete! I like to test the module at this point; in the event there is an issue and you need to take it apart, it is annoying to have to remove all the knobs. Better to test it before putting the knobs on in the first place.

As with all fresh builds, you should test the power rails for continuity and power the module from a bench power supply before plugging it into your modular synth. To test this module, set all knobs to 50% and flip both toggle switches down. Patch an audio signal to the input, preferably something with a wide range of harmonics present. Listen to the output and wiggle each knob to confirm it responds properly. Flip through each of the six filter modes and ensure they each sound as you would expect.

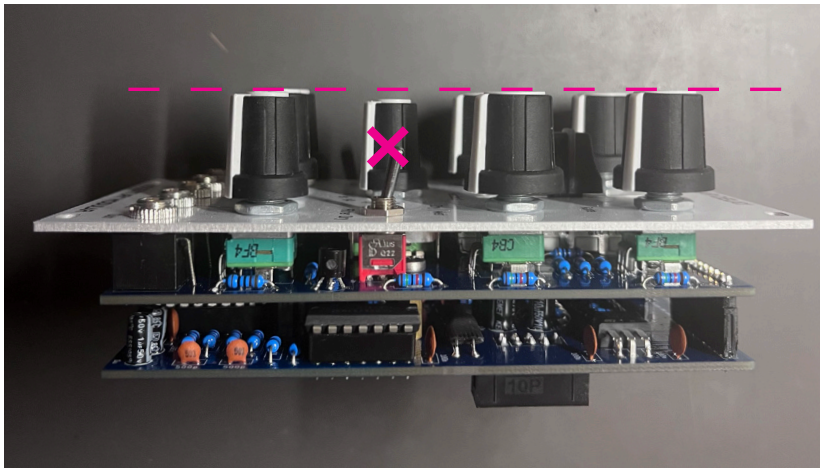
If your module is not working as expected, you will need to do some troubleshooting. Check all polarized components: transistors, voltage regulator, electrolytic capactitor, and the vactrol. Make sure your ICs are in the correct orientation and that you used the correct resistor and capacitor values. Check your soldering work for bridges and cold joints.

If you are really unable to get it working, send some high resolution photos of both sides of each board, along with a detailed description of the issue, to

Support@MidCentury-Modular.com

KNOBS

When placing the knobs on this module, you will run into an issue where the rotary switch's knob sits lower than the potentiometers' knobs. This is because the shaft on the rotary switch is a slightly different shape than the pot shafts. This won't affect the functionality of the module, but it looks a bit unprofessional.



If you want it to look nicer, there is a trick you can do. Put knobs on the seven potentiometers first, and save the rotary switch for last. Place a knob on the rotary switch but do not push it all the way down. Leave it sitting **higher** than the other knobs.

Now, turn the module face down and gently press it against a flat surface such as your desk/table/workbench. This will ensure that all the knobs sit at the same height, the same consistent distance from the panel.

Congratulations! You have successfully built a Lunar Delay. Enjoy your new module!