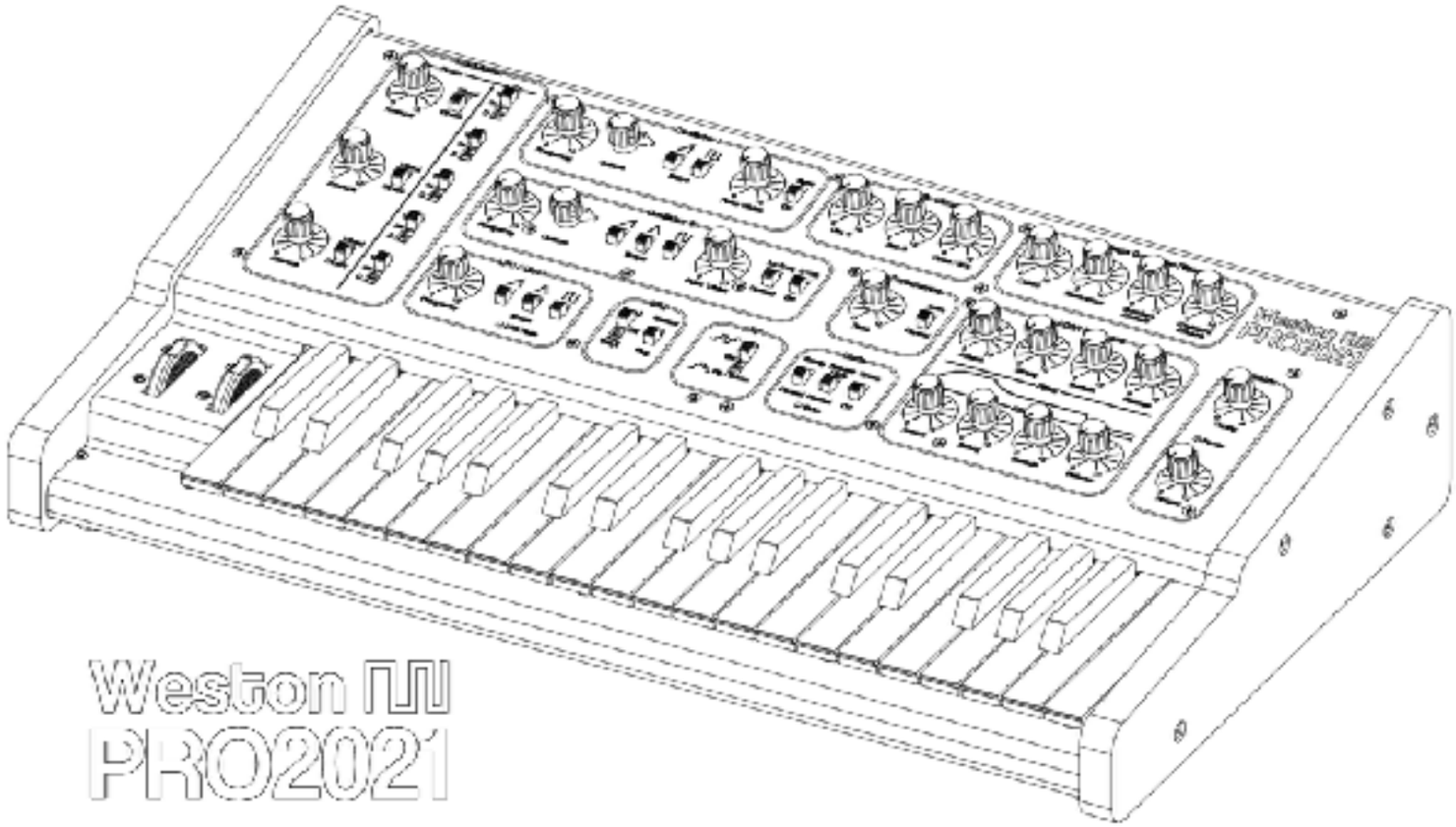
PRO2021 Kit Build Documentation



Revision 03

By Devin Weston / Weston Precision Audio

April 07, 2019

Intro

Overview

PRO2021 is a powerful DIY mono synth which is cloned after the famous Pro-One by Sequential Circuits. Released in 1981, the Pro-One was unique among mono synths in many ways; it exploited the fairly new Curtis Electronics ICs such as the CEM3340 VCO and CEM3320 VCF. It also utilized an Intel microprocessor to handle the keyboard inputs, which as a side benefit, allowed the Pro-One to have a rudimentary on-board sequencer and arpeggiator! In many ways, the Pro-One is a little brother to the legendary predecessor poly synth, the Prophet 5, and one can see many similarities when comparing the 2 schematics. The Pro-One is just much simpler, thankfully :)

Why PRO2021?

Due to supply and the rise in popularity of all things vintage, vintage Pro-Ones tend to fetch a fair amount of money on the used market, and often will need repair, especially to the keybed. The PRO2021 project started with the author wanting to produce a copy of the classic Pro-One using modern components such as the newly available clones of the classic CEM chips from Alfa and Curtis. After finding interest from the DIY community, the author decided to offer DIY kits so many other musicians and hobbyists could build their very own PRO2021 from the ground up!

Notes On Building

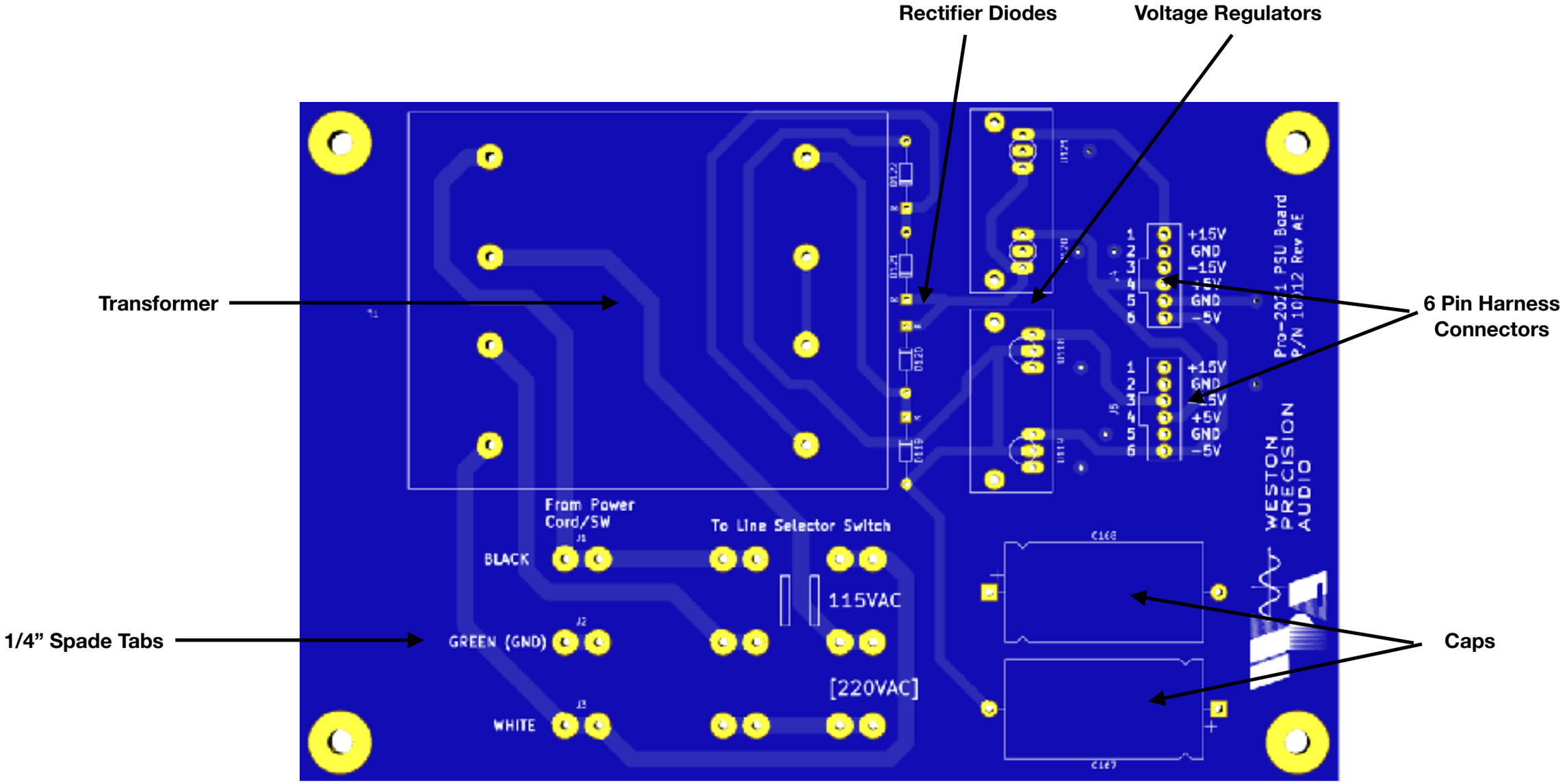
Building a PRO2021 will take some time. There are many components on the 5 PCBs, as well as a fair amount of mechanical pieces. This is not a project you should rush. Have fun with it, and be diligent in checking that all values of components are correct before soldering. It's much easier to get it right the first time than to try and track down an error later!! If you are not familiar with soldering, please take some time to find some tutorials online before starting this project. There is nothing fun about chasing down failures caused by poor soldering on a complex and large circuit such as this synthesizer!

Additional notes:

- PRO2021 uses the Music Technologies Group "TurboCPU" to replace the original Intel 8021 processor. You can find them here to purchase this nice little module: http://www.musictechnologiesgroup.com/ . Take time to check out the TurboCPU manual too, as it contains features beyond the original Pro-One CPU.
- PRO2021 uses a Fatar 37 key OEM keyed. These cannot be purchased in single quantities from Fatar directly. Two sources that carry them are https://www.newgroove.it and TechMechs: https://techsmechsvintagesynth.com/. Another is to scavenge one from a dead synth. Moog, Access, and others have used the Fatar keypads in their synths. Look at ebay, etc..
- The VCO, VCF, and Envelope chips, and Alfa CA3280 clone can be purchased through Erica Synths, Thonk UK, Modular Addict, and more.
- All other electronic components can be purchased through Mouser. Cart links are provided in the original project thread on MuffWiggler: https://www.muffwiggler.com/forum/viewtopic.php?t=209075

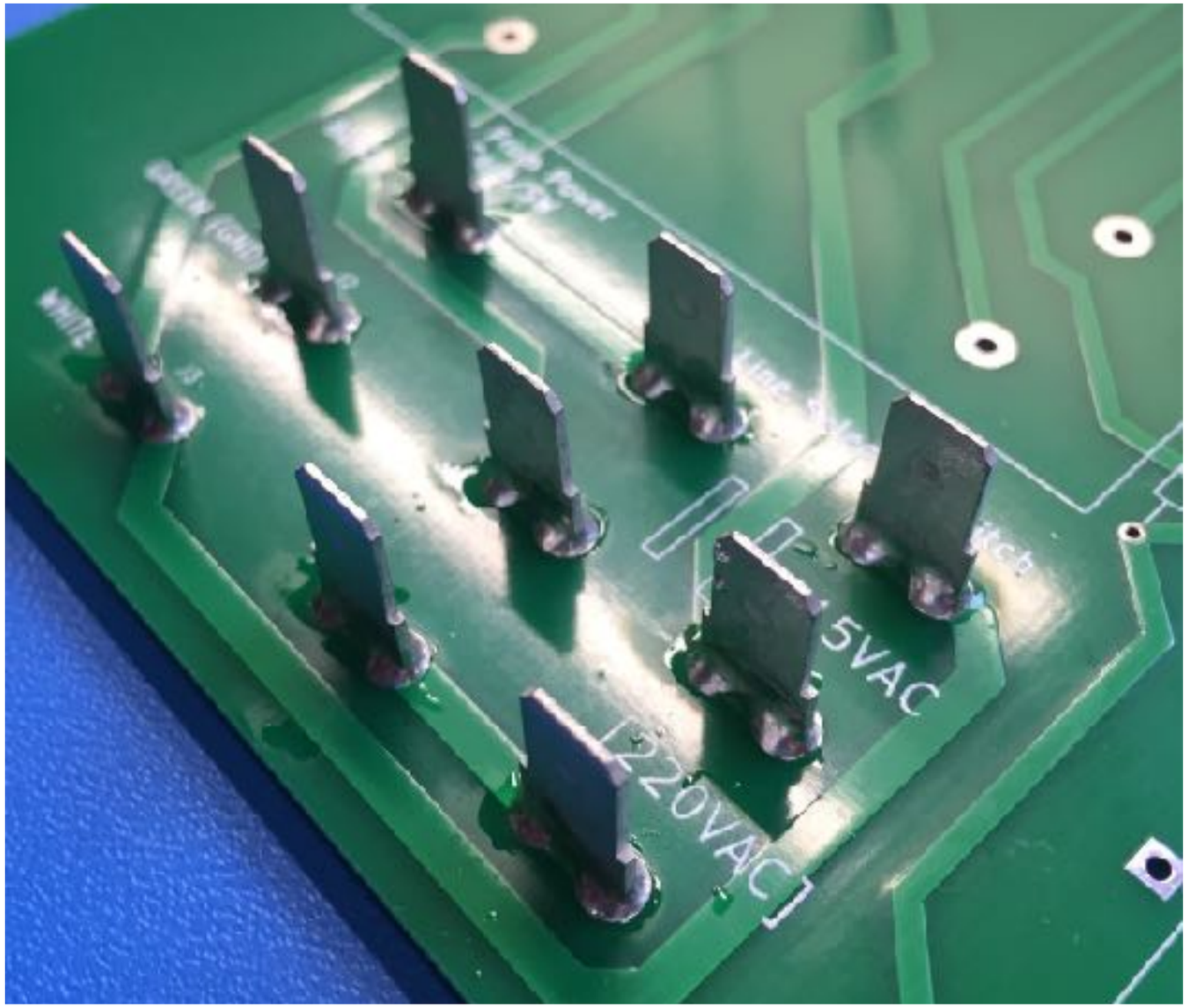
Build the Power Supply

Power Supply PCB Component Locations



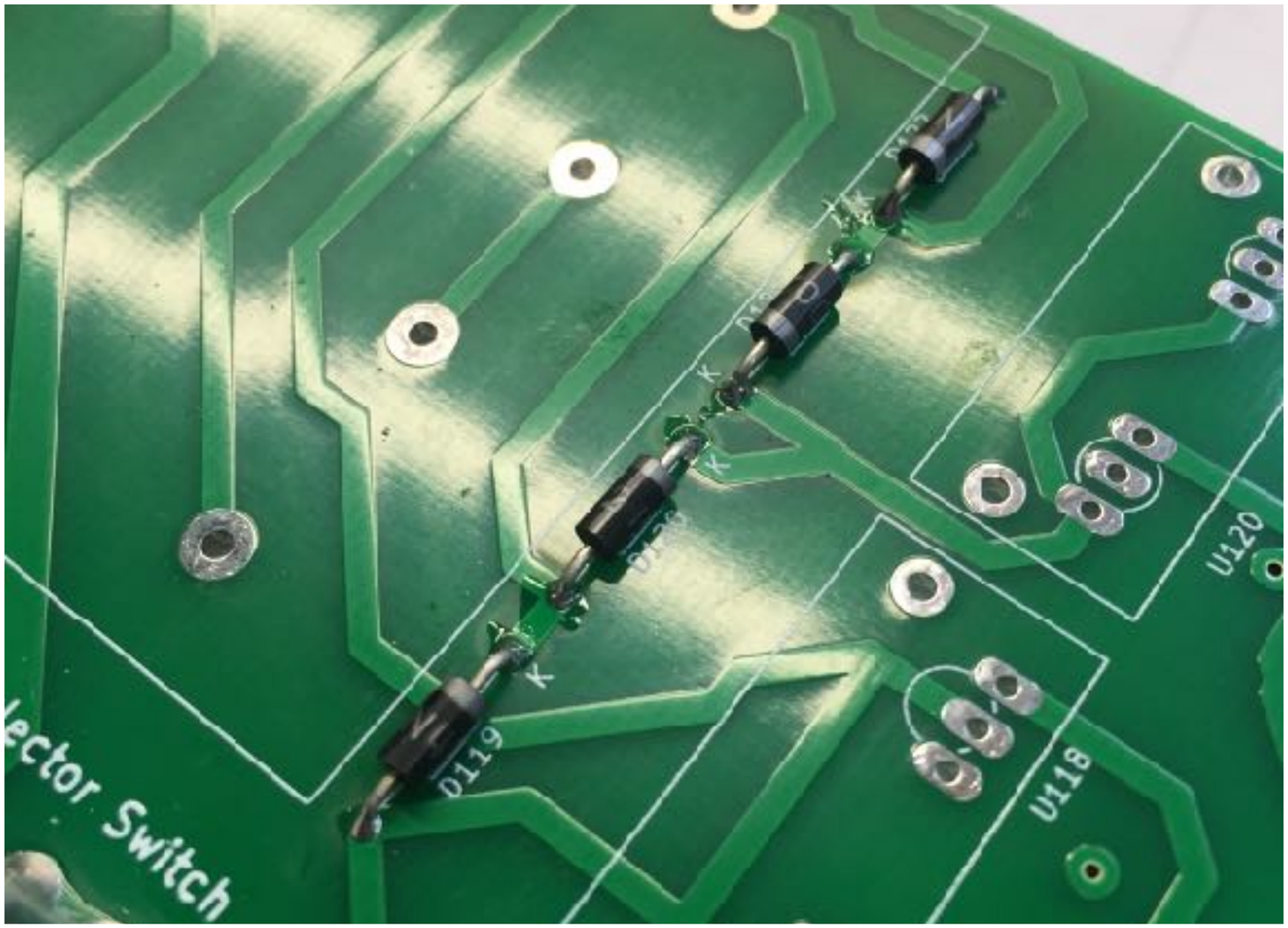
Build the Power Supply

1. Solder on the 1/4” spade tabs. Solder the bottom first, then add a fillet to the top sides for strength. These are big parts and will require a lot of heat to get the solder going.



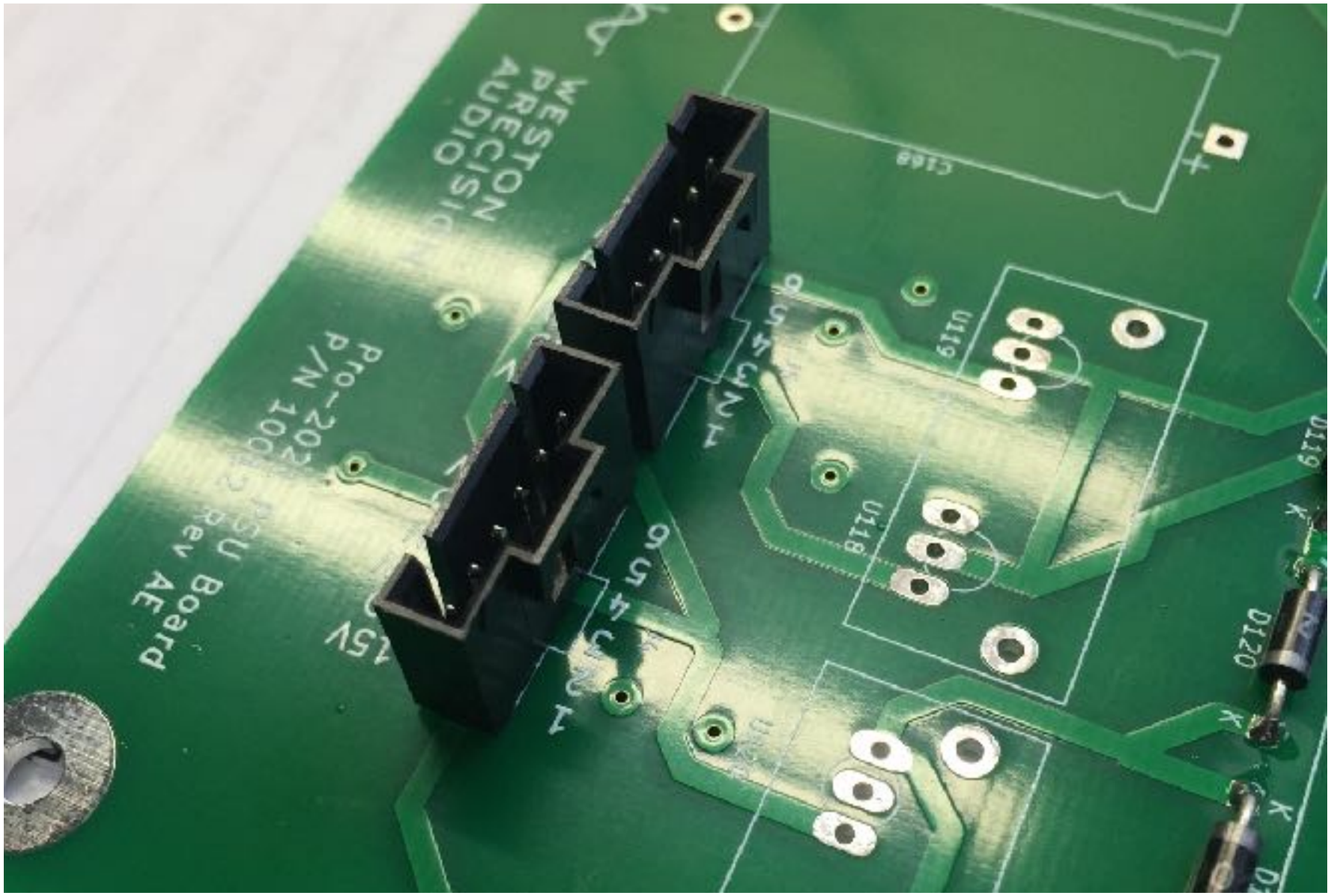
Build the Power Supply

2. Solder the rectifier diodes. Note the orientation of each!!



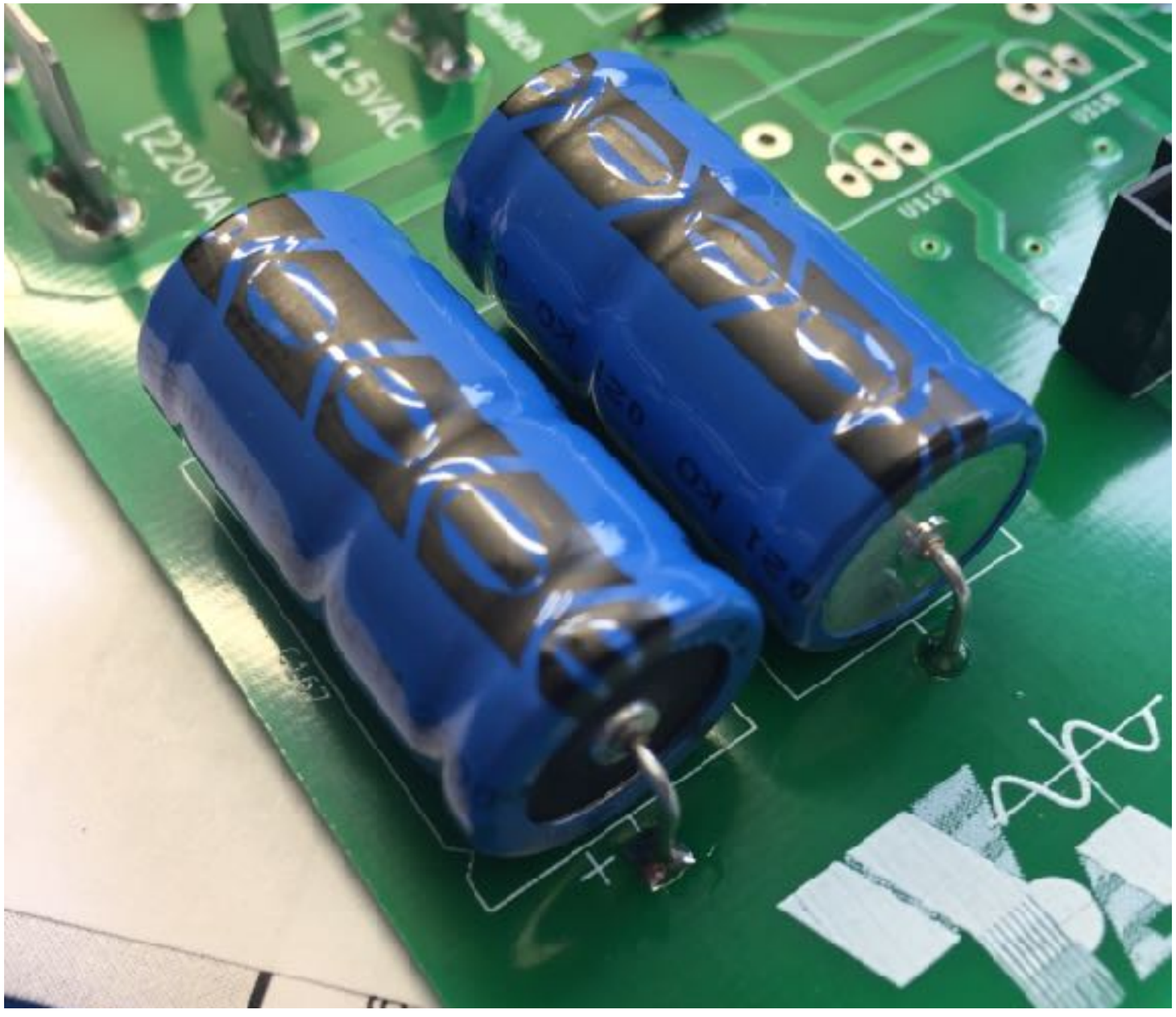
Build the Power Supply

3. Solder the 6 pin Molex headers. Note the orientation!



Build the Power Supply

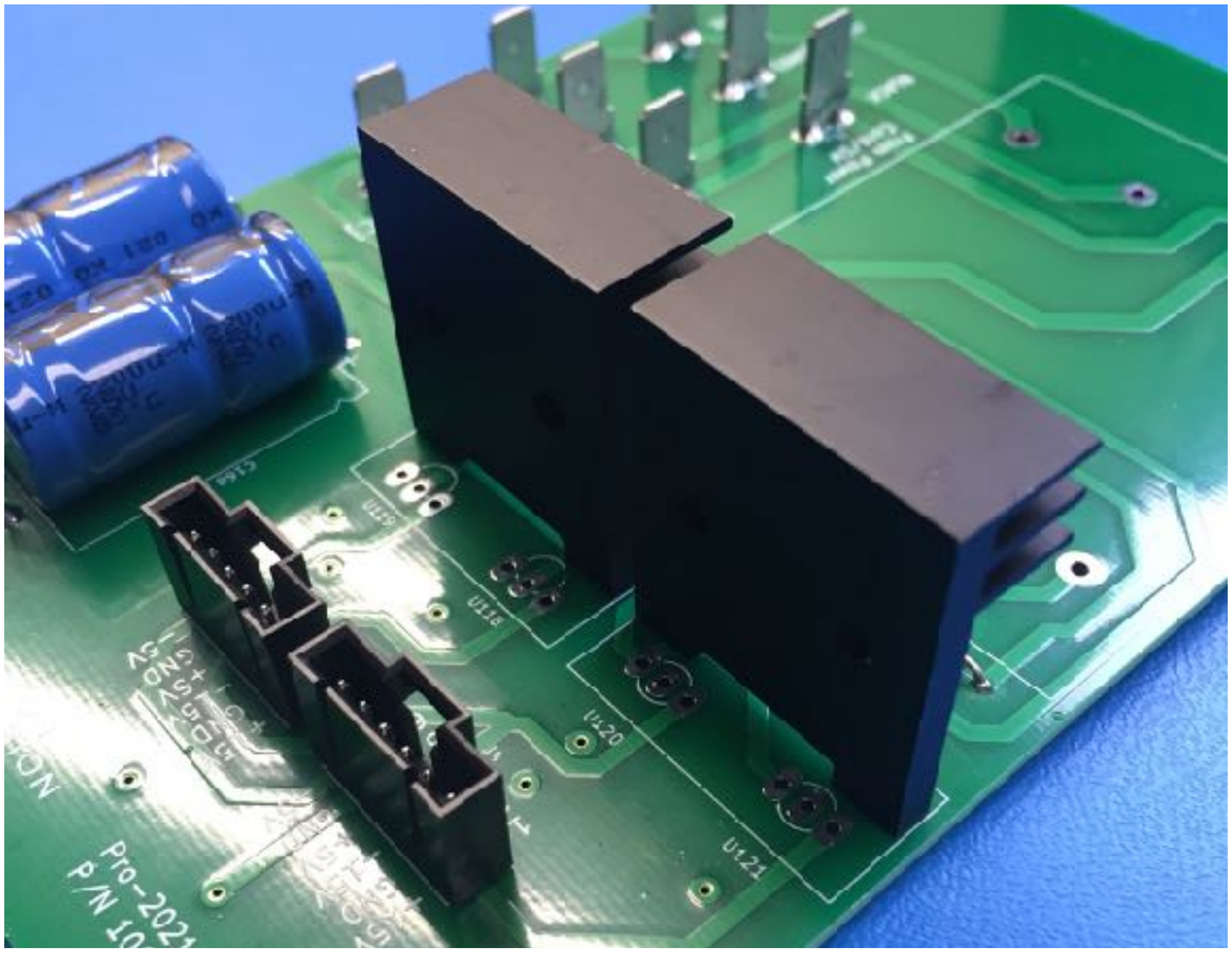
4. Solder the 2200uF caps. Note the orientation!



Build the Power Supply

5. Solder on the heat sinks in the orientation shown. Make sure they are straight and flush.

6. Clean the board with an applicable flux removal process and completely dry the board.



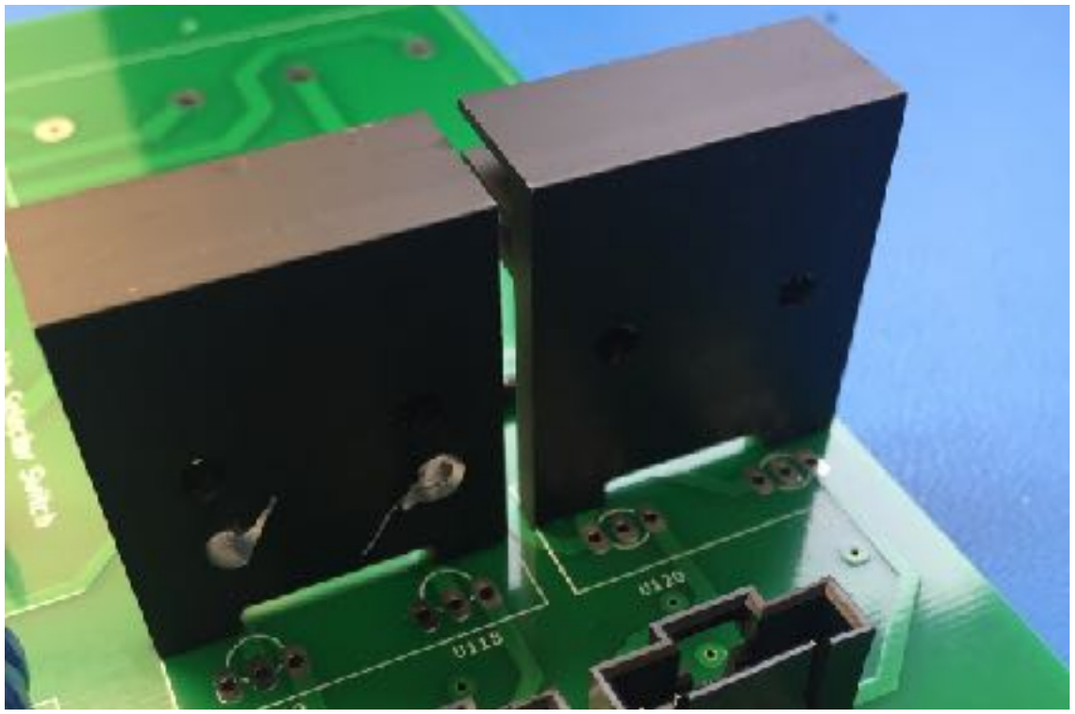
Build the Power Supply

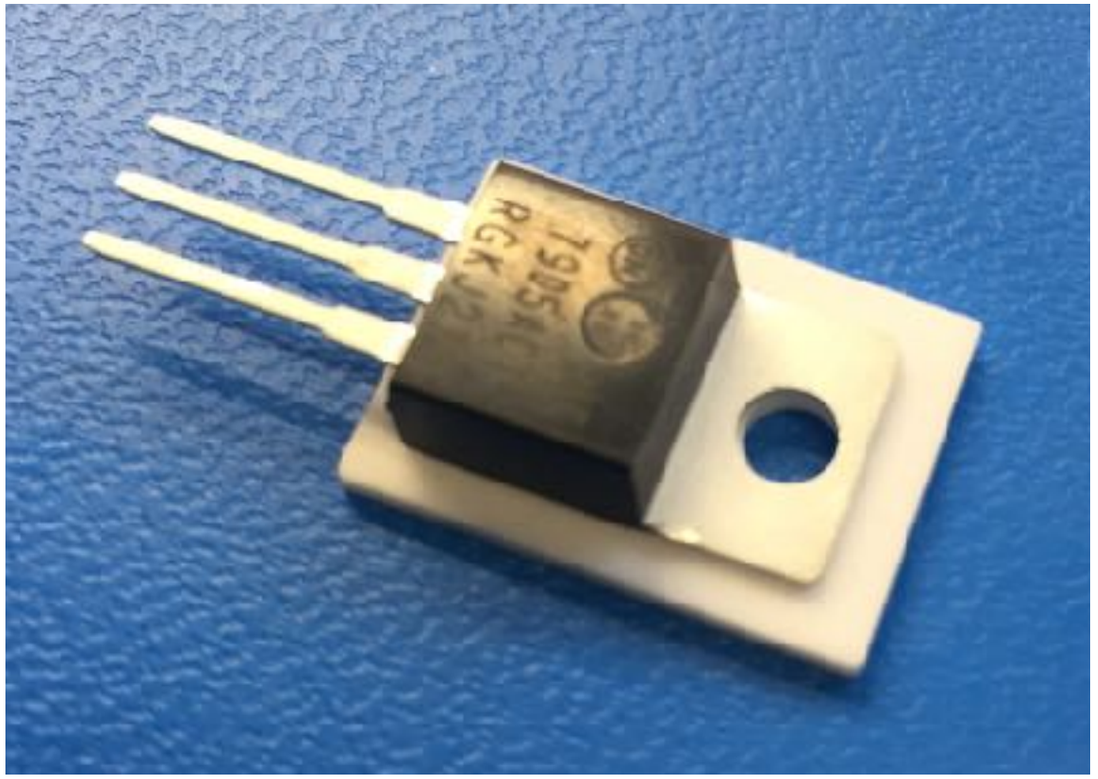
7. Put a dab of heat sink compound as shown where the **negative** regulators go.

8. Put a dab of heat sink compound on each **negative** regulator and stick the mica insulator to them.

Voltage Regulators For PSU

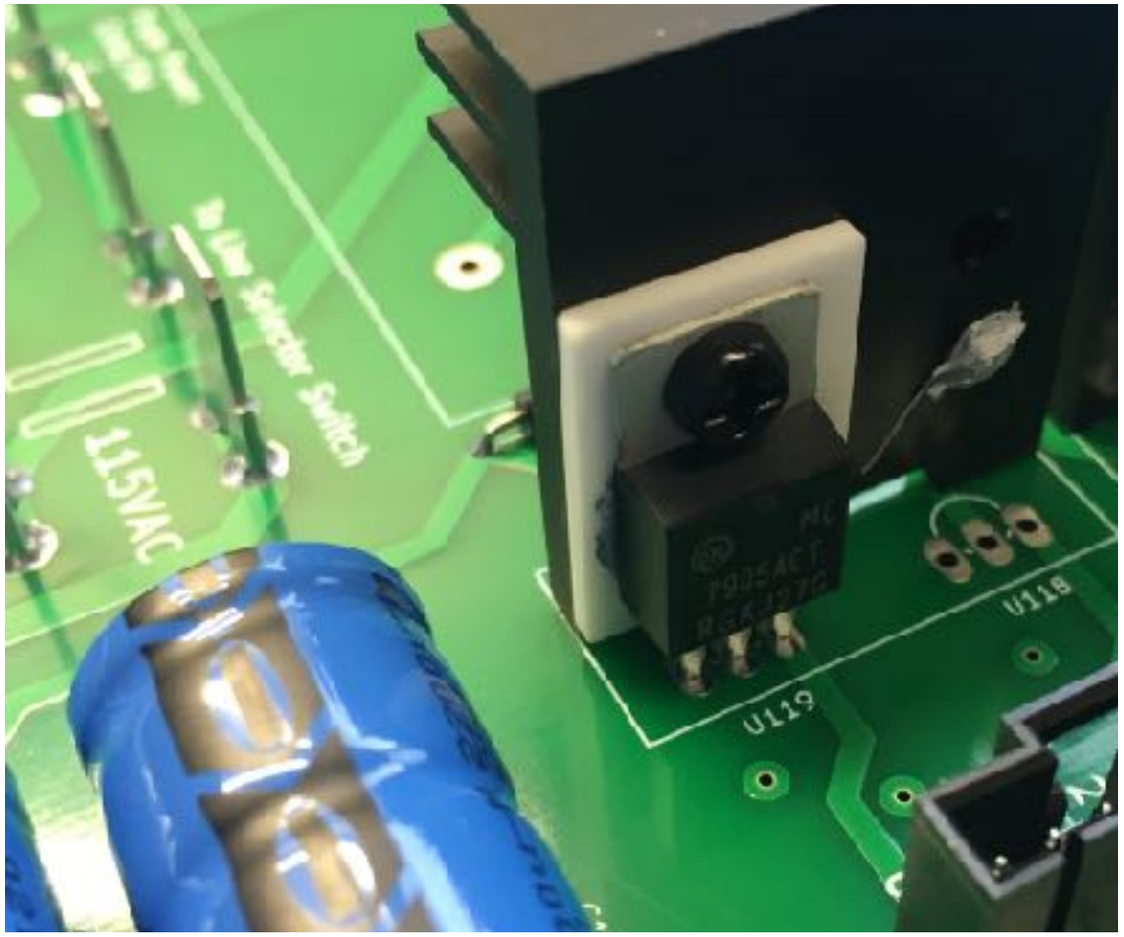
Postition	Regulator	Output
U118	7915	-15V
U119	7905	-5V
U120	7805	+5V
U121	7815	+15

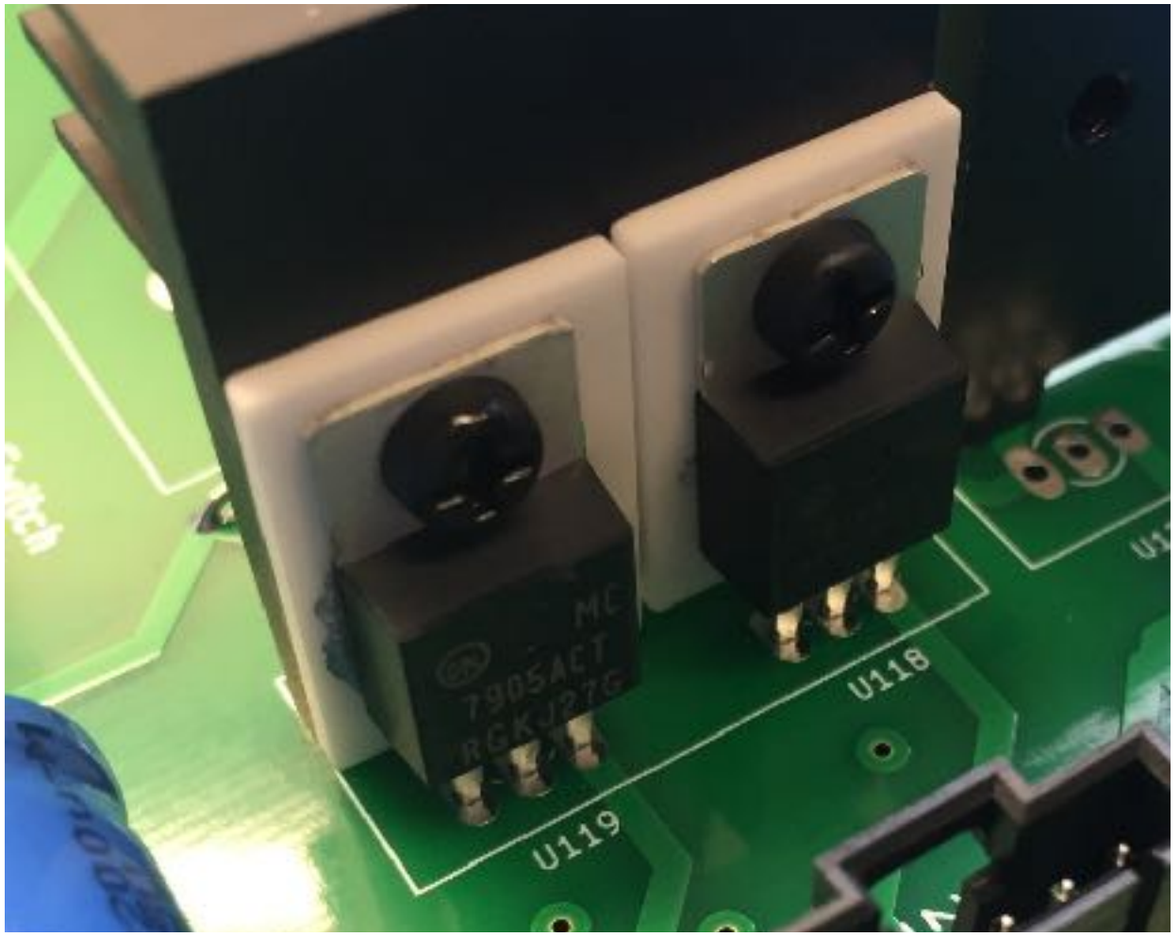




Build the Power Supply

9. Place each negative regulator into the correct spot and secure them with an M3 x 6 plastic fastener. Warning: Do not over torque the plastic screws. They are obviously not Strong as a metal screw but are required here to obtain the necessary electric insulation from the heat sink.





Build the Power Supply

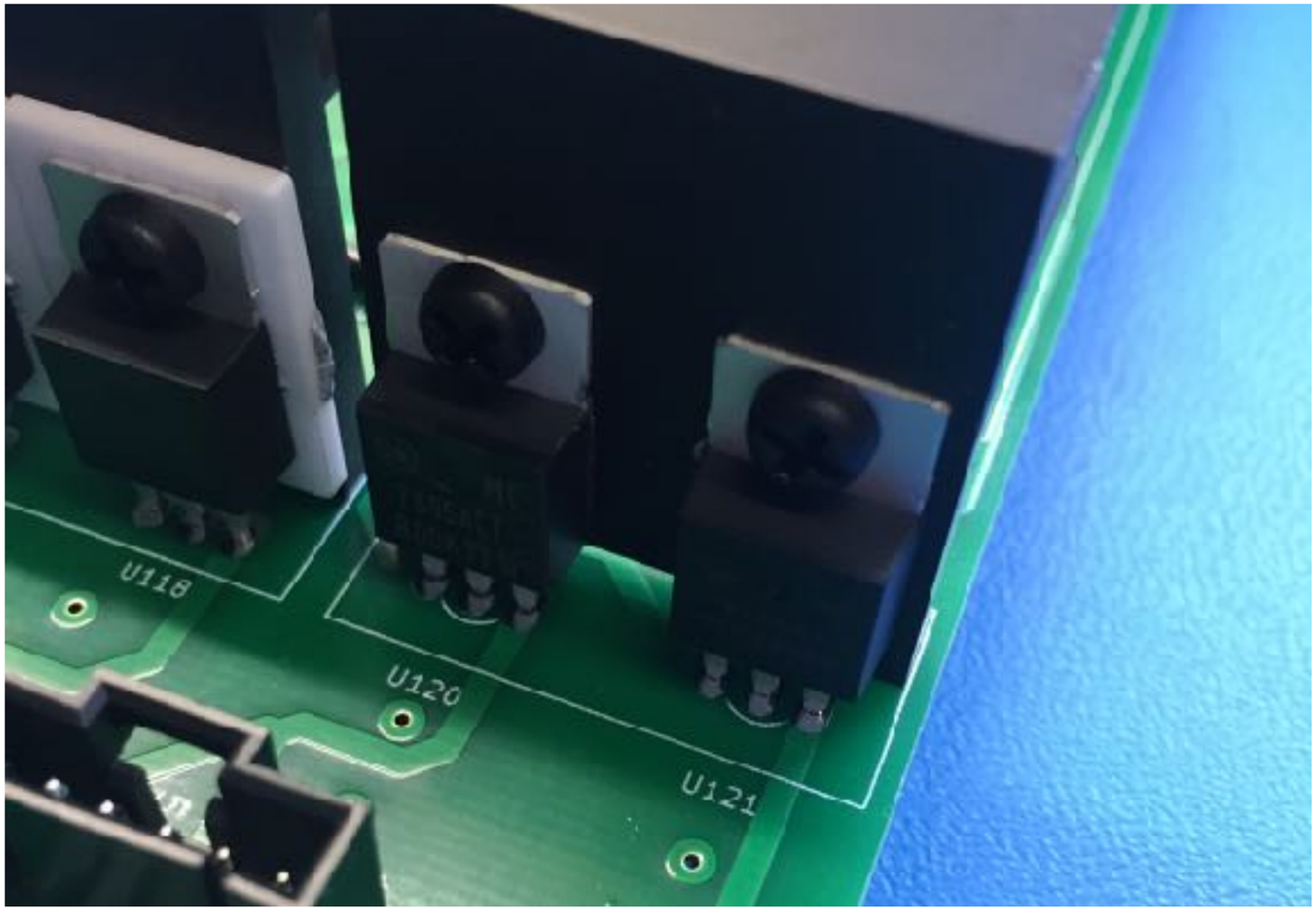
10. Repeat the previous step for the 2 **positive** voltage regulators. Everything is the same except the positive regulators will not use the mica spacers, as can be seen by the different distance from the heat sink to the pins.

11. Clean the board of flux, if you prefer, using an applicable method for your solder.

11. Flip the board over and solder the terminals of all the voltage regulators and trim the legs.

Voltage Regulators For PSU

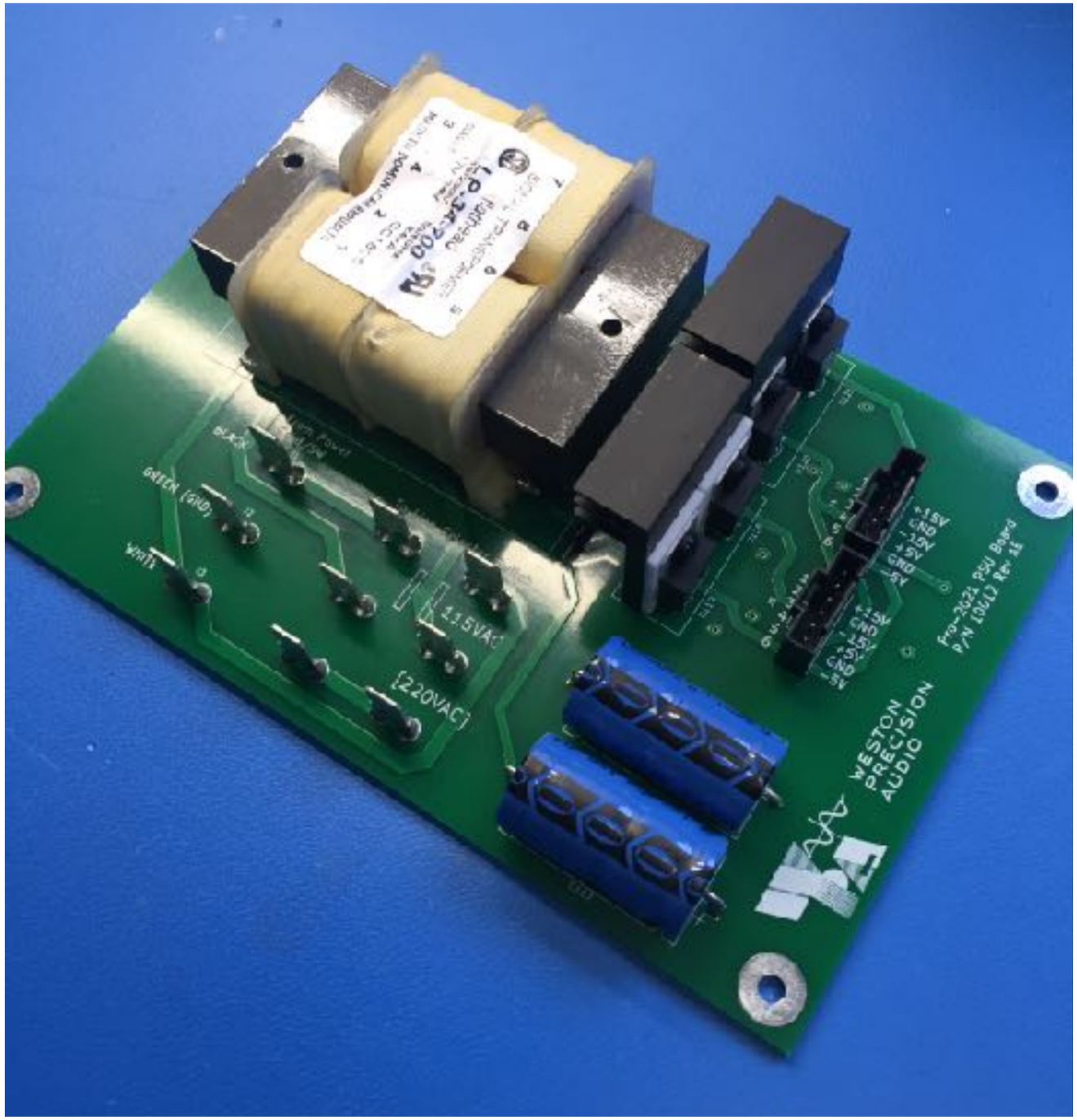
Postition	Regulator	Output
U118	7915	-15V
U119	7905	-5V
U120	7805	+5V
U121	7815	+15



Build the Power Supply

12. Solder on the transformer. Note the pins 1,2,3,4 (supply side go to the left edge of the board (furthest from the regulators)

13. That’s it! PSU build is done!



Build PCB “A”

Start by soldering all the resistors using the table below. All should be 1% unless otherwise specified. All of the resistors are through hole type for this board.

Tips:

-It’s easiest to print out these sheets and check off each “R” value as you install it with a red pen!

-Use a template to pre-bend the legs at a right angle to the right hole spacing.

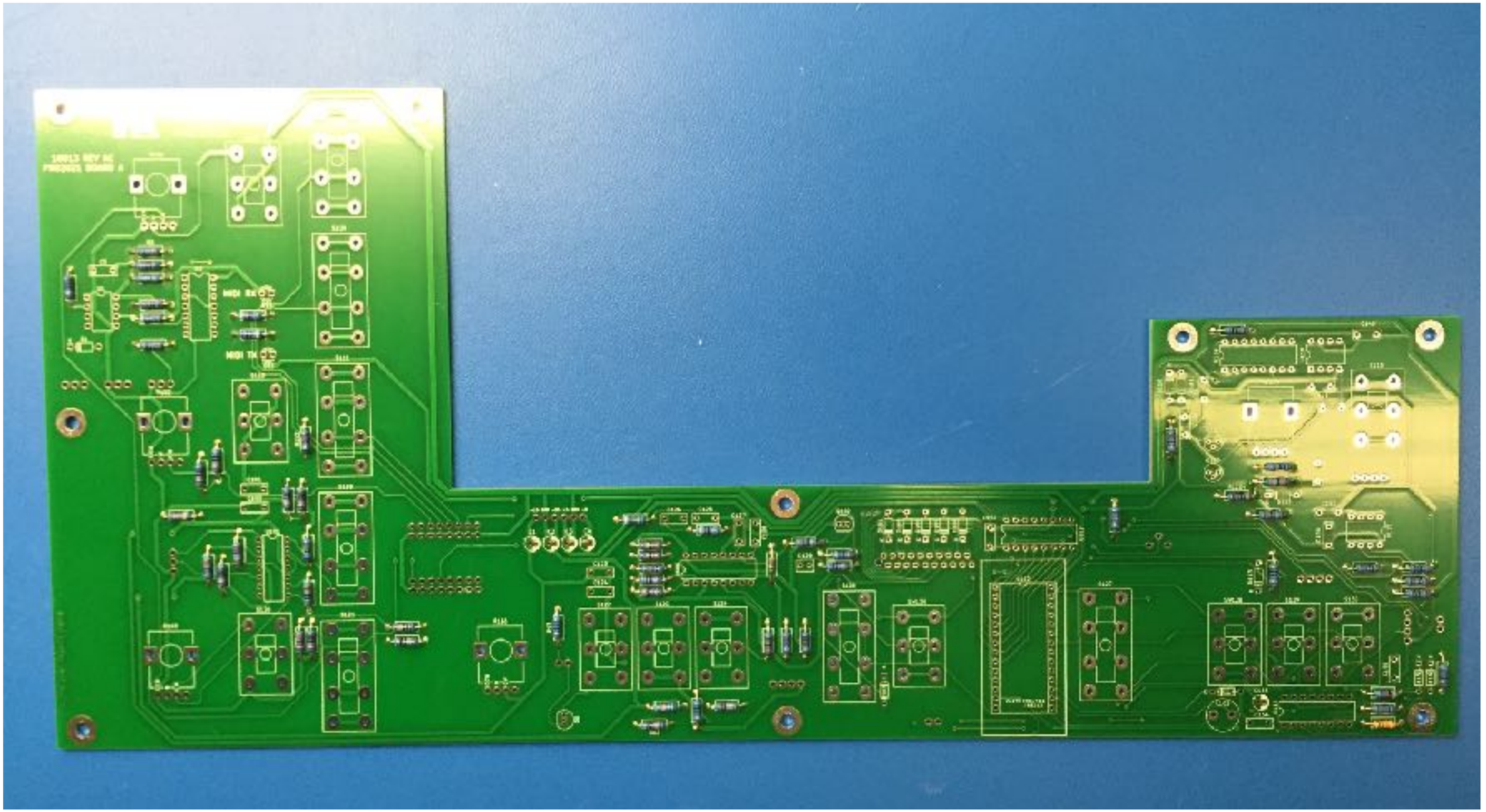
-Don’t forget to trim your legs after soldering!

PRO2021-A Resistors

Location	Value
R1 R3 R4 R5 R6	220 Ohm
R2	4.7K
R104	43K
R109	220K
R1100	820 Ohm
R1101	47K
R1102 R1103 R1135 R1136 R9	10K
R1104	560K
R1105	110k
R1106	820k
R1107	12k
R1108	62k
R112 R113	68K
R115	33K
R1165	13k
R1166	160k
R1167	390k
R1176 R1178	10K
R1177 R1179	150K
R1180	1.5K
R1183 R1185	10k
R1184	24k
R1186	15k
R1187 R13	1.5K
R105 R106 R107 R108 R12 R110 R111 R114 R116 R12 R151	100K
R189	2.2M
R190	820K
R152 R191	200K
R192	5.1K
R1164 R193	30K
R194 R195 R7 R8	470
R198	1.8k

Build PCB “A”

All resistors added to PCB “A”



Build PCB “A”

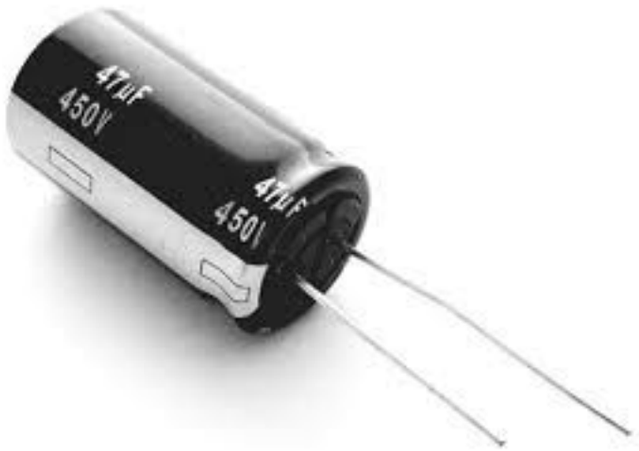
Next add the capacitors. Make sure all of the film capacitors (the boxy ones) are flush flat with the PCBs and not sticking up, except for C154. C154 should have the legs bent and the cap played on it’s side as shown in the picture below. Note the orientation of the electrolytic capacitors!

PRO2021-A Capacitors

Location	Value	Capacitor Type
C1 C2 C3 C4	2.2 35V	Electrolytic
C125 C126 C127 C141	0.01	Rectangular Film
C129	200pF	Multilayer Ceramic
C101 C102 C123 C124 C128 C138 C139 C140 C142 C149 C150 C151 C152 C154 C5	0.1	Rectangular Film
C153	1.0uF 10V	Electrolytic
C155	0.001	Rectangular Film

C154 Special Treatment

Electrolytic Capacitor (polarized!)



Rectangular Film Capacitor (not polarized)

Multilayer Ceramic (MLCC) Capacitor (not polarized)

Build PCB “A”

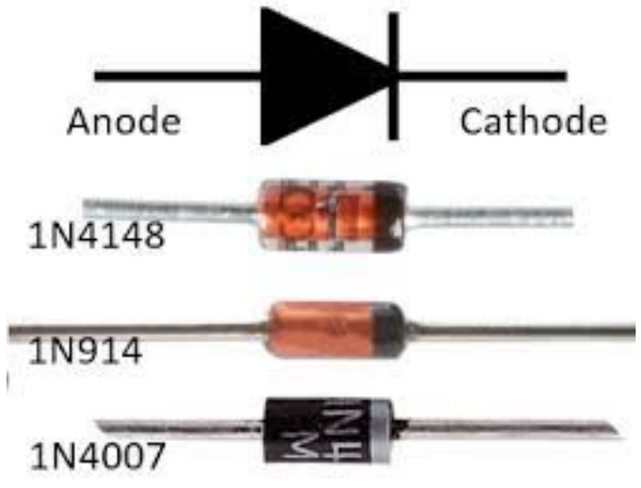
Next add the diodes and PCB mount LEDs (optional - see notes in table). Note the orientation of both the diodes and the LEDs! These are polarized components and must be installed correctly.

PRO2021-A Diode Locations

Location	Value	Note
D1	1N4148	
D103	OPTIONAL 1N914	Changes default behavior of Arpeggiator options of “UP / UP-DOWN” to “UP / DOWN”. Recommended to leave off at first.
D104 D105 D106 D107 D108 D113 D114 D115	1N914	
D110 D111 D112 D116	1N914	
DS1 DS2	3mm LED (Optional)	PCB mount MIDI receive (Rx) and Transmit (Tx) LEDs. Rx should blip every time a key is pressed. Helpful for knowing board is working when getting set up, but not necessary for operation!

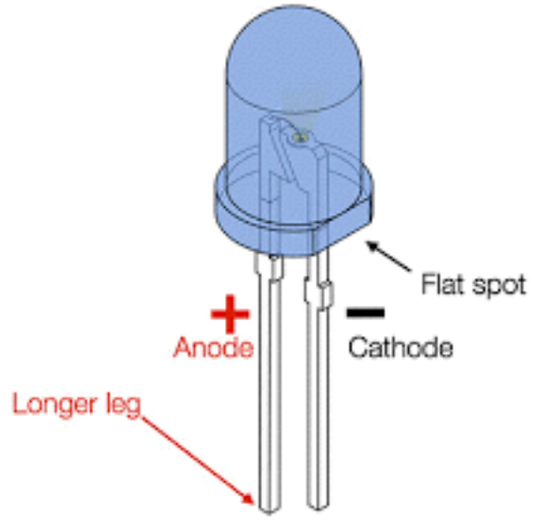
Diodes

Note orientation



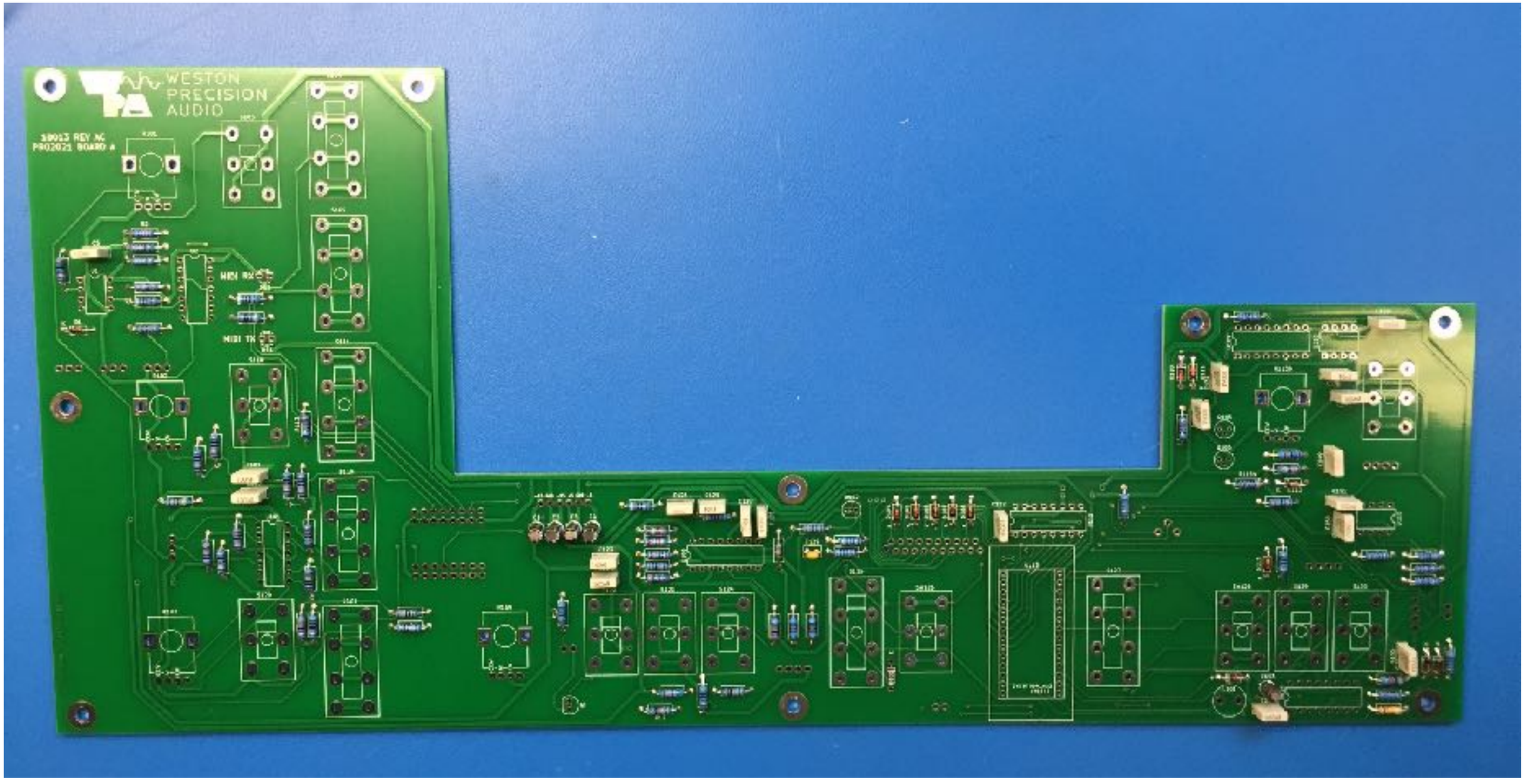
LEDs

Note orientation



Build PCB “A”

All caps and diodes added to PCB “A”



Build PCB “A”

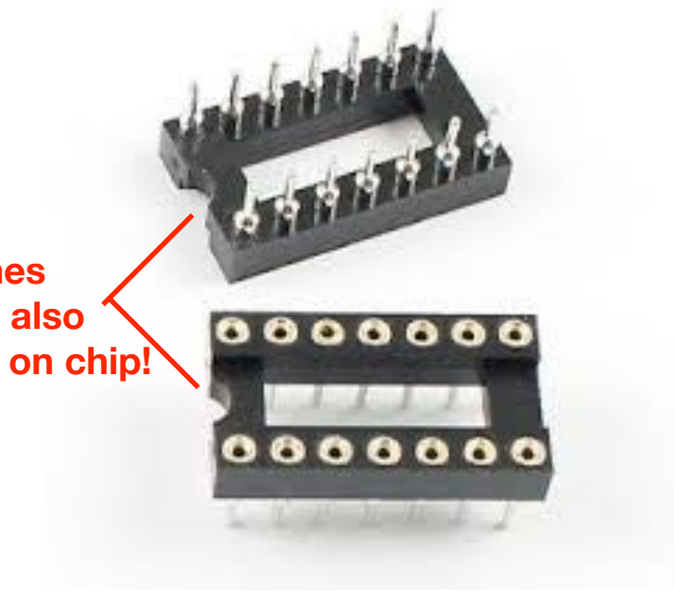
At this point in the build, you will add the special Dual-Wiping DIP socket used for the TurboCPU module. You should also add DIP sockets for the rest of the ICs if you are using them. If not, move on to the next step after installing the TurboCPU DIP socket. We HIGHLY recommend using DIP sockets on all chips, as it makes diagnoses and swapping chips MUCH easier later on if necessary. Always use the round style DIP sockets (except for the special TurboCPU one!), not the side wiping type, which can be highly unreliable.

Please come back and see this table later when it is time to install the chips into their sockets.

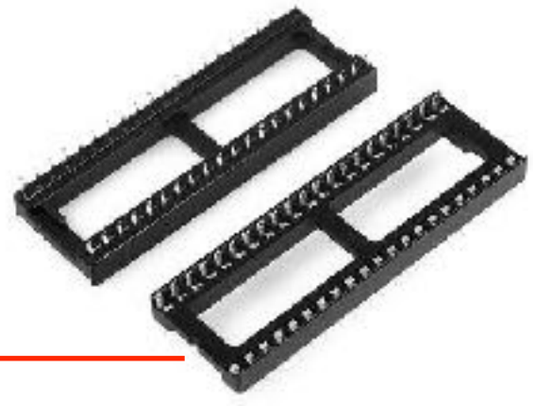
PRO2021-A IC/Sockets

Location	Chip Type	Socket Type
U1	6N137	DIP-8
U101	LM348	DIP-14
U106	CEM3340	DIP-16
U109	LM13700	DIP-16
U110	TL081	DIP-8
U112	AD558JN	DIP-16
U113	Intel8021 / TurboCPU	DIP-28 DUAL SIDE WIPING
U114	4049	DIP-16
U115	MC1458	DIP-8
U2	7414	DIP-14

Round pin DIP socket



Dual side-wiping DIP socket (For Turbo CPU).



Build PCB “A”

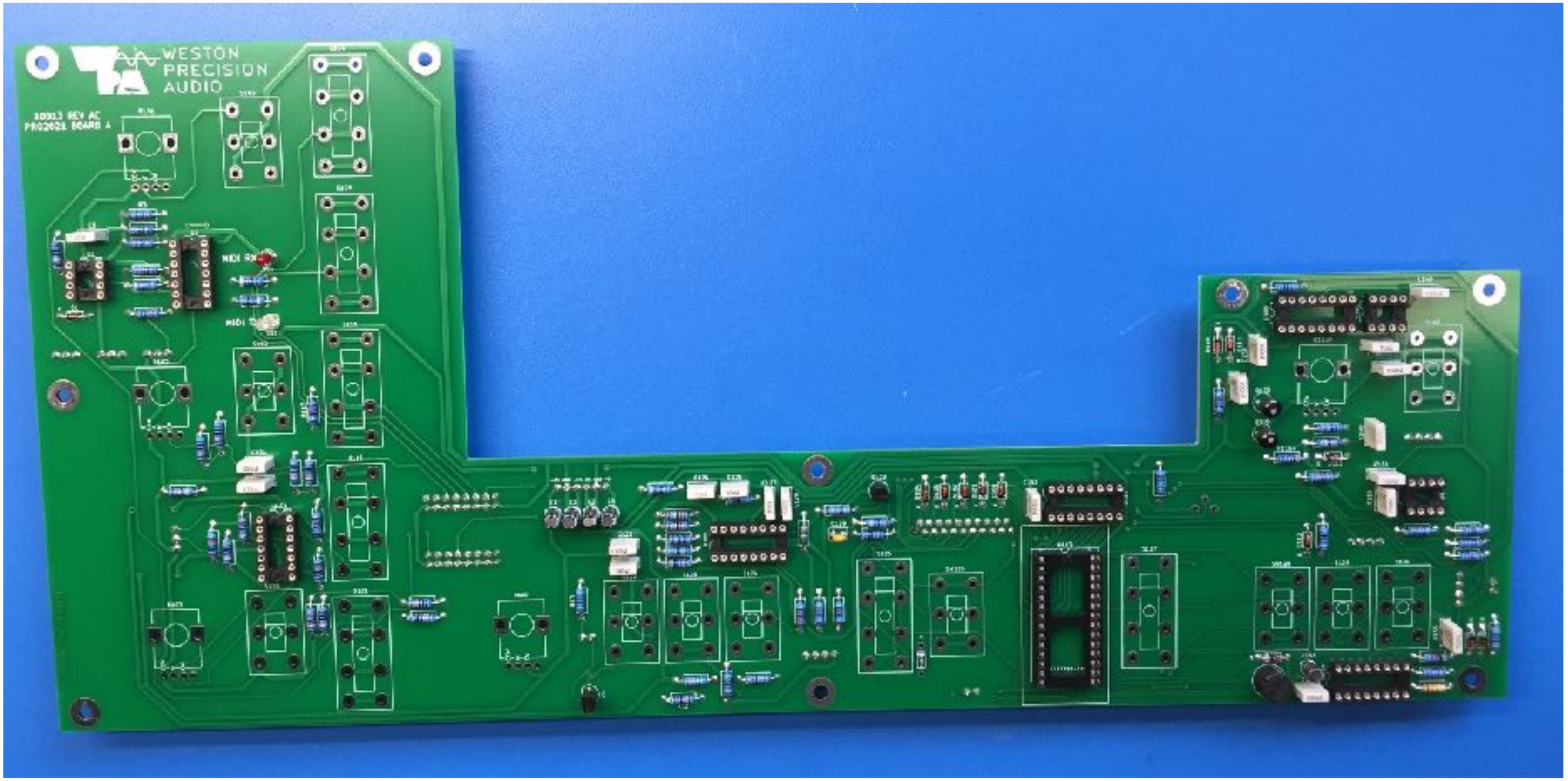
Next add the transistors to the front of the board. The orientation will be obvious.

PCB “A” Transistors

Q1 Q102	2N3904
Q105 Q106	2N4250

Build PCB “A”

All DIP sockets and transistors added to PCB “A”



Build PCB “A”

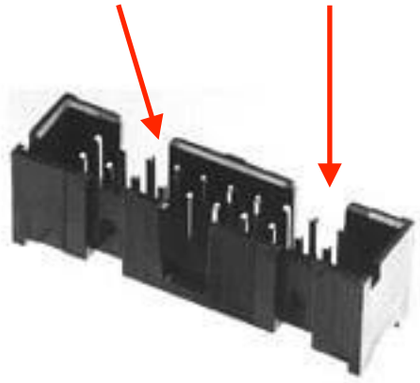
Next add all of the connector headers. These, as the silkscreen on the PCB shows, go on the BACK of the PCB. Their orientation is also noted via the outline shown on the PCB.

PRO2021-A Connectors

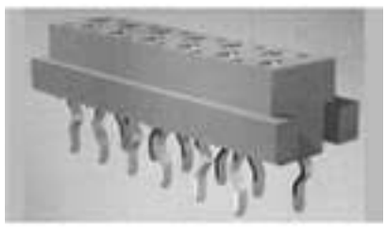
J1	GATE LED	Molex 2 Pin Male Header
J10	MOD WH	Molex 3 Pin Male Header
J109	Keyboard	20 Pin MicroMatch PCB Header
J11	Power	Molex 6 Pin Male Header
J12	MIDI IN	Molex 3 Pin Male Header
J13	MIDI OUT	Molex 3 Pin Male Header
J14	MIDI THRU	Molex 3 Pin Male Header
J15	MIDI INTERFACE TO TURBO CPU	Molex 2 Pin Male Header
J16	A-B Interconnect RIGHT	16 Pin Ribbon Header
J17	A-B Interconnect LEFT	16 Pin Ribbon Header
J18	LFO LED	Molex 2 Pin Male Header
J2	CV In	Molex 4 Pin Male Header
J3	GATE/CLK IN	Molex 4 Pin Male Header
J4	CV Out	Molex 4 Pin Male Header
J5	GATE OUT	Molex 4 Pin Male Header

Molex “SL” Header
(retention tab shown on PCB)

16pin Ribbon Header
(retention tabs shown on PCB)



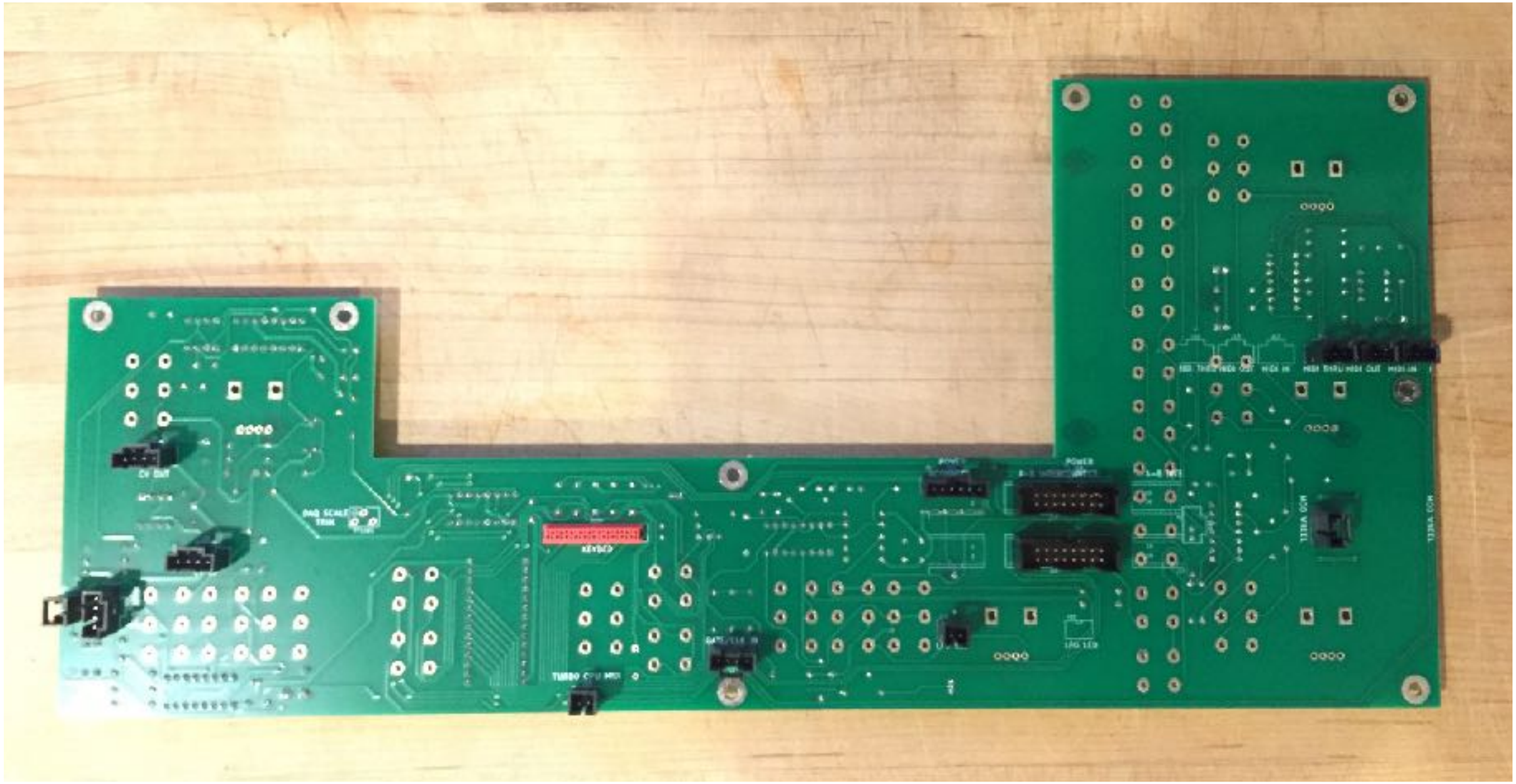
MicroMatch Keyboard header



Build PCB “A”

All connectors added to PCB “A”

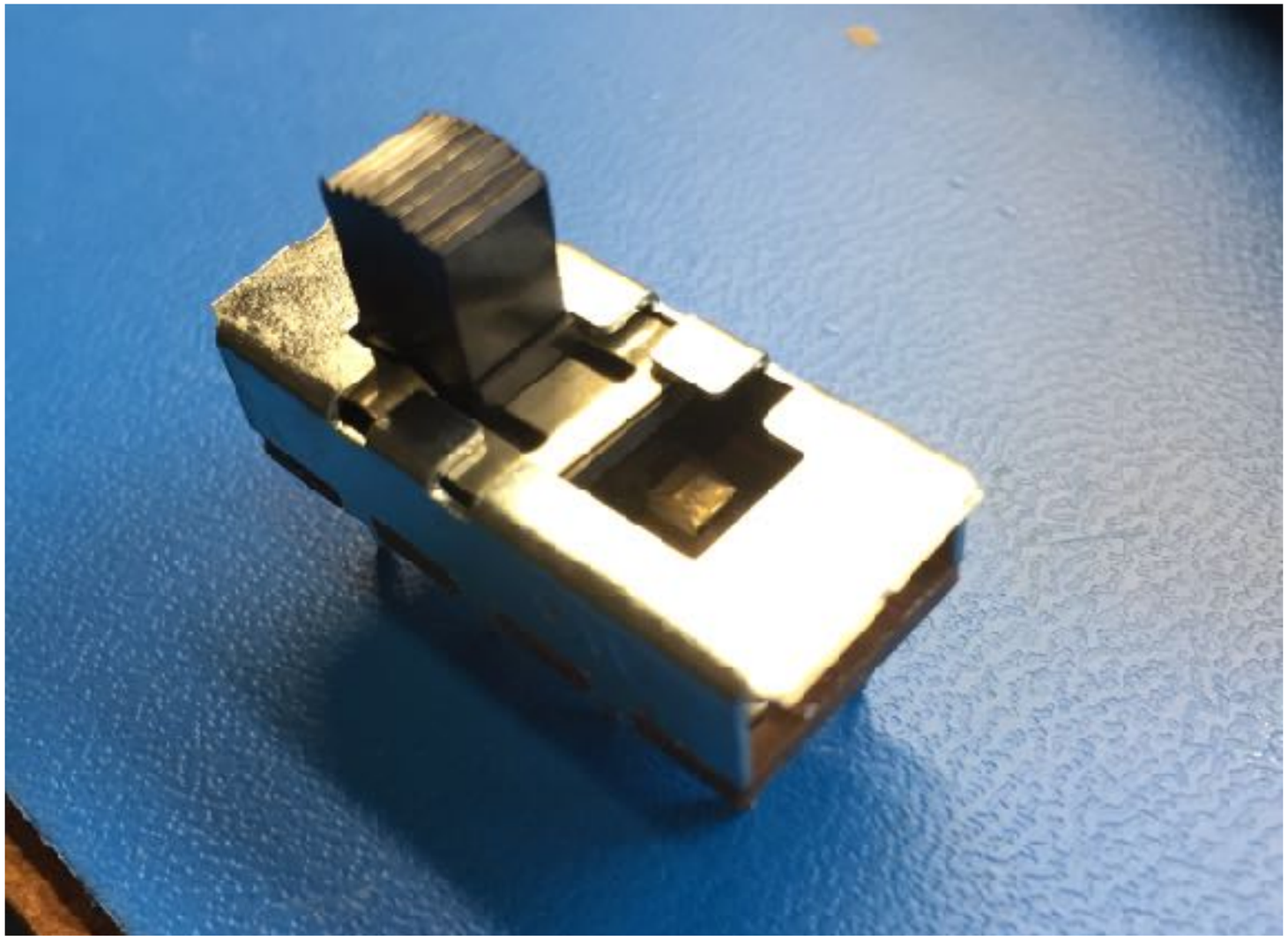
At this point, if you are going to do a flux clean on the board, do so with the appropriate process for your solder.



Build PCB “A”

1. Remove the “wings” from all of the 3 position slide switches. You can do this with large cutting pliers, or just by grabbing the tab with regular flat front pliers and wiggling the tab until the metal fatigues and the tab breaks off.
2. Remove ONE wing (you can do both if you want, it’s just extra work) from 8 of the dual position slide switches in the same manner as the previous step.

The 3 position switches should look as shown below.



Build PCB “A”

1. Before soldering any potentiometers, decide which knobs you will be using! If you are using the slip-on knobs purchased from Altitude909 (Michigan Synth Works), you will need to shorten EVERY pot by 4.5-5mm.
2. If you are using the set-screw style “Jove” knobs from Modular addict, you don’t need to do anything to the pots.

“Jove” Knobs (setscrew)

No need to shorten pots

Slip-on knobs

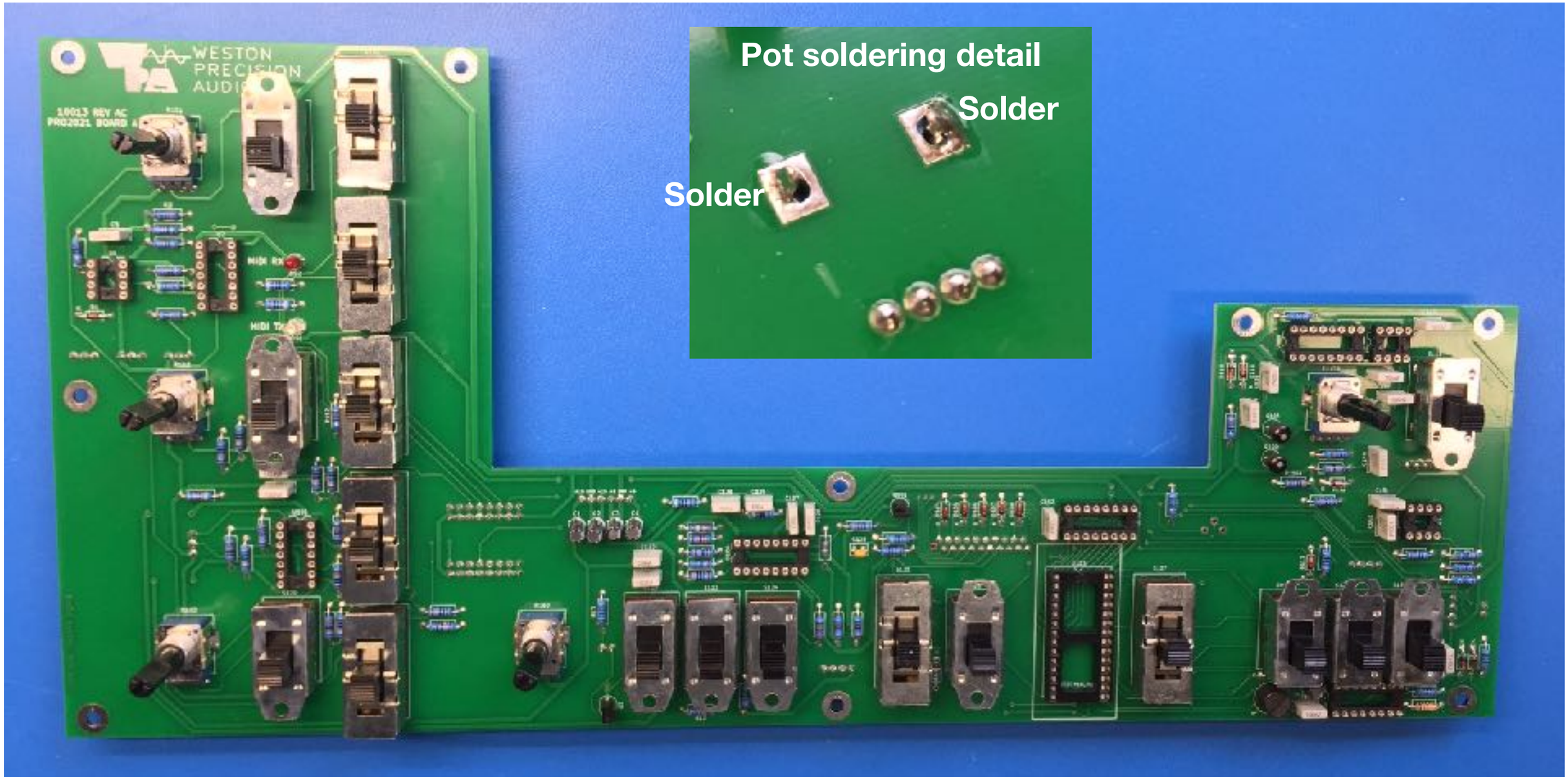
Need to shorten pots

Build PCB “A”

1. Solder all the switches onto the board as shown. The footprint will make it obvious which type of switch goes on. Just make sure to orient the switches with 1 tab removed as shown in the picture. I you just removed all the tabs, the orientation doesn’t matter
2. Solder all the potentiometers onto the board. Note the location of each 100k Linear (marked 100kB) and 100k Audio (100kA). Be sure to put some solder on the retaining tabs of the pots for mechanical strength as shown in the detail view.
3. Solder the trimmer potentiometer onto the back side of the board.

Table 1

R101 R102 R103	100k AUDIO Potentiometer
R1139 R160	100k LINEAR Potentiometer
TP1181	1k Trimmer Potentiometer (Back Side Of Board)



Build PCB “B”

1. Building PCB “B” will proceed the same was as PCB “A”. Start with the resistors on the front side of the board.
2. The 2 sets of 6 resistors marked as needing to be matched within .01% of each other may be installed on the thru hole pad OR the surface mount pads, NOT BOTH.

PCB “B” Resistors

Location	Value
R1109	7.5K
R1110	90.9K
R1114	56K
R1115	3.9M
R1118 R1141 R1145	68K
R1121	4.7K
R1122	10K
R1124	2.7K
R1125 R1126 R1169	1K
R1128	27K
R1140 R1142	20K
R1143	3.9K
R1147	150K
R1144 R1149 R1150 R1151	240K
R1152	220K
R1155 R1212	1.5K
R1153 R1158 R1161	91K
R1162	51K
R1163	3K
R1170	39K
R1173	75K
R1174	2K
R117 R118	8.2K
R1197 R1201	24K
R1196 R1200 R1206 R1208	1.8M
R1198 R1207	2.0M

PCB “B” Resistors, Continued

R1123 R1127 R124 R144	1M	
R127 R145	100K 1%	
R131	1.2M	
R1129 R1130 R1131 R1168 R1172 R141	47K	
R126 R146	4.7M	
R153	300K	
R121 R133 R162 R163 R164 R167	100K .01%-A	These 6 must be matched to each other by 0.01%
R1116 R1120 R1137 R1157 R1175 R1199 R1209 R171 R183	200K	
R1113 R1148 R172	120K	
R142 R155 R173 R174 R175 R176	100K .01%-B	These 6 must be matched to each other by 0.01%
R182	33K	
R1112 R1117 R1119 R1146 R1154 R1156 R1159 R1160 R1171 R125 R147 R161 R166 R169 R170 R178 R179 R180 R181 R184 R185 R186	100K	

Note: these 2 resistors control the amount of the pitch bend. With 8.2k and the Standard PRO2021 bender (10k), you will get a bit less than an octave. I prefer Less range than this and use 15k resistors. You are free to adjust them to customize To your needs. You can also keep the 8.2k resistors in, and run a 1 Meg pot (wired as A rheostat) in Line with the wiper of the bender pot and get an adjustment from max range down to 1/2 step.

Build PCB “B”

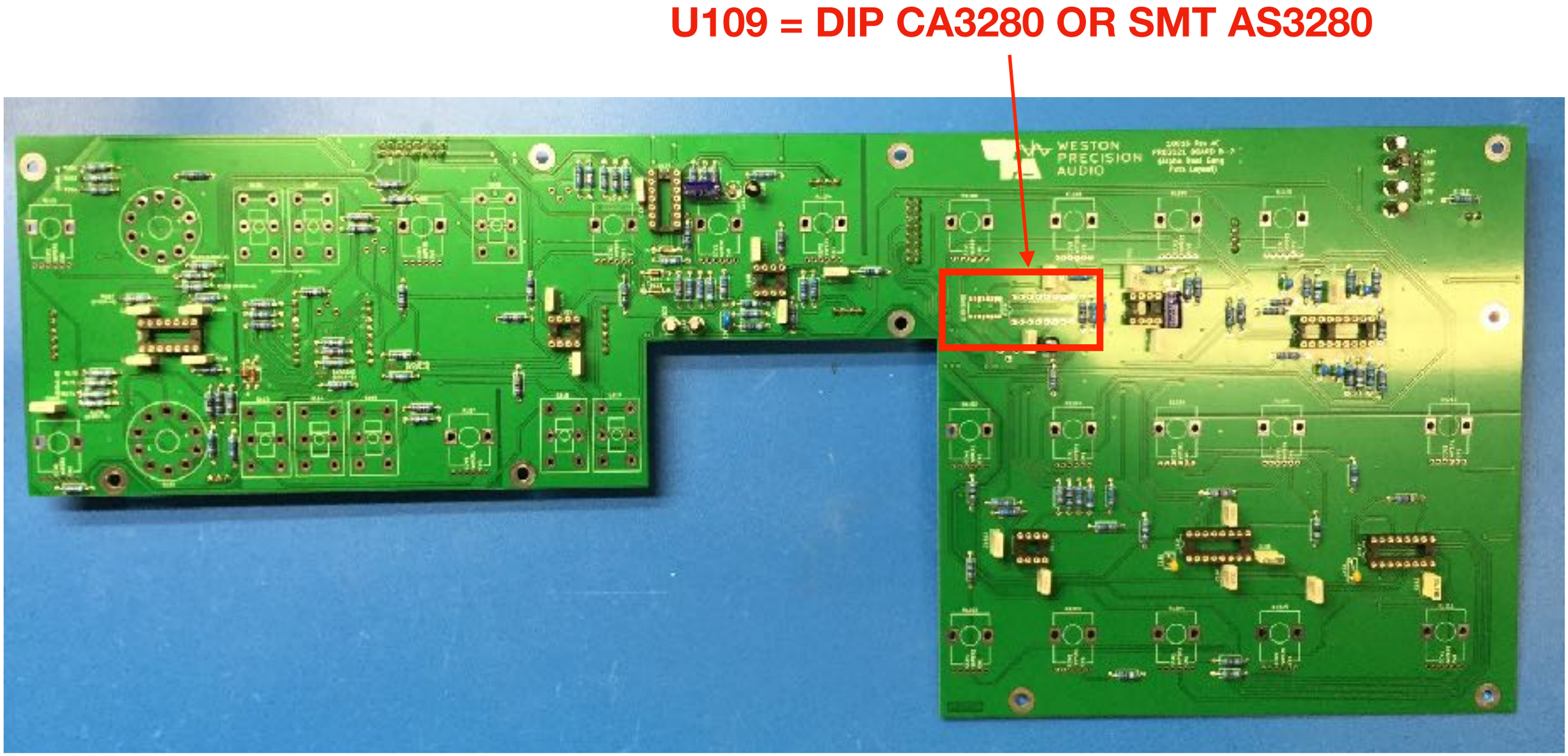
1. Next add the capacitors and diodes to the front of the board.
2. Add the DIP sockets if you are using them (again, highly recommended!)
3. Add the transistors (Q103 and Q104, 2N4250) to the front of the board.

PCB “B” Capacitors and Diodes

Location	Value
C110 C130 C137 C157	0.01u
C116 C117 C118 C119	0.001u
C1 C131 C132 C133 C143 C2 C3 C4	2.2uF 50V
C120 C121 C134 C135 C138 C139 C1402 C1422 C146 C1502 C1512 C159 C160	0.1u
C136	10pF
C144 C145 C147 C148	150 pF
C156	0.039u
C161	0.039u
C158 C162	0.02u
D101 D102 D109 D117 D118	1N914

Build PCB “B”

1. View of PCB “B” with resistors, diodes, caps, and DIP sockets stuffed.
2. Note that U109 should be populated with either a DIP socket / CA3280 OR a surface mount AS3280, NOT both.



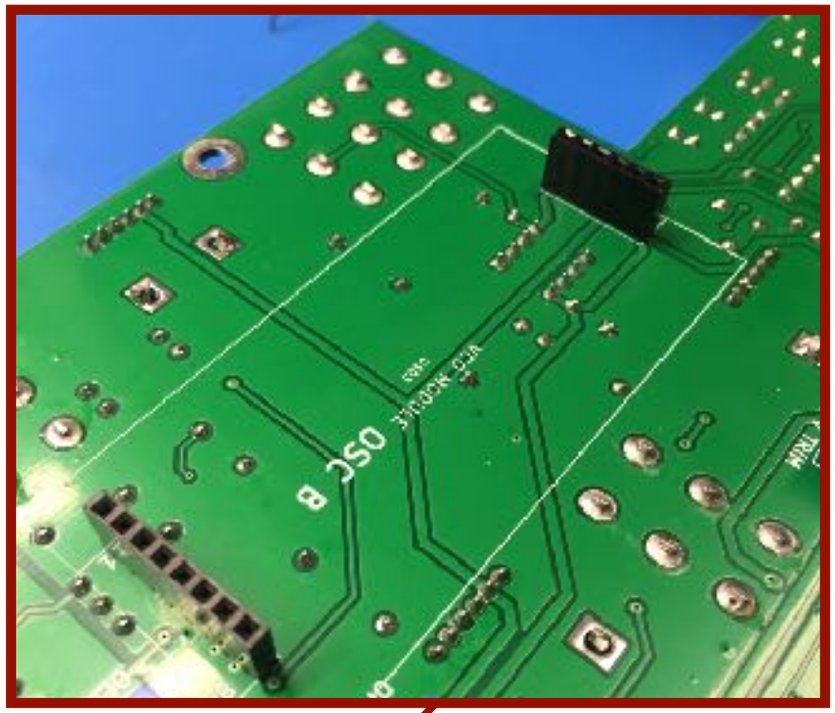
Build PCB “B”

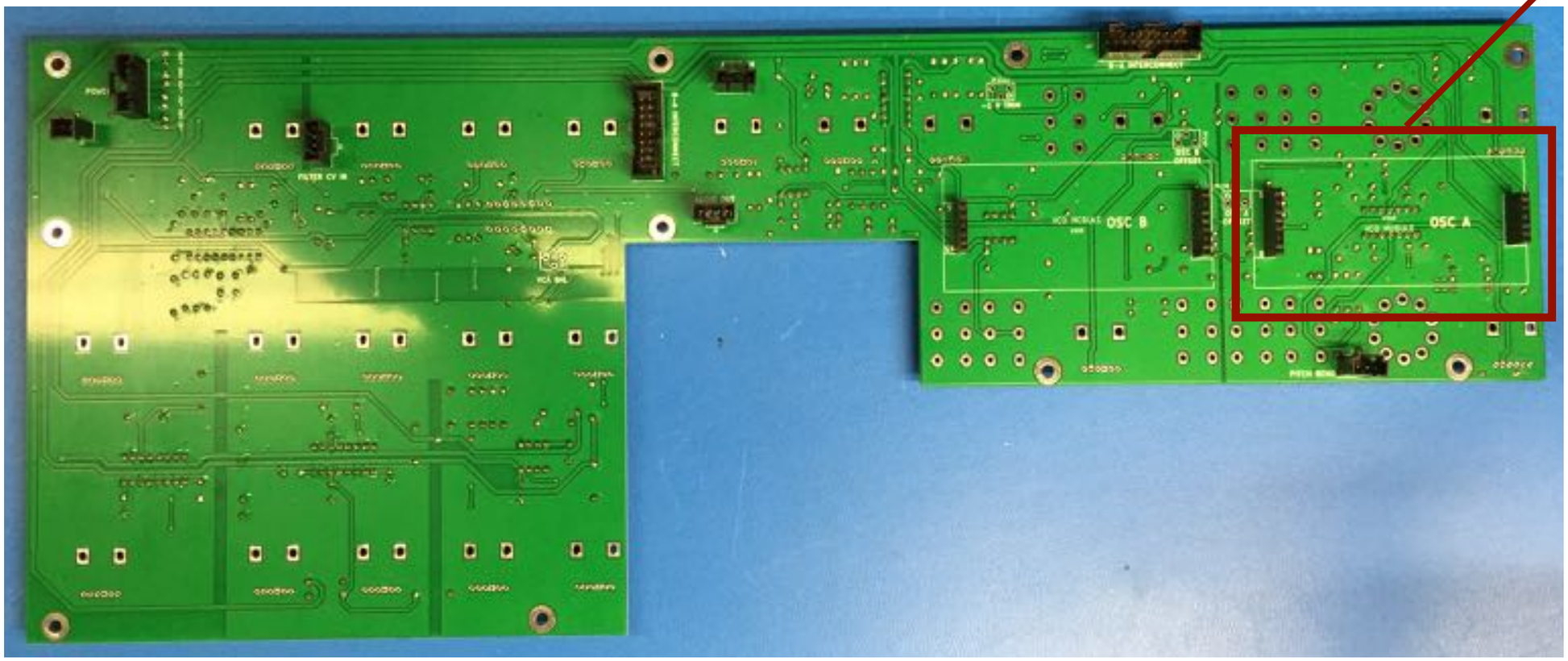
1. Add the connectors to the back of the board according to the table below
2. Also add the 6 and 8 pin header receptacles to the VCO card areas.

At this point, if you are going to do a flux clean on the board, do so with the appropriate process for your solder.

PCB “B” Connectors

J12	1	PITCH WHEEL 10K	Molex 3 Pin Male Header
J16	1	B-A Interconnect RIGHT	Tyco 16 Pin Ribbon Header
J17	1	B-A Interconnect LEFT	Tyco 16 Pin Ribbon Header
J18	1	Power	Molex 6 Pin Male Header
J6	1	FILTER CV IN	Molex 4 Pin Male Header
J7	1	AUDIO IN	Molex 4 Pin Male Header
J8	1	AUDIO OUT	Molex 4 Pin Male Header
J9	1	PWR LED	Molex 2 Pin Male Header





Build PCB “B”

1. Solder the switches. The B board only has DPDT switches and it will be obvious where they go.
2. Mark the “1” pin of the 2 rotary switches with a marker and make sure it is aligned properly and add the 2 rotary switches to the front of the board.
3. Solder the pots per the table shown. Again, the pots should have all been trimmed per page 24 at this point if they need to be.
4. Add the trim pots to the BACK of the board according to the table below.

PCB “B” Potentiometers

R1132 R1133 R1134 R1211	100K AUDIO
R1188 R1189 R119 R1190 R1191 R1192 R1193 R1194 R1195 R1202 R1203 R1204 R1205 R1210 R139 R165 R187	100K LINEAR

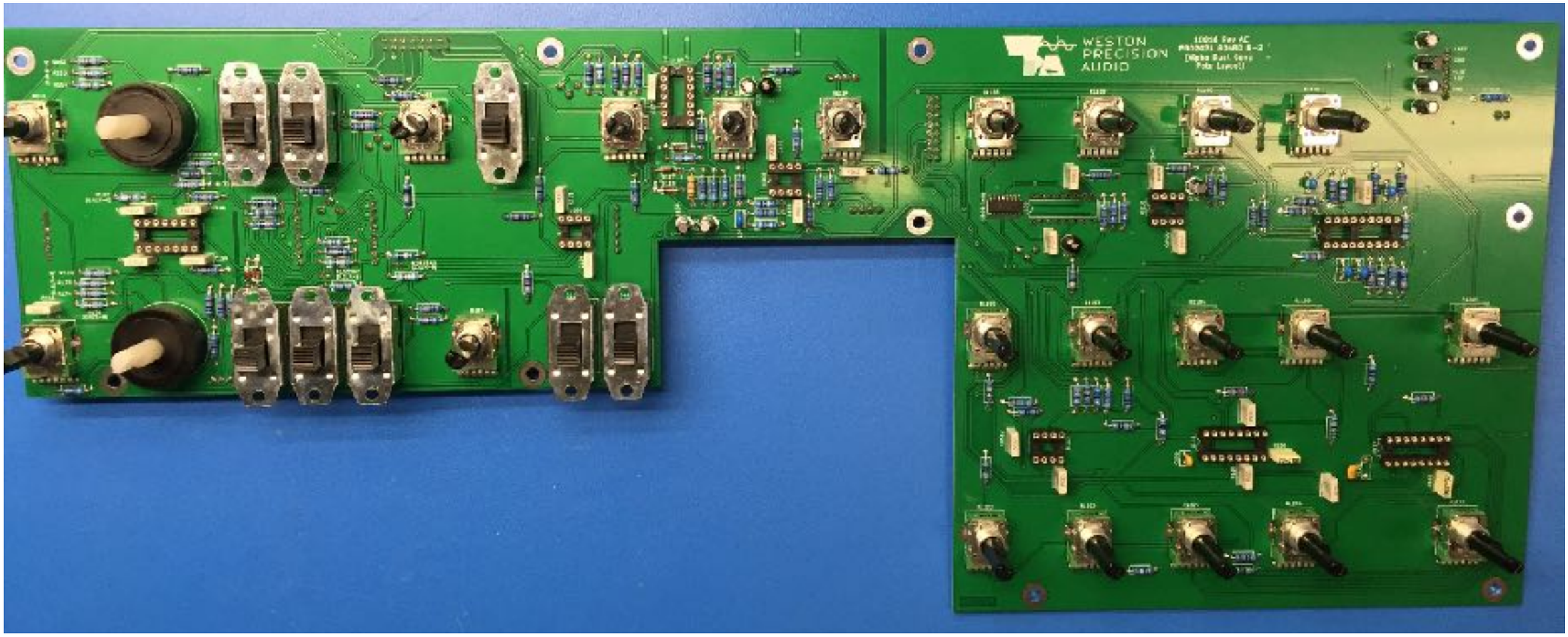
PCB “B” Trim Pots

TP1111	20K
TP1138 TP120 TP140	100K

Marking the “1” pin of the rotary switches:

Build PCB “B”

1. PCB “B” Completed



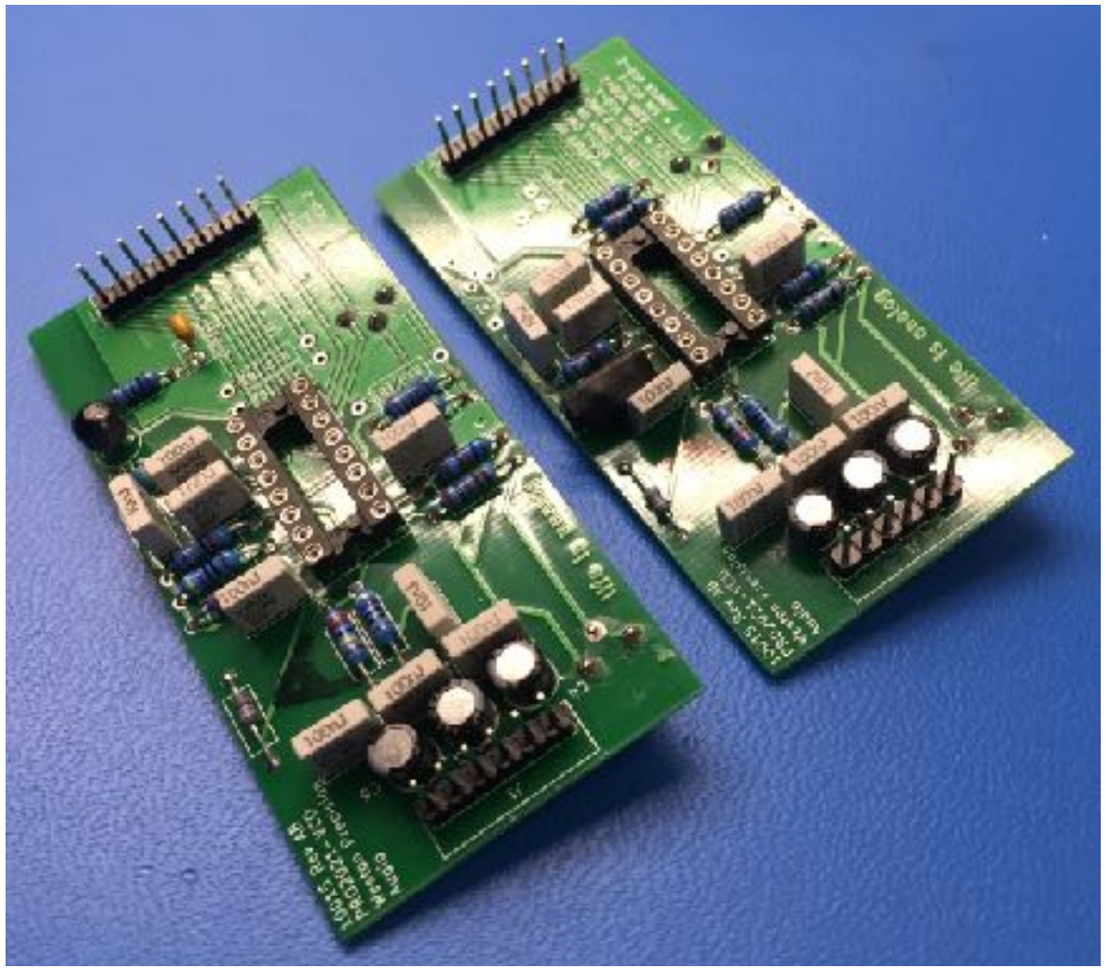
Build VCO PCB Cards

1. Assemble the VCO cards per the table below. Note that they are stuffed slightly different for VCO 1 vs VCO 2. The table and the silkscreen on the boards indicate the differences. Write with a sharpie on each board which one is 1 and 2 to keep them straight. All components EXCEPT the trim pots go on the FRONT of the PCBs. Trimpots go on the back.

VCO PCBs Components

Location	Component	Note
C1 C12 C2 C3 C5 C9	0.1u	Rectangular film cap
C10	200pF (OMIT FOR VCO 2)	MLCC
C11	1000pF	Rectangular film cap
C13 C4 C6	10u	Electrolytic radial
C7 C8	0.01u	Rectangular film cap
J1	Power	Male pin header
J2	Signals	Male pin header
Q1	2N4250 (OMIT FOR VCO 2)	
R1	26.7K 1%	
R10	10K (OMIT FOR VCO 1)	
R11	100K (OMIT FOR VCO 1)	
R12	560K (OMIT FOR VCO 1)	
R13	47K (OMIT FOR 2)	
R14 R16	10K (OMIT FOR 2)	
R2	5.62K 1%	
R20	1.82K 1%	
R3	270K (VCO 1) / 820K (VCO 2)	
R4 R7	470 Ohm	
R5	820K	
R6	2.2M 1%	
R8	10K (JUMPER FOR VCO 2)	
R15 R9	20K (OMIT FOR VCO 2)	
TP1	5K TP	
TP2	25K TP	
U102	CEM3340(VCO)	

VCO PCBs completed



Prepare PCBs

1. Now that the PCBs are all stuffed, if you have installed DIP sockets for the ICs, install the ICs on all the boards. The 2 VCO cards simply get a CEM3340 chip. The “A” and “B” boards are as follows in the tables below.

PCB “A” ICs

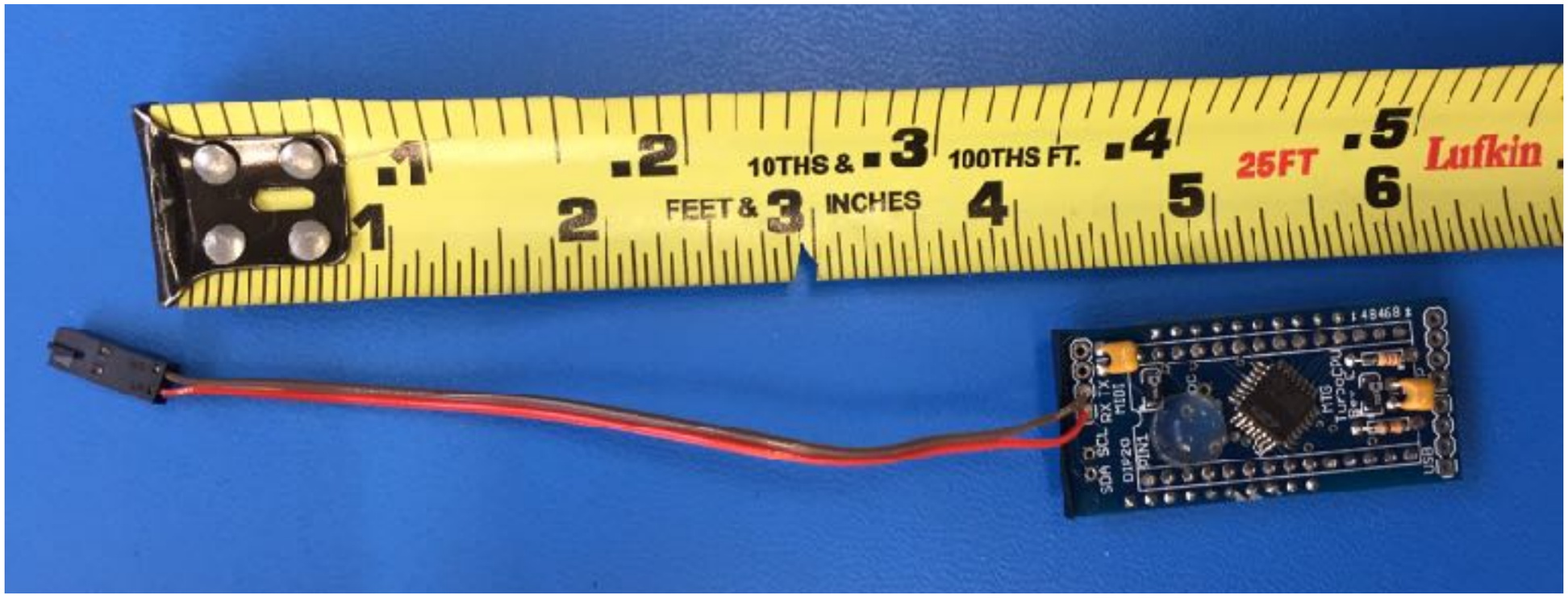
U1	6N137
U101	LM348
U106	CEM3340
U109	LM13700
U110	TL081
U112	AD558JN
U113	TurboCPU
U114	4049
U115	MC1458
U2	7414

PCB “B” ICs

U102 U103	PRO2021-VCO Card
U104 U107	LM348
U105	TL082
U108	NE5532
U109	CA3820 / AS3280
U110	TL081
U111	AS3320
U115	MC1458
U116 U117	AS3310

Prepare PCBs

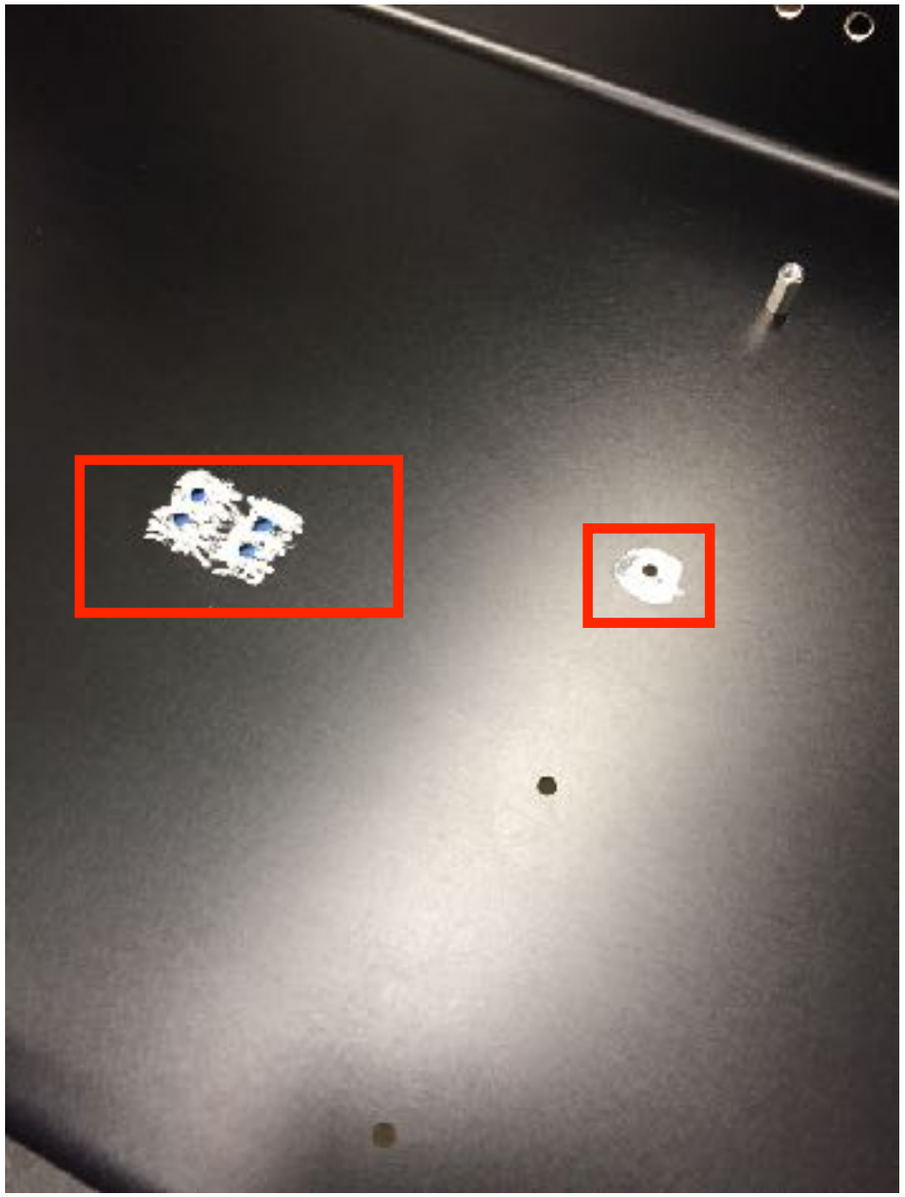
1. Prepare a 2 wire pigtail with molex 2 pin connector as shown and solder it to the Turbo CPU for the MIDI Connection. RX = Pin 1, TX = Pin 2.
2. Install the Turbo CPU into it’s socket and connect the wire from step (1) to the connector J15 on board “A”



Lower chassis wiring

1. On the INSIDE of the chassis bottom (the one without all the holes for the pots/switches), grind or sand away the anodizing on the small pattern of 4 holes in the middle, and 1 of the 4 holes in the upper right corner for the power supply. This is for grounding purposes and is very important.
2. On the INSIDE of the chassis top, grind away the anodizing on the 2 holes on the lower middle of the panel.

Chassis bottom grounding



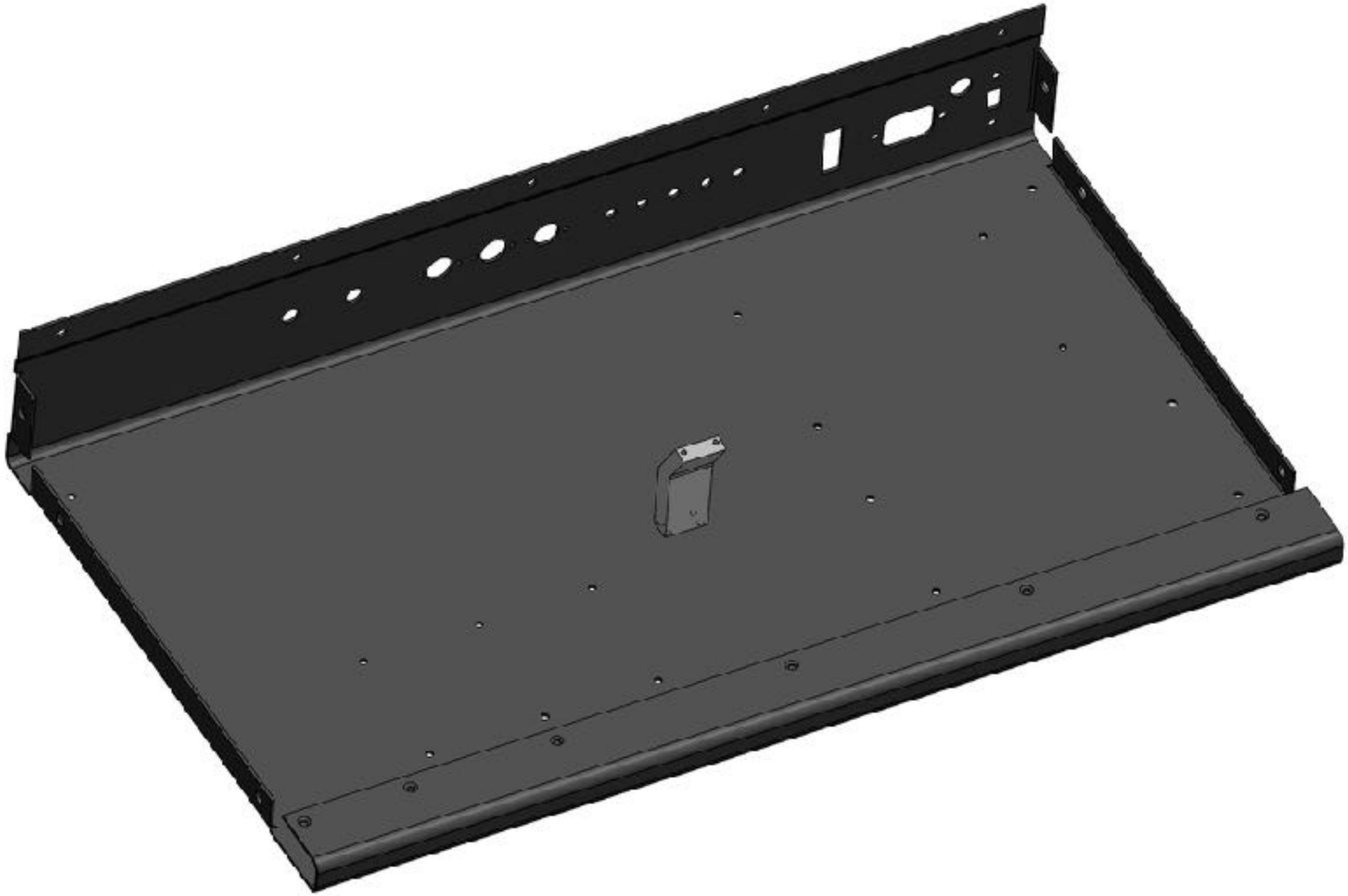
Chassis top grounding

Lower chassis wiring

1. Install the 4 standoffs for the power PCB as shown. They are a 1/2” long standoff from **BAG B** and a 3/16” long pan head screw from **BAG F**.

Lower chassis wiring

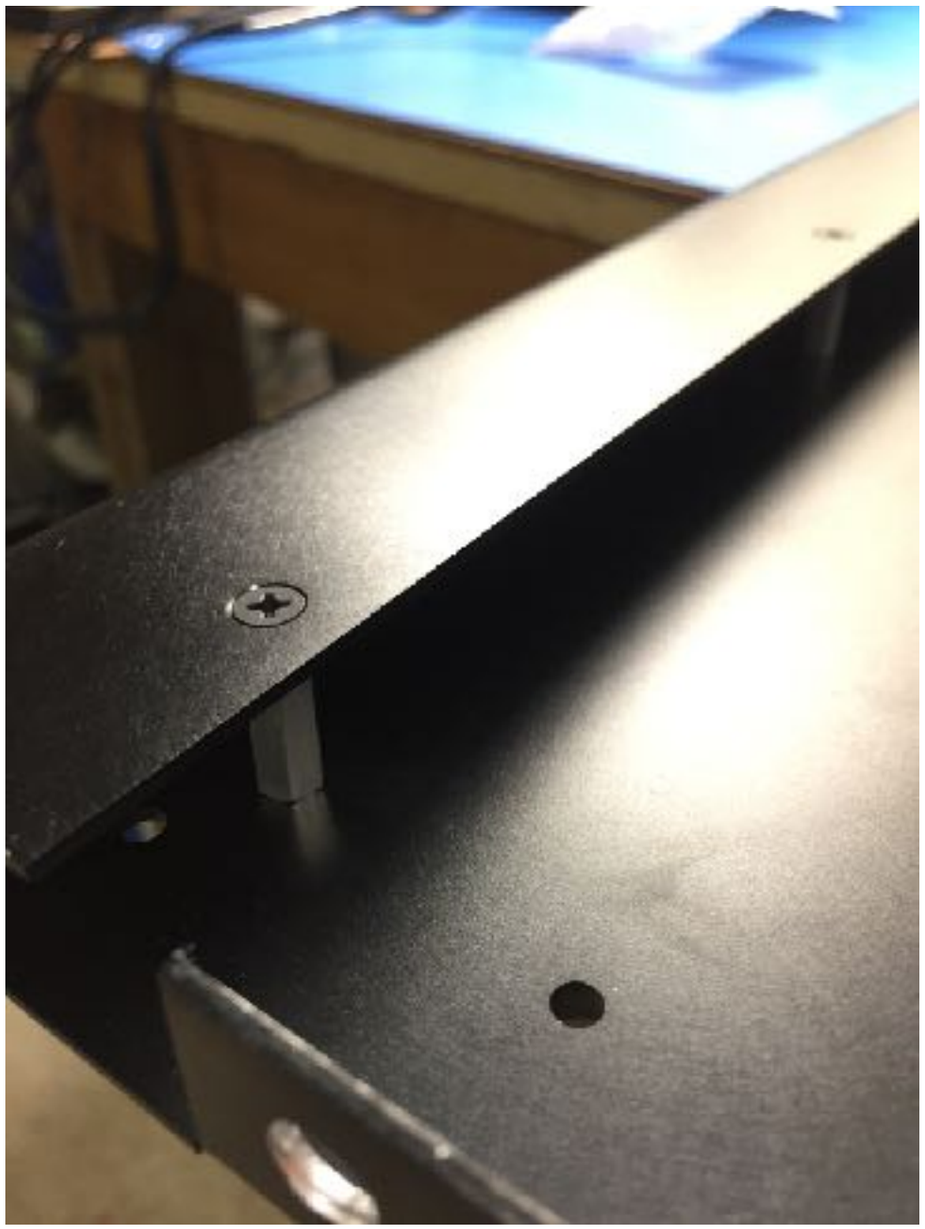
1. Install the Chassis Brace in the orientation shown using 4 3/16” long pan head screws from **BAG F**. Leave all 4 screws a tiny bit loose so it can slip around in plane for now.



Lower chassis wiring

1. Install and tighten the 4 3/4” long standoffs from **BAG B** with 8 flat head 1/4” long black screws from **BAG C** (4 from the top and 4 from the bottom) to the chassis bottom. This adds rigidity to the chassis.

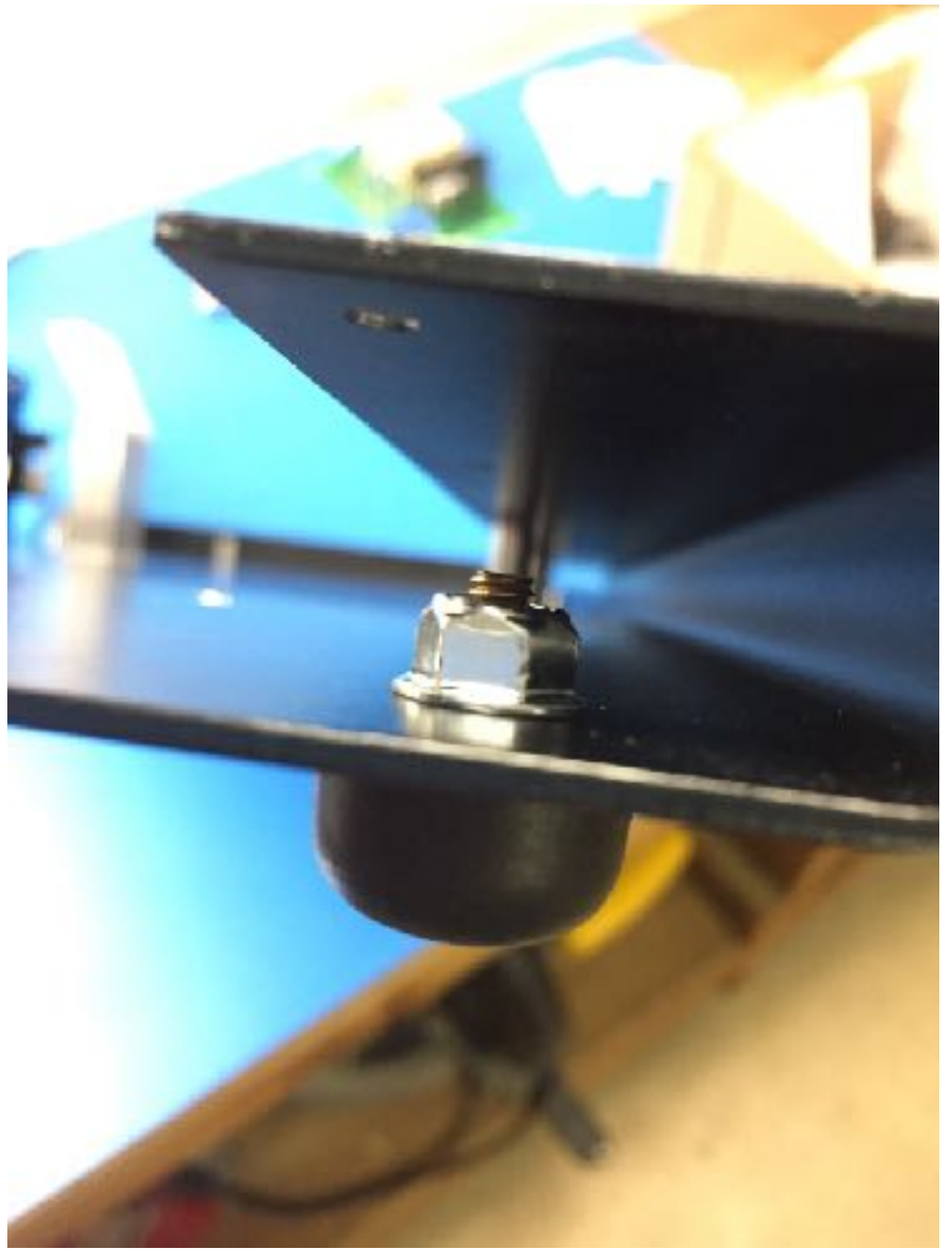
Top side

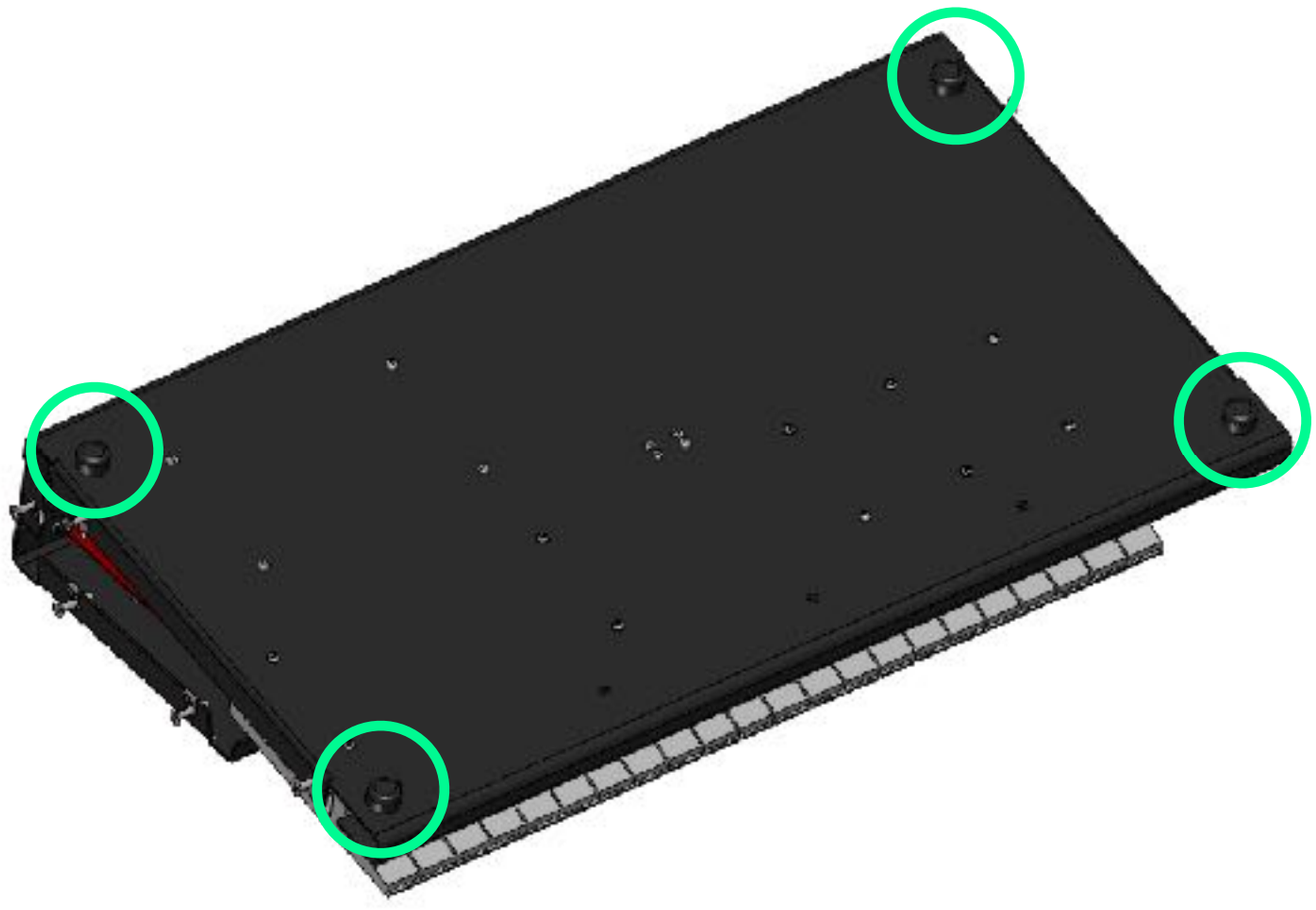


Bottom side

Lower chassis wiring

1. Install the 4 feet with locknuts from **BAG I** to the chassis bottom. The go on the 4 far corners and are installed as shown. It's easiest to get the started then tighten them by holding the rubber with your hands and tightening the lock nut with a combination wrench.





Lower chassis wiring

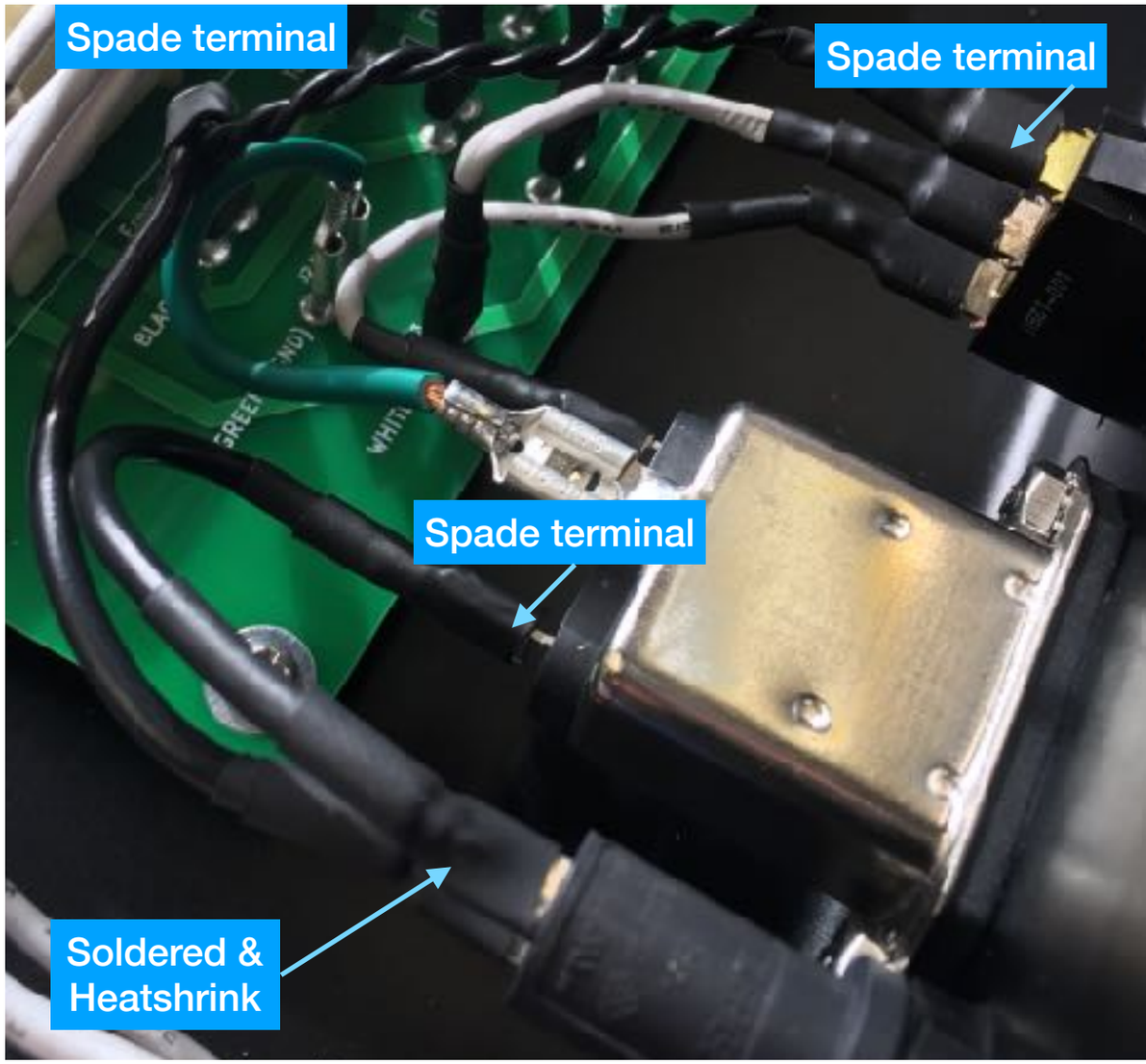
1. Install the completed Power PCB on the standoffs you added and secure the board as shown with 4 3/16” long pan head screws from **BAG F.**

Lower chassis wiring

1. Pop the power switch into the lower chassis, **gold terminal facing up.**
2. Install the Power receptacle into the lower chassis with 2 3/8” long black flat head screws and 2 6-32 kep nuts from **BAG D.**

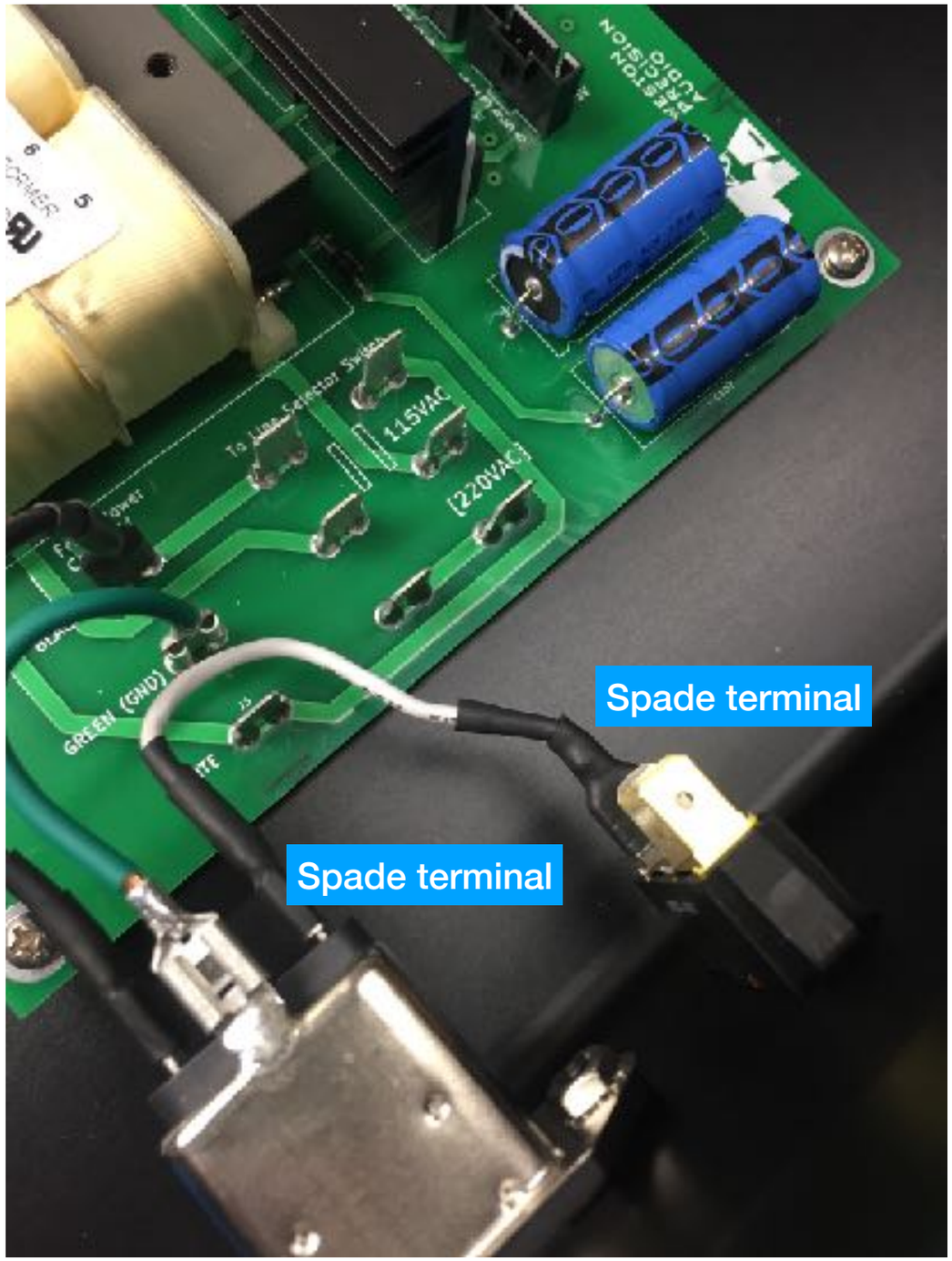
Lower chassis wiring

1. Install the fuse holder into the lower chassis as shown.
2. Run 14 gage black wires as shown: 1 from 1 side of the power receptacle to the fuse, and 1 from the fuse to the “Black” terminal on the power supply. Run a 18 gage black wire from the same spade terminal at the “black” label on the PSB to the to (gold) terminal on the power switch. **Use 1/4” spade female terminals to connect to all of the terminals in the PSU board and to the power receptacle and power switch. You can get these at most home stores, etc…**



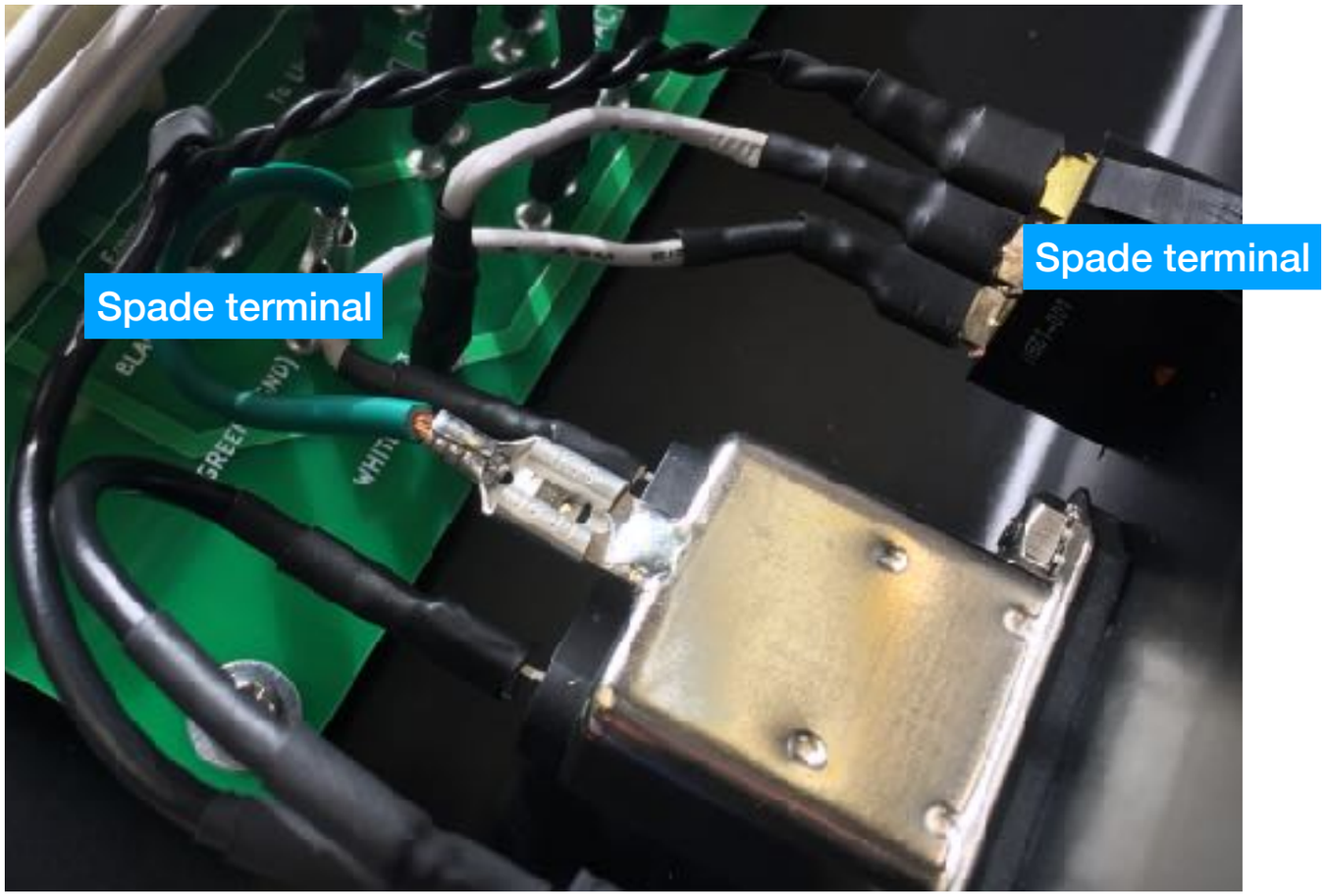
Lower chassis wiring

1. Run a 14 gage green wire from the ground terminal of the receptacle to the ground terminal on the PSU PCB.
2. Run a 14 gage white wire from other terminal of the power receptacle to the lower terminal of the power switch.



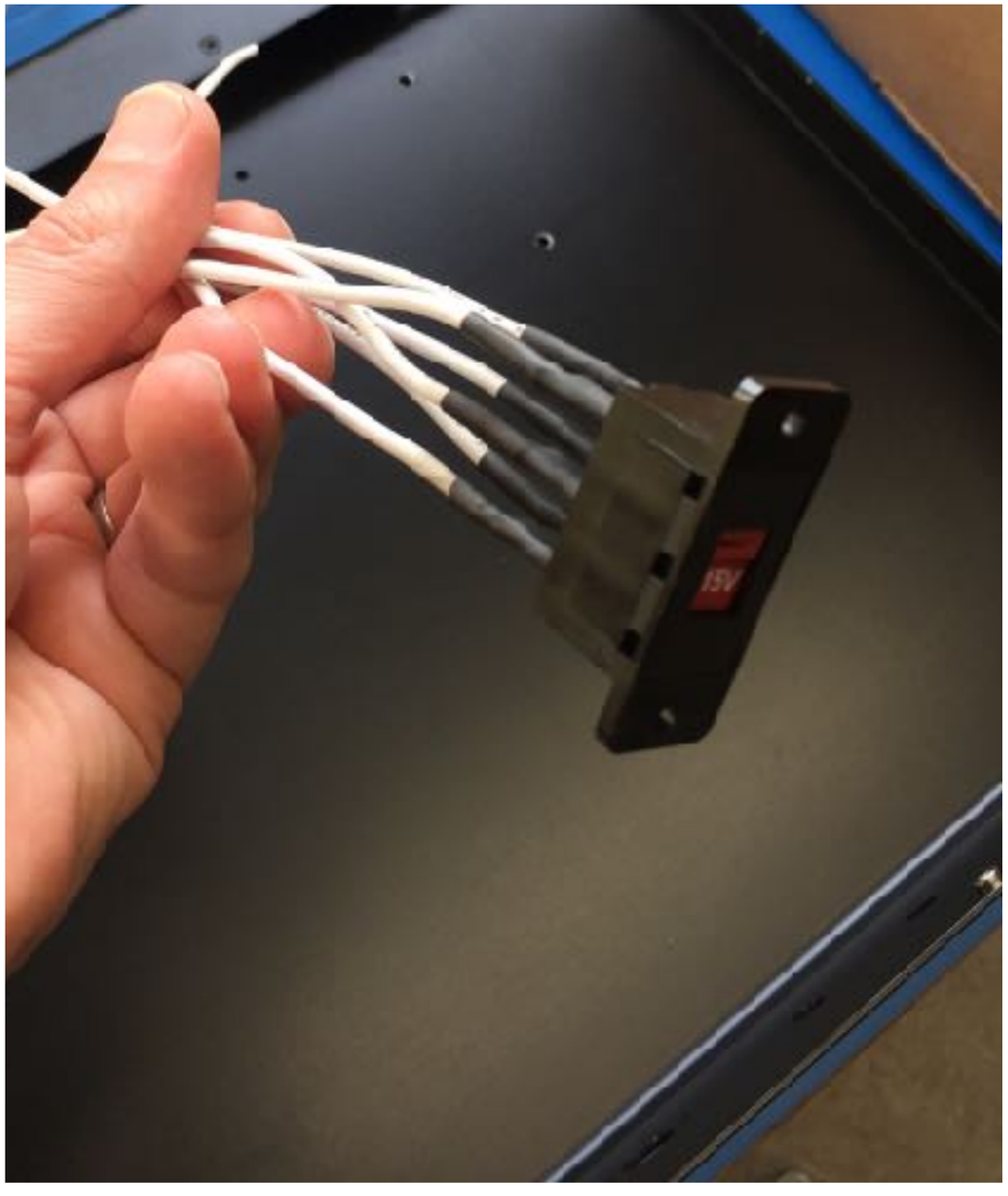
Lower chassis wiring

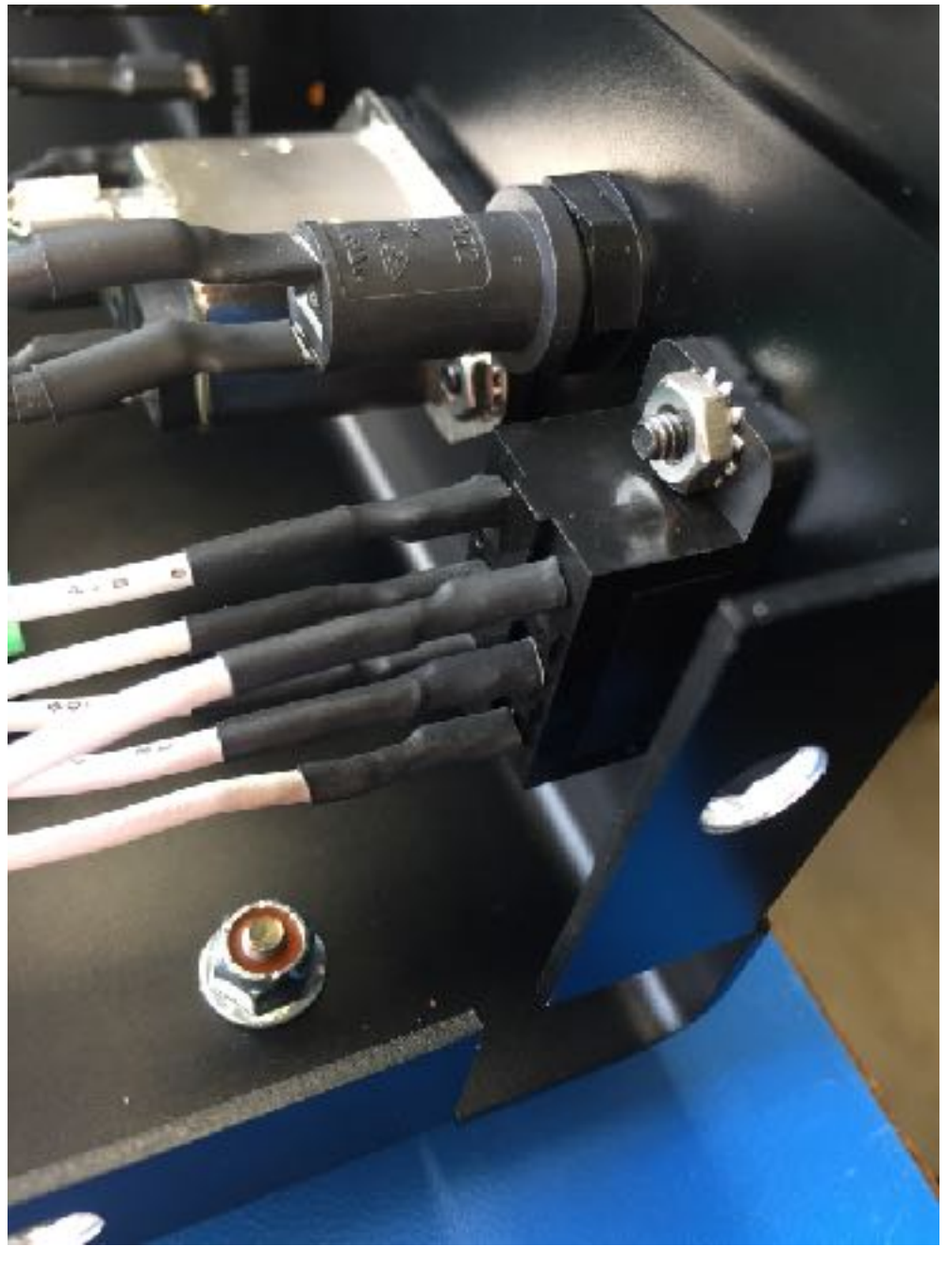
1. Run a 14 gage white wire from the remaining (middle) terminal of the power switch to the “white” labeled terminal on the PSU board.



Lower chassis wiring

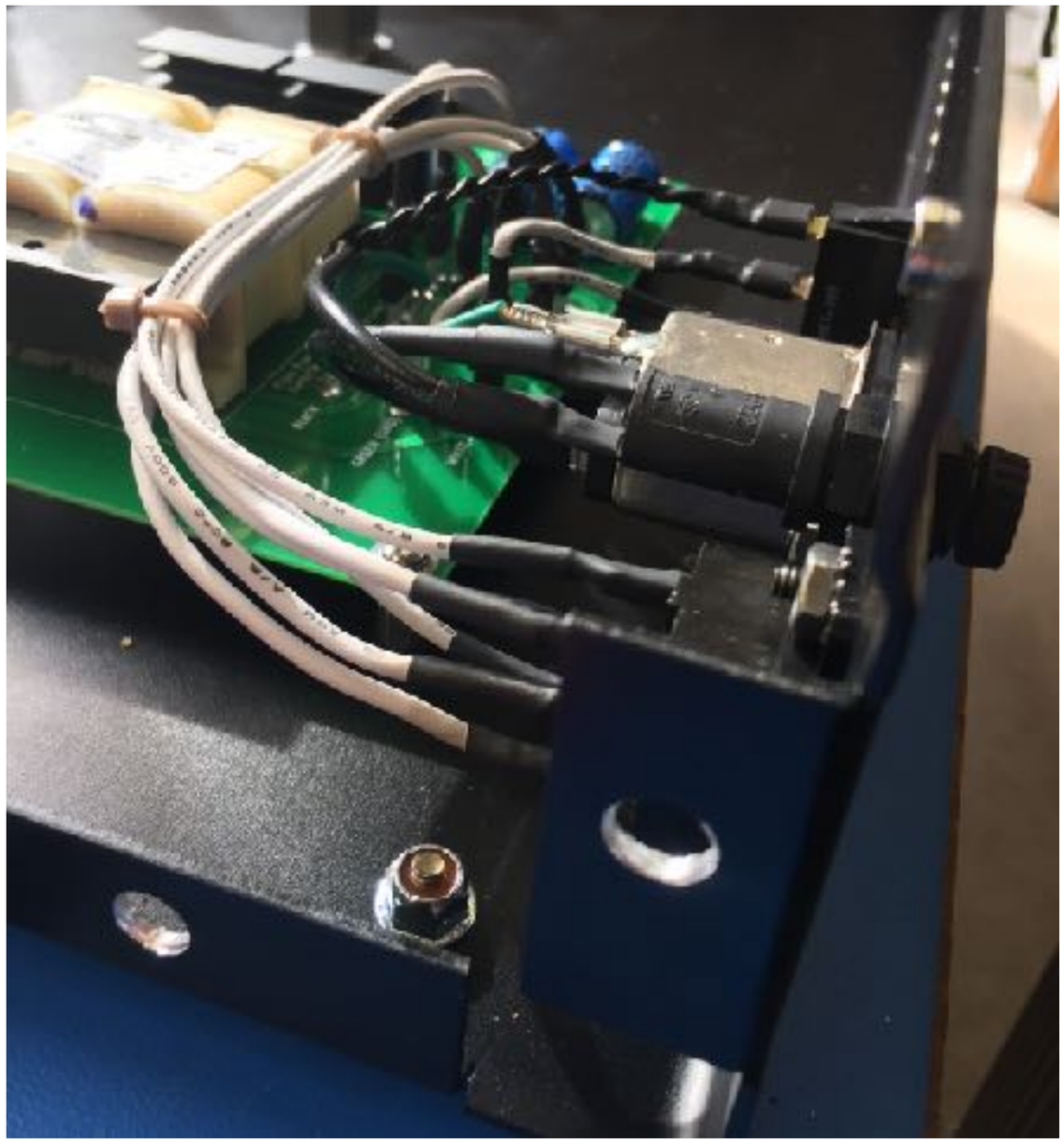
1. Solder about 10” of 14 gage white wire and heat shrink to all 6 terminals on the line voltage selector switch as shown.
2. Mount the selector switch with the text facing up into the chassis from the back using the remaining 2 nuts and screws from **BAG D**.





Lower chassis wiring

1. Clip the 6 wires from the line selector to the appropriate length and terminate each with a 1/4” spade receptacle and connect them to the PSU board as shown. The image of a DPDT switch on the silk screen on the board indicates clearly which wire should go to which terminal.
2. Bundle up the white wires with zip ties and clean them up as shown.



Lower chassis wiring

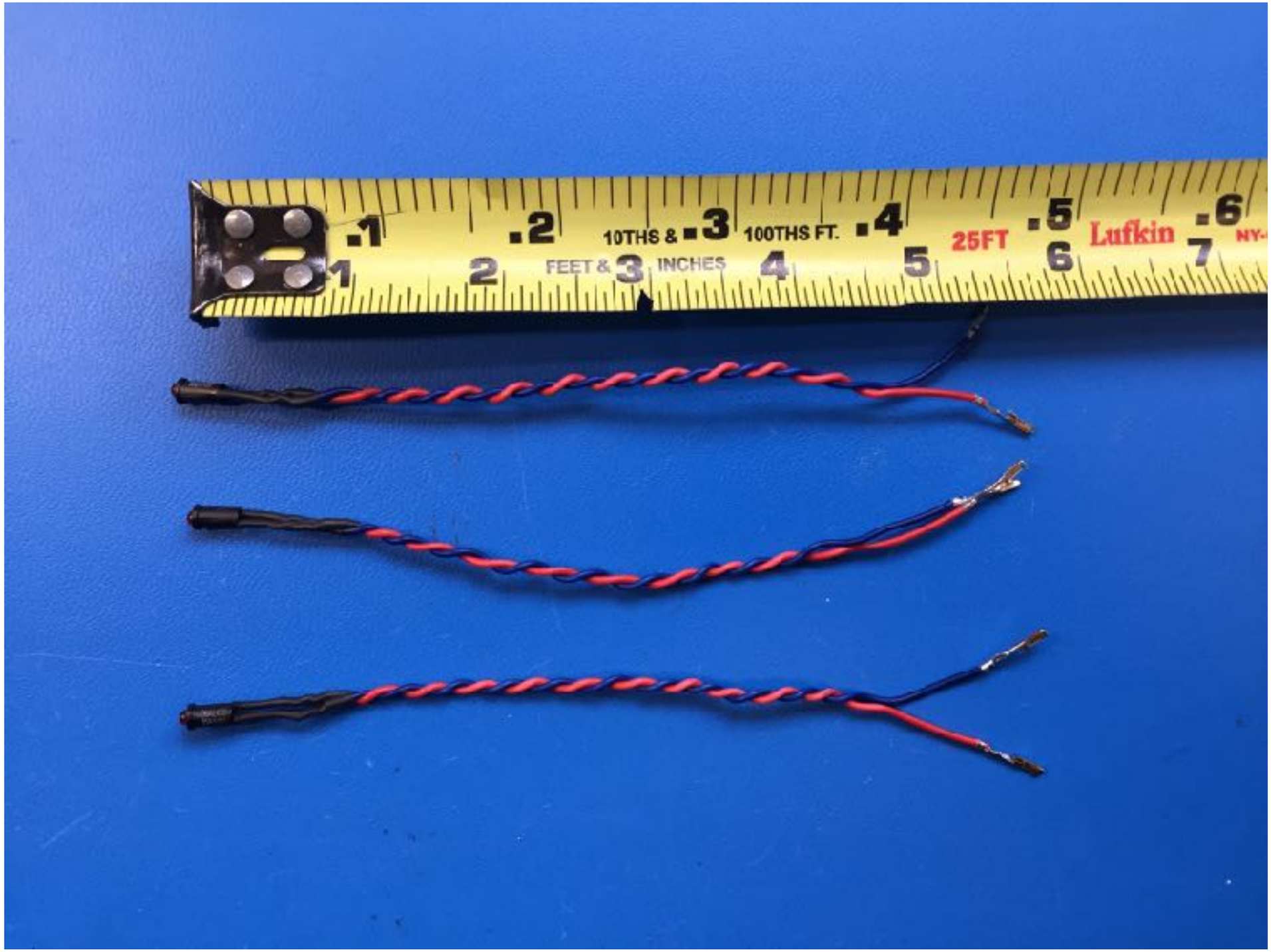
1. Install the Fatar 37 key keybed to the lower chassis using the 10 “plastite” self tapping screws from BAG G. The torx bit required for these screws comes in BAG G as well. Do not over tighten these screws, as the base material of the keyed is molded plastic! You just need to get them snug.

Upper chassis wiring

1. Attach all 18 of the PCB mounting standoffs (1/2” long) from **BAG B** to the chassis top and tighten as shown with screws from **BAG C**.

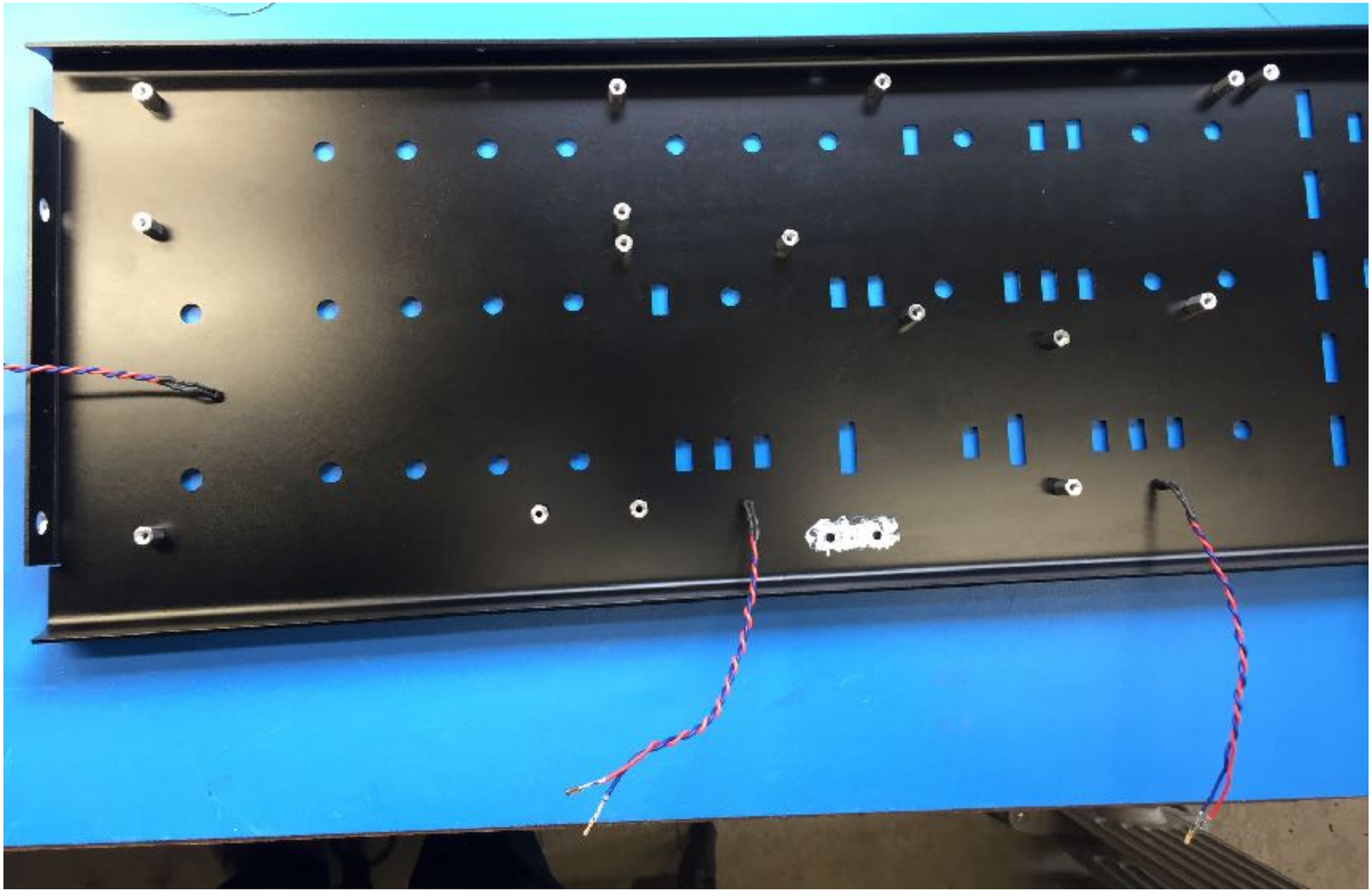
Upper chassis wiring

1. Prepare your LED pigtails by soldering and heat shrinking 22-26 gage wire to each one at the length shown. Note your wire colors (cathode and anode) and twist the wires together and terminate each one with a Molex pin. DO NOT INSERT THE PINS INTO THE HOUSINGS ON THESE YET.



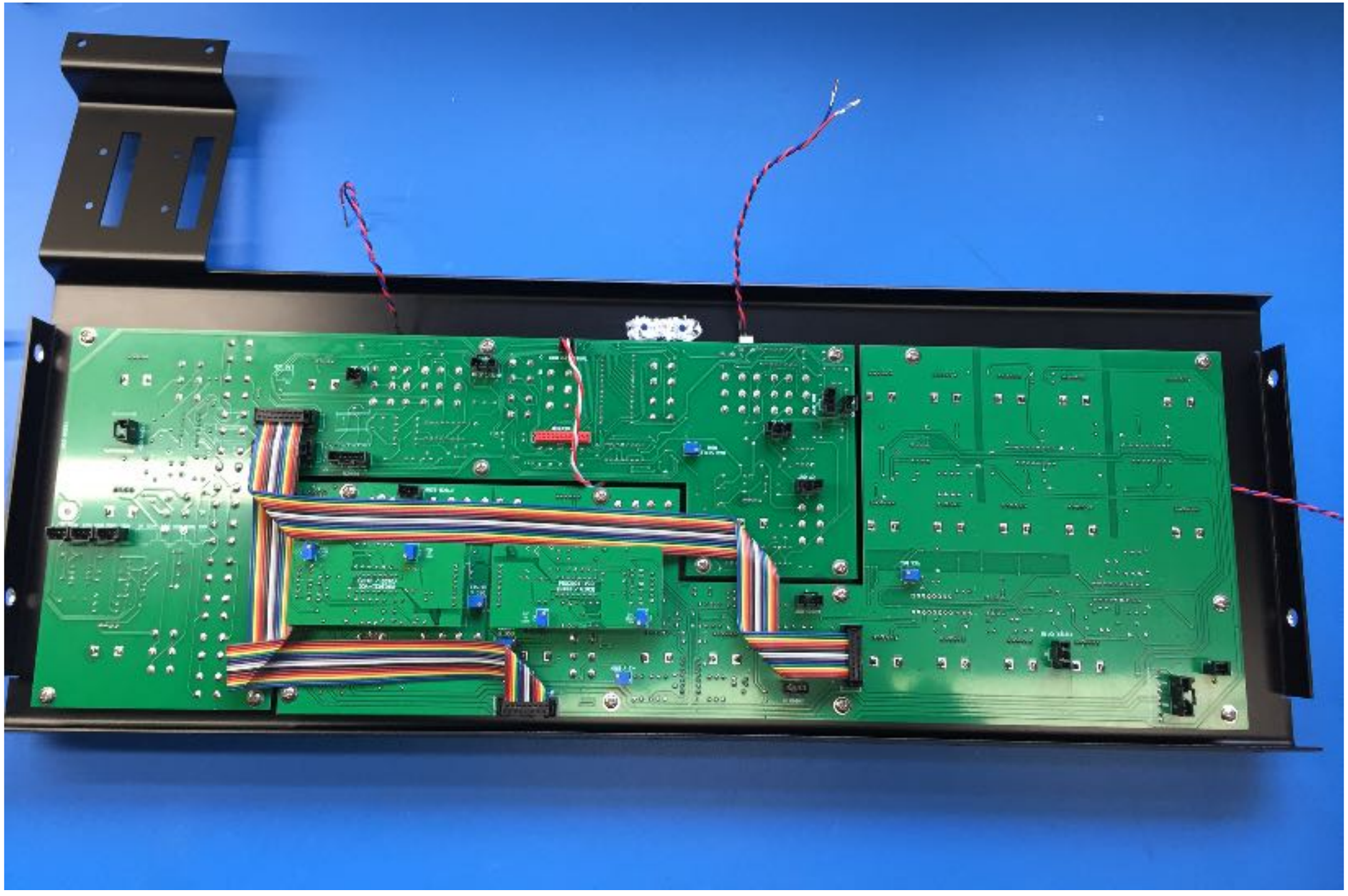
Upper chassis wiring

1. Insert each LED pigtail into their holes from the front as shown, all the way flush to the sheet metal.
2. You can now insert the molex pins into their 2 pin housings.



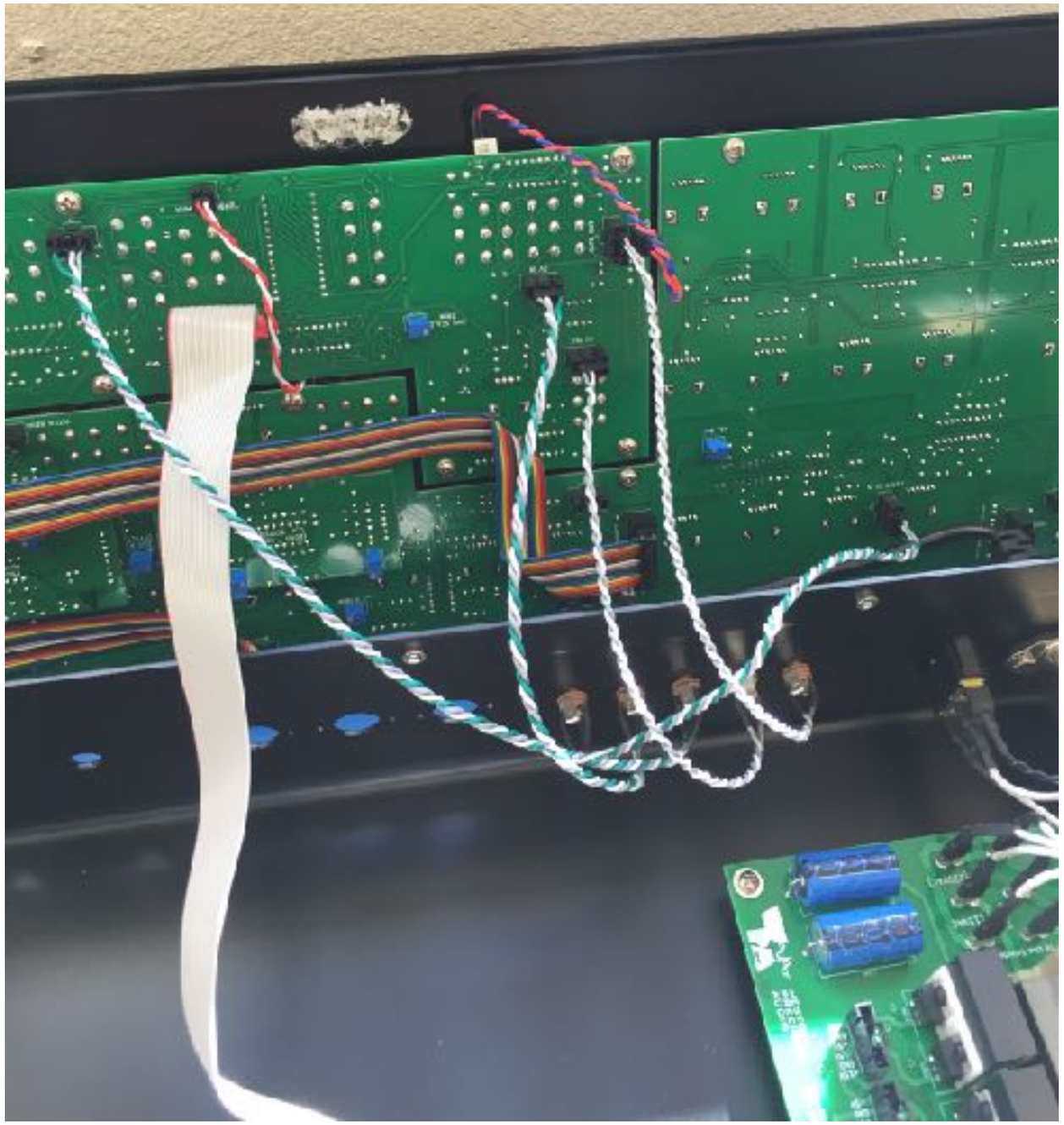
Upper chassis wiring

1. Mount the PCBs into the chassis top with 3/16” long pan head screws from **BAG F.**
2. Run the included A-B flat cables as shown, folding them to keep the run neat and clear all the trim pots as shown. Note the double-back fold done on the lower left (shorter) ribbon cable.

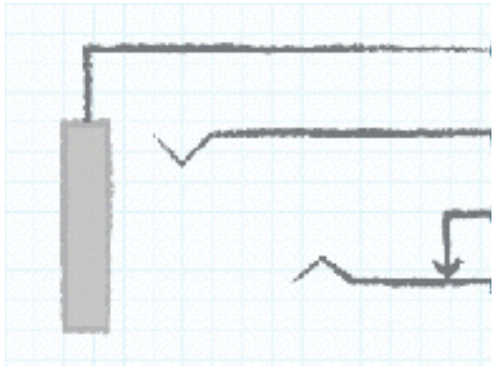


I/O wiring

1. Set the chassis bottom on your work surface and prop the chassis top up against a wall or other object so it is very close (~2-4”) from the chassis bottom to being wiring.
2. Run twisted wire pigtails from each of the 1/8” jacks on the lower chassis at appropriate lengths as shown and terminate with molex connectors according to the chart shown. **All of the CV/gate ins and outs will use 4 pin molex connectors.**



Jack schematic



Sleeve

Ring

NC (normal close) Tip Switch

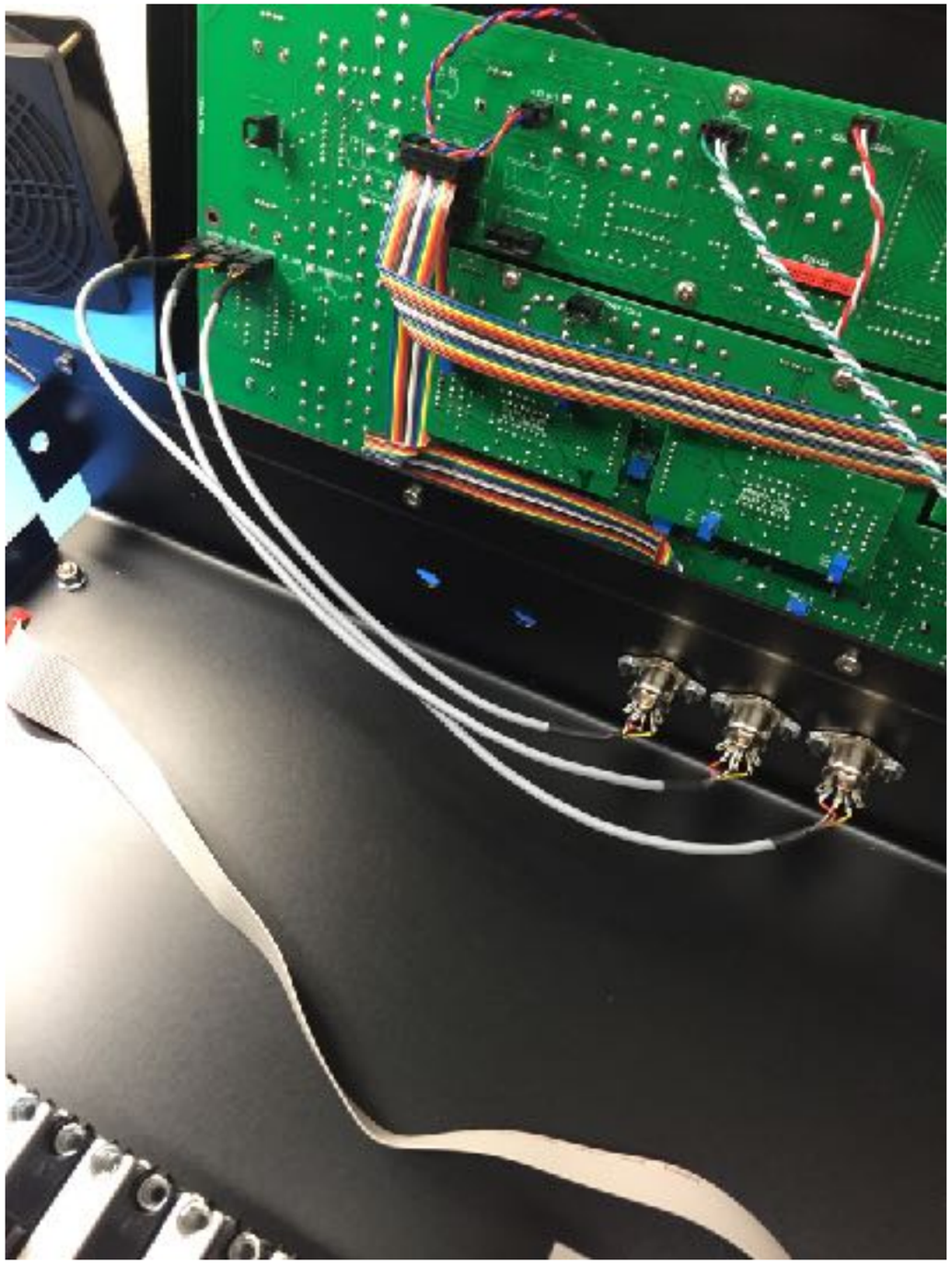
Tip

Gate/CV Ins and Outs wiring

Connector	Jack Connection	Molex Pin Number
GATE OUT	Tip	2
	Sleeve	1
GATE IN	Tip	2
	Sleeve	1
	NC Sleeve Switch	4
CV OUT	Tip	2
	Sleeve	1
CV IN	Tip	2
	Sleeve	1
	NC Sleeve Switch	4
FILTER CV IN	Tip	2
	Sleeve	1
	NC Sleeve Switch	4

I/O wiring

1. Make MIDI in, out, and thru pigtails in a similar fashion to the previous step, **but using the included 4 wire shielded cable.** According to the table below. The MIDI connectors have the pin numbers conveniently printed on the back of them.

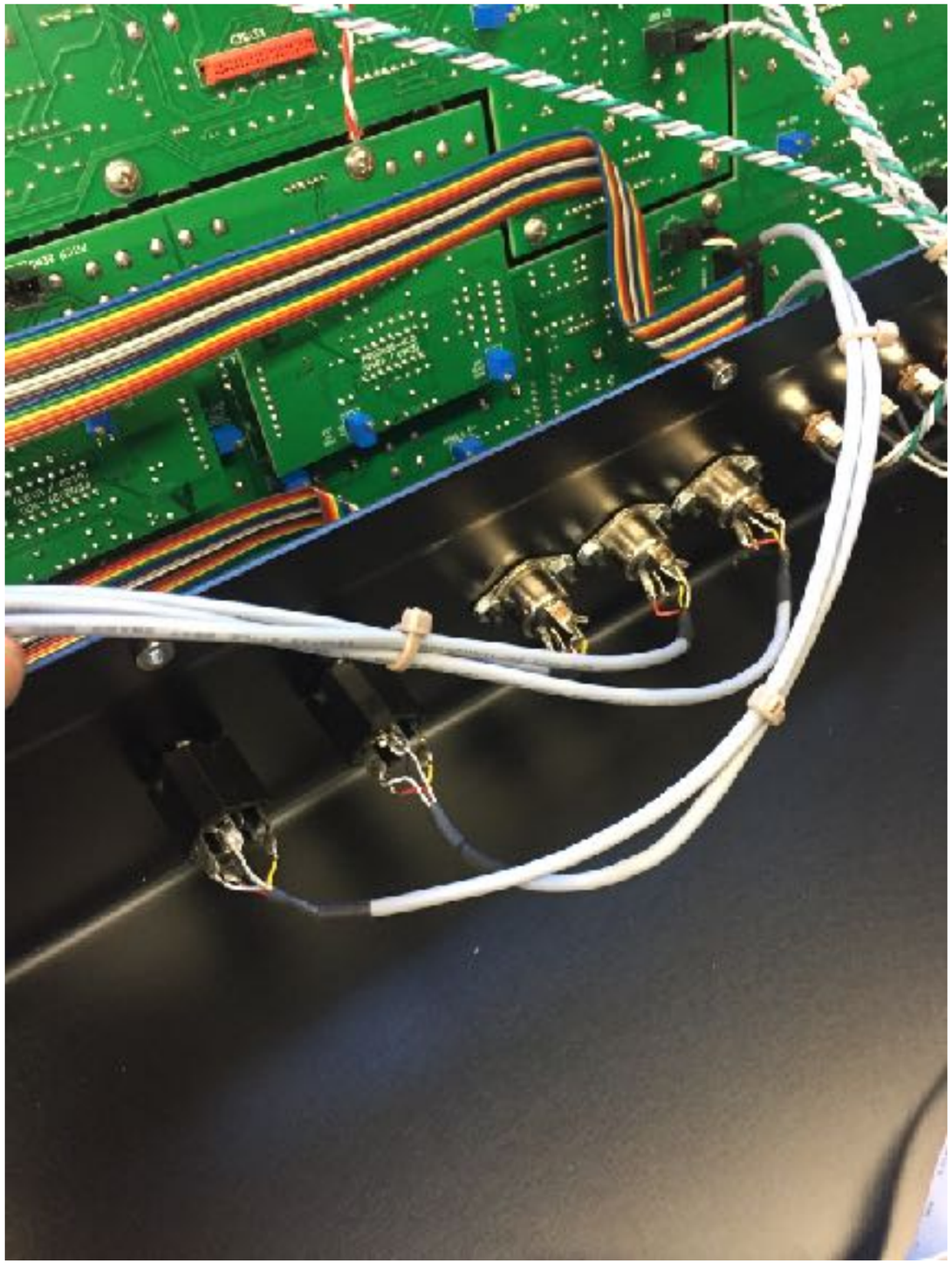


MIDI harness wiring

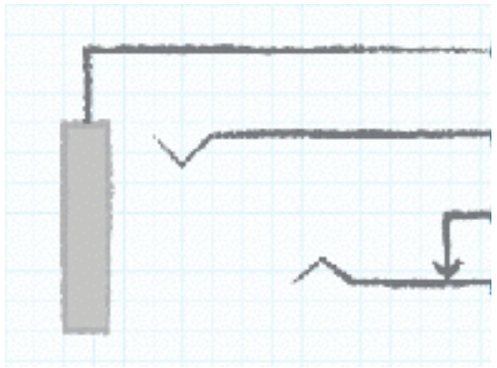
Connector	Cable Length	Jack Connection	Wire Color	Molex Pin Number
MIDI IN	15”	Pin 2	Black	N/C
		Pin 4	Red	2
		Pin 5	Yellow	3
		Body tab	Shield	N/C
MIDI IN	14”	Pin 2	Black	1
		Pin 4	Red	2
		Pin 5	Yellow	3
		Body tab	Shield	N/C
MIDI IN	13”	Pin 2	Black	1
		Pin 4	Red	2
		Pin 5	Yellow	3
		Body tab	Shield	N/C

I/O wiring

1. Make audio in and out pigtails in a similar fashion to the previous step for the 1/4” jacks, **also using the included 4 wire shielded cable.** According to the table below.
2. Neatly bundle together the Audio and MID harnesses 2 times along their length with zip ties as shown/



Jack schematic



Sleeve

Ring

NC (normal close) Tip Switch

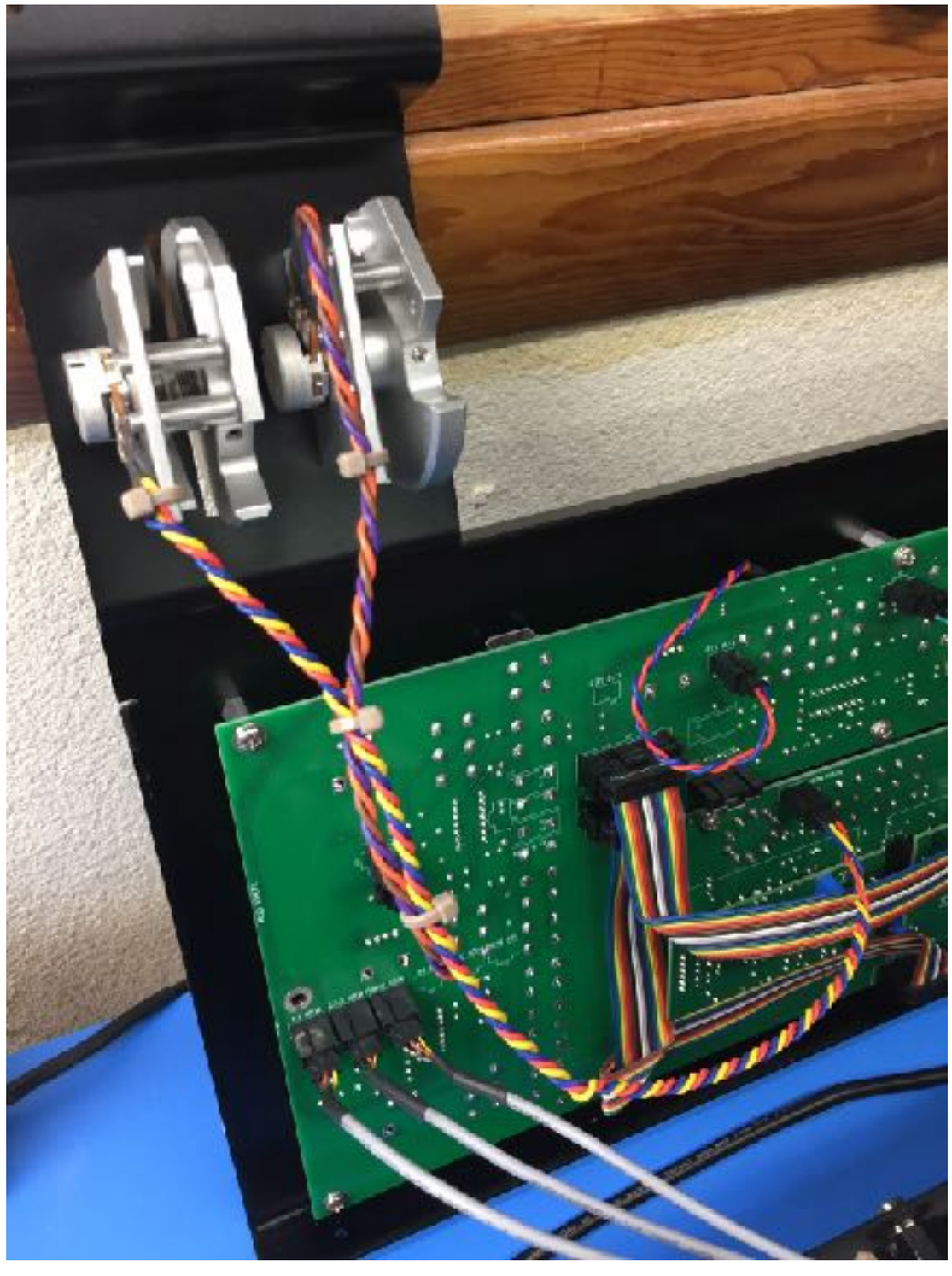
Tip

Audio harness wiring

Connector	Cable Length	Jack Connection	Wire Color	Molex Pin Number
AUDIO IN	13”	Tip	Red	2
		Ring	Yellow	3
		Sleeve	Black & Shield	1
		NC Tip Switch	White	4
AUDIO IN	13”	Tip	Red	2
		Ring	Yellow	3
		Sleeve	Black & Shield	1
		NC Tip Switch	White	4

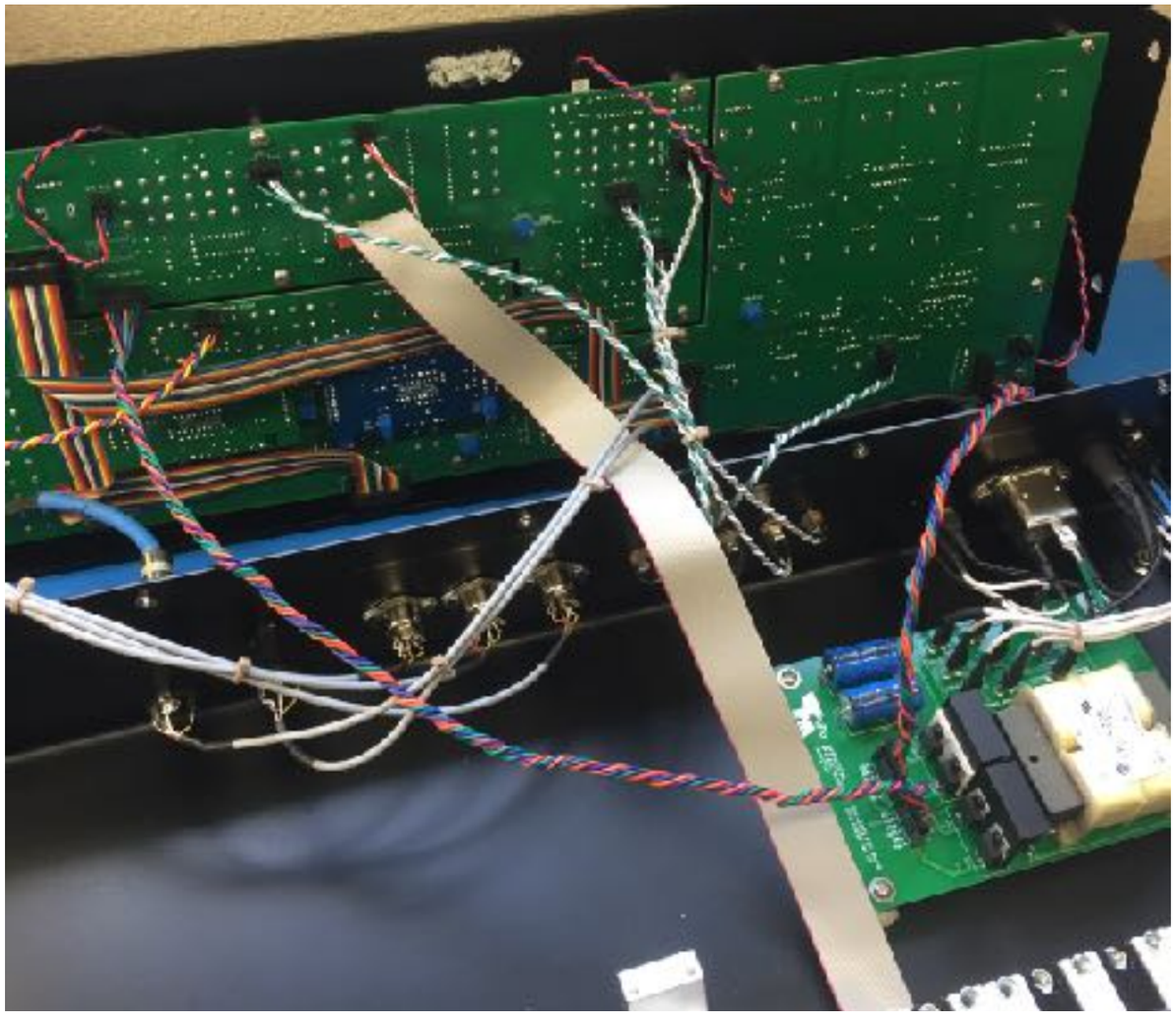
Final connections

1. Install the bender and pitch wheel assemblies to the upper chassis as shown with screws from **BAG C**.
2. Plug each into the appropriate connector and bundle the wires 2 times with zip ties as shown.



Final connections

1. Make 2 twisted power cable assemblies according to the table below and connect them as shown. It is recommended you use color coded wires and keep track of them, but it is not essential. The table shows what the author did for prototype builds.
2. Neaten up the CV/gate wires with zip ties as shown.
3. Connect the keyboard ribbon cable to the red connector on PCB “A”. Note the orientation peg that goes into the hole.

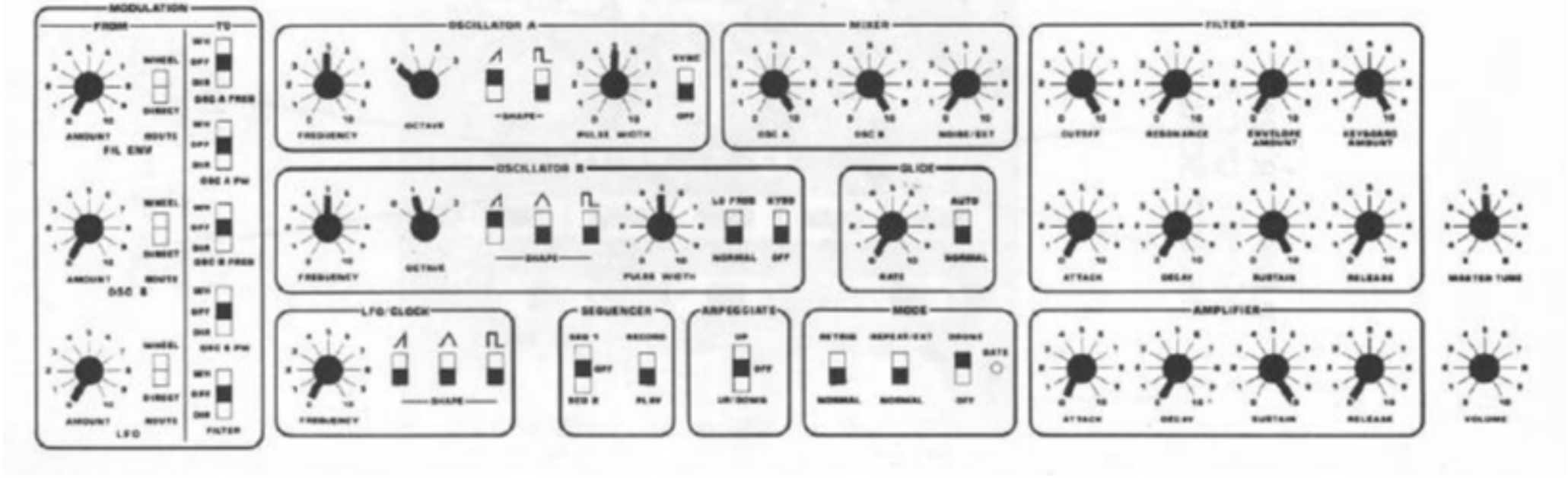


Power harness wiring

Connector	Cable Length	Function	Wire Color	Molex Pin Number
Power Cable “A”	18”	+15V	Red	1
		GND	Green	2
		-15V	Blue	3
		+5V	Orange	4
		GND	Green	5
		-5V	Purple	6
Power Cable “B”	12”	+15V	Red	1
		GND	Green	2
		-15V	Blue	3
		+5V	Orange	4
		GND	Green	5
		-5V	Purple	6

Power up and calibration

1. Leave the unit as it is laid on from the previous steps, where the chassis top is propped up but near the chassis bottom.
2. Disconnect the 2 power cables from PCB “A” and PCB “B”.
3. Decide whether you are using 120VAC supply or 240VAC supply and select the red line selector switch on the back of the unit accordingly. Install the appropriate fuse into the fuse holder (250mA for 120V supply and 125mA for 240V supply).
4. Turn on the main power switch and get out your volt meter. Probe the pins of each power cable to make sure you have the right voltages (+/- 15V and +/- 5V) and the power supply is working.
5. Assuming the power supply is OK, turn the power switch off and reconnect the power cables to PCBs “A” and “B”
6. Turn the unit back on and connect a 1/4” mono instrument cable to your amplifier or other speaker system.
7. Patch the unit up according to the image below. **If everything went well, you should be able to hit some keys and get sound!**
8. If you don’t hear anything, don’t forget to check that the volume pot (lower right) is up some and that your speakers are working.

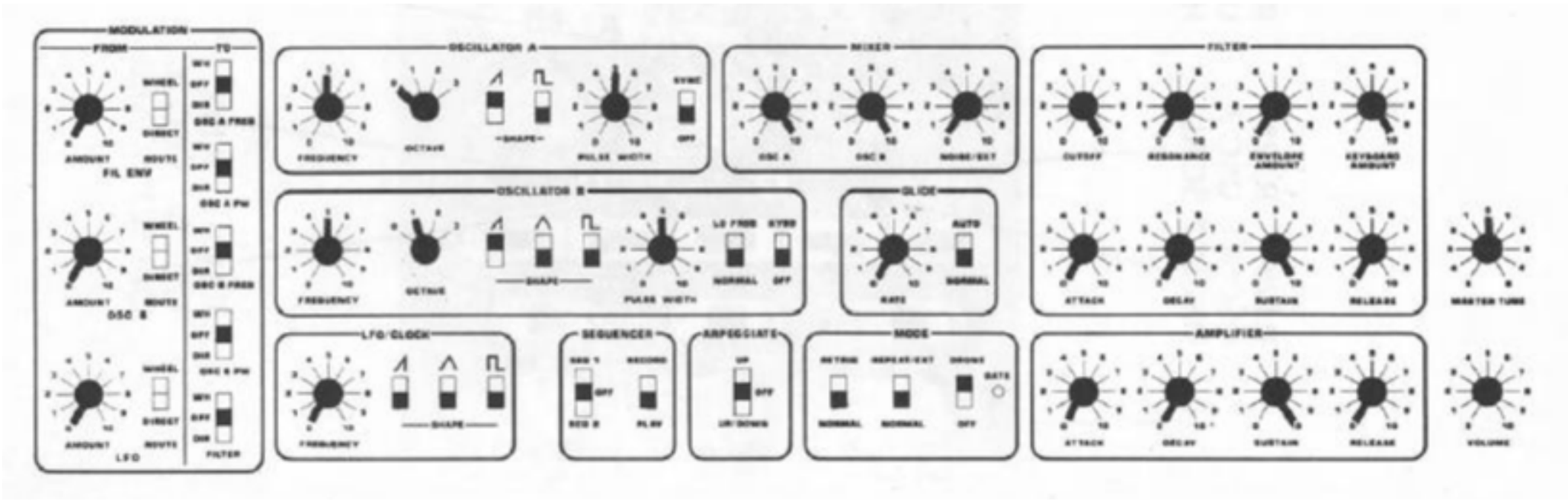


DAC Scale Calibration

1. At this point, so long as you have both oscillators working and can hear sound as you press keys, you should be ready to calibrate the unit. The first step to is to calibrate the DAC (digital to analog converter).
2. Turn on the unit and let it warm up for at least 15 minutes.
3. Hook up a DMM capable of 3 1/2 digits to the CV OUT jack wires (signal and ground).
4. Hit low C (C0) and note the value. The exact value doesn't matter but it should be something low like .0900 V.
5. Hit high C (C3) and note the reading. Adjust trim pot TP1181 DAC Scale Trim to get the reading on the DMM to be exactly 3V plus your value from step (4) (ie 3.0900 V).
6. Repeat steps 4 and 5 until you have an exact 3V difference between CV OUT voltage of C0 and C3.

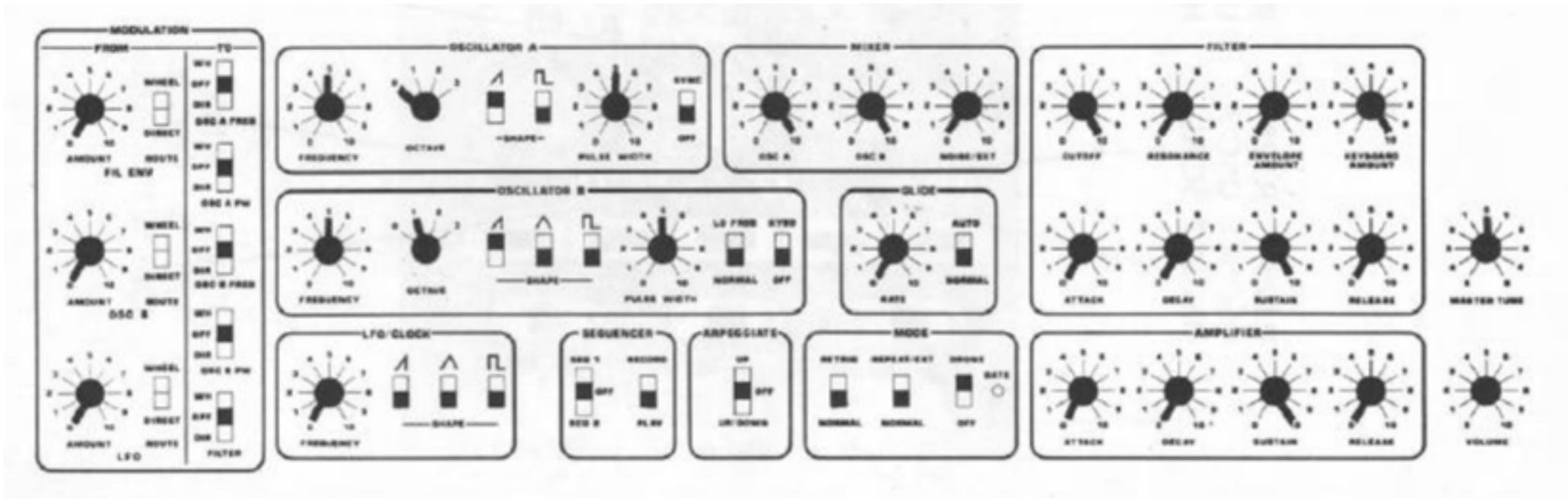
VCO 1 Calibration

1. Set up the unit to the patch shown below. Turn VCO 1 mixer level up and VCO 2 mixer level to 0.
2. Hook up the AUDIO OUT of the unit to your favorite tuner or tuner app or oscilloscope.
3. Hit low C and adjust TP120 VCO 1 Offset to get a C1 (32.7 Hz).
4. Hit high C and adjust VCO 1’s TP1 (Osc Scale) to get a C4 (261.63 Hz).
5. Repeat steps 3 and 4 until both C1 and C4 are tuned to within 1 cent.
6. Make sure low C is still tuned to C1 (32.7 Hz) and adjust VCO 1 frequency pot on the front panel if necessary.
7. Turn VCO 1 octave switch to Octave 3
8. Hit low C and Adjust TP1111 -1V Trim to get C4 (261.63 Hz).
9. Switch VCO 1 to octave 0 and repeat steps 6-9 to minimize out-of-tune cents while changing octaves.
10. Switch VCO 1 to octave 3 and hit high C and adjust VCO 1’s TP 2 (Osc Hi Trim) to minimize out-of-tune cents between hitting a low C and a high C.



VCO 2 Calibration

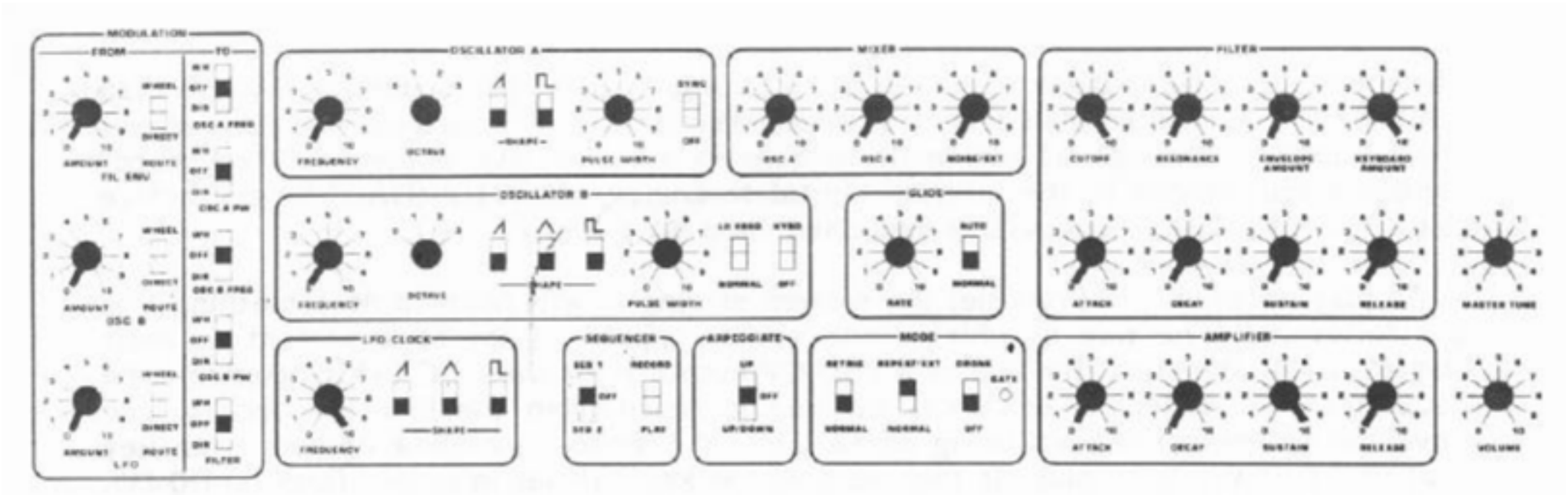
1. Set up the unit to the patch shown below. Turn VCO 2 mixer level up and VCO 1 mixer level to 0.
2. Hit low C and adjust TP120 VCO 2 Offset to get a C1 (32.7 Hz).
3. Hit high C and adjust VCO 2’s TP1 (Osc Scale) to get a C4 (261.63 Hz).
4. Repeat steps 3 and 4 until both C1 and C4 are tuned to within 1 cent.
5. Switch VCO 2 to octave 3 and hit high C and adjust VCO 2’s TP 2 (Osc Hi Trim) to minimize out-of-tune cents between hitting a low C and a high C.



VCA Balance Calibration

1. Set up the unit to the patch shown below.
2. Turn up your amplifier or speaker system to a fairly high volume to where you hear noise and thumping from the LFO.
3. Adjust TP1138 VCA Balance to minimize envelope bleed-thru.

Congratulations, your PRO2021 is now set up and calibrated. Just a few more steps and you'll be ready to play!



Finish the wood end cheeks

1. It is up to you what you want to do to finish the wood end cheeks, but here is some information that may be helpful and a recommended way to treat the wood ends. The end cheeks are made out of cherry wood and a few tips will help you make them beautiful!
    1. Cherry naturally DARKENS over exposure to UV light over time. So it's generally not advised to add any dark stains as it WILL get darker over time.
    2. Cherry shows scratches so always sand WITH THE GRAIN, not in circles, etc..
2. To finish the ends in a beautiful, natural manner, follow the following steps:
    1. Sand with a coarse sanding block (~100 Grit), with the grain to remove any scratches or tooling marks as you see fit.
    2. Sand with a medium sanding block (~150 Grit) with the grain on all surface except the pocket.
    3. Sand with a fine sanding block (~250 grit) with the grain on all surfaces except the pocket.
    4. Apply two liberal coats of Tung oil, rubbing in with a rag.
    5. Wait a day and make sure the parts are clean and there's no excess oil on them.
    6. Apply two coats of Howard Wood Shield Premium Paste Wax per the directions on the can.
    7. Your end cheeks should now be smooth and beautiful, with the rich grain of the Cherry showing through!!

Cherry is a unique and beautiful wood that darkens over time!

Final Assembly

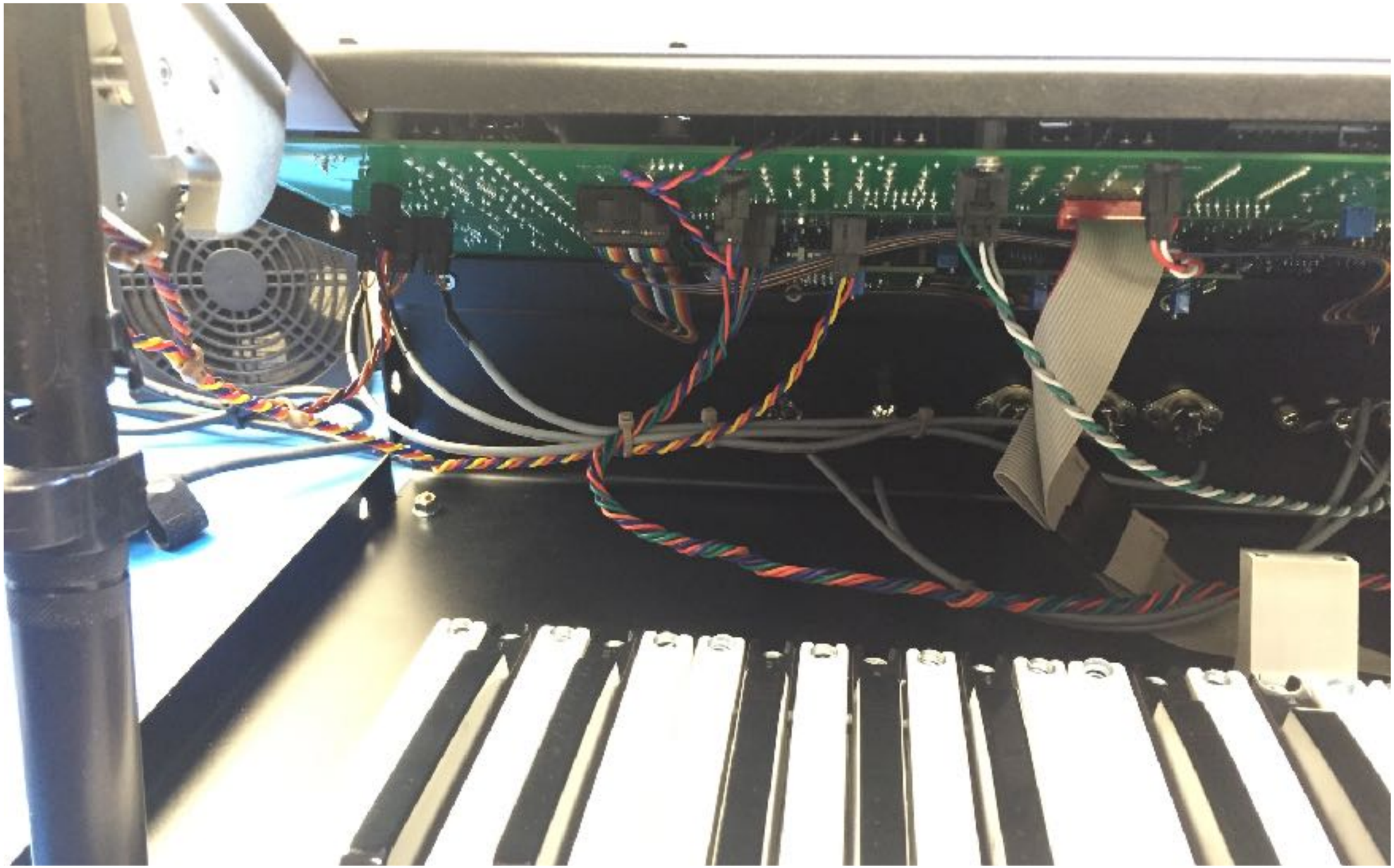
1. Prop the case top over the case bottom with a stick or similar item (solder sucker stick shown), being careful to not wrench on the PCBs in tieback where the lower chassis sheet metal sticks up!

Final Assembly

1. Tie up the “B” per cable out of the way with the neighboring CV cable as shown so that bot cables steer clear of the power supply cables and heat sinks.
2. Make sure any other cable are steered clear of the power supply and heat sinks.
3. Tie or tape up the extra length of the keyboard flat cable as shown.

Final Assembly

1. Tie up the MIDI cables with the “A” per cord and modulation board as shown.

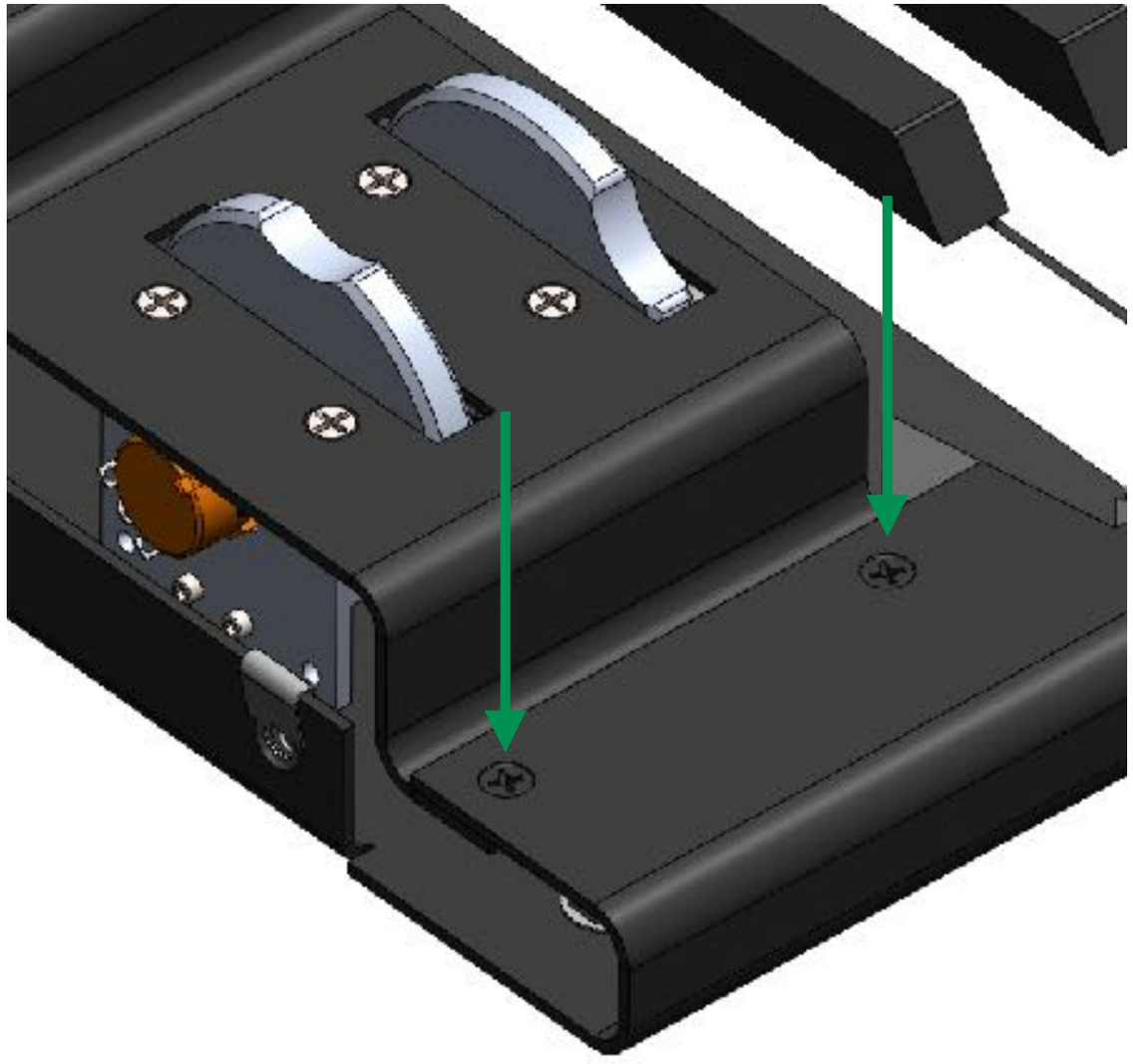


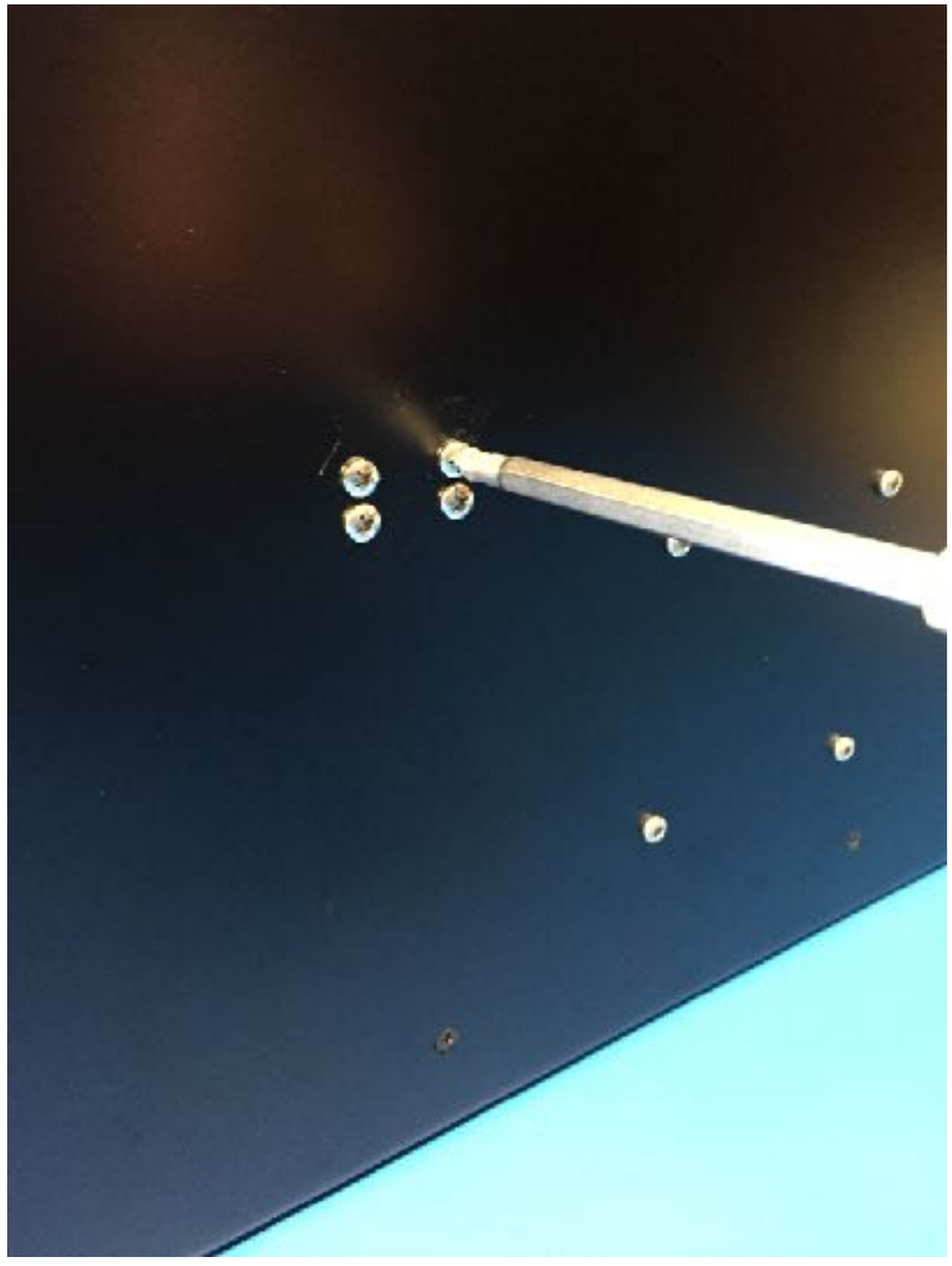
Final Assembly

1. Carefully remove your propping device and lower the upper chassis down onto the lower chassis. The pitch bend side of the upper chassis needs to slip underneath the lower chassis.
2. Install and tighten 2 screws from **BAG C** into the chassis brace as shown in the first picture.
3. Install and tighten the 4 screws form **BAG C** where the top chassis meets the bottom chassis as shown in the second picture.

Final Assembly

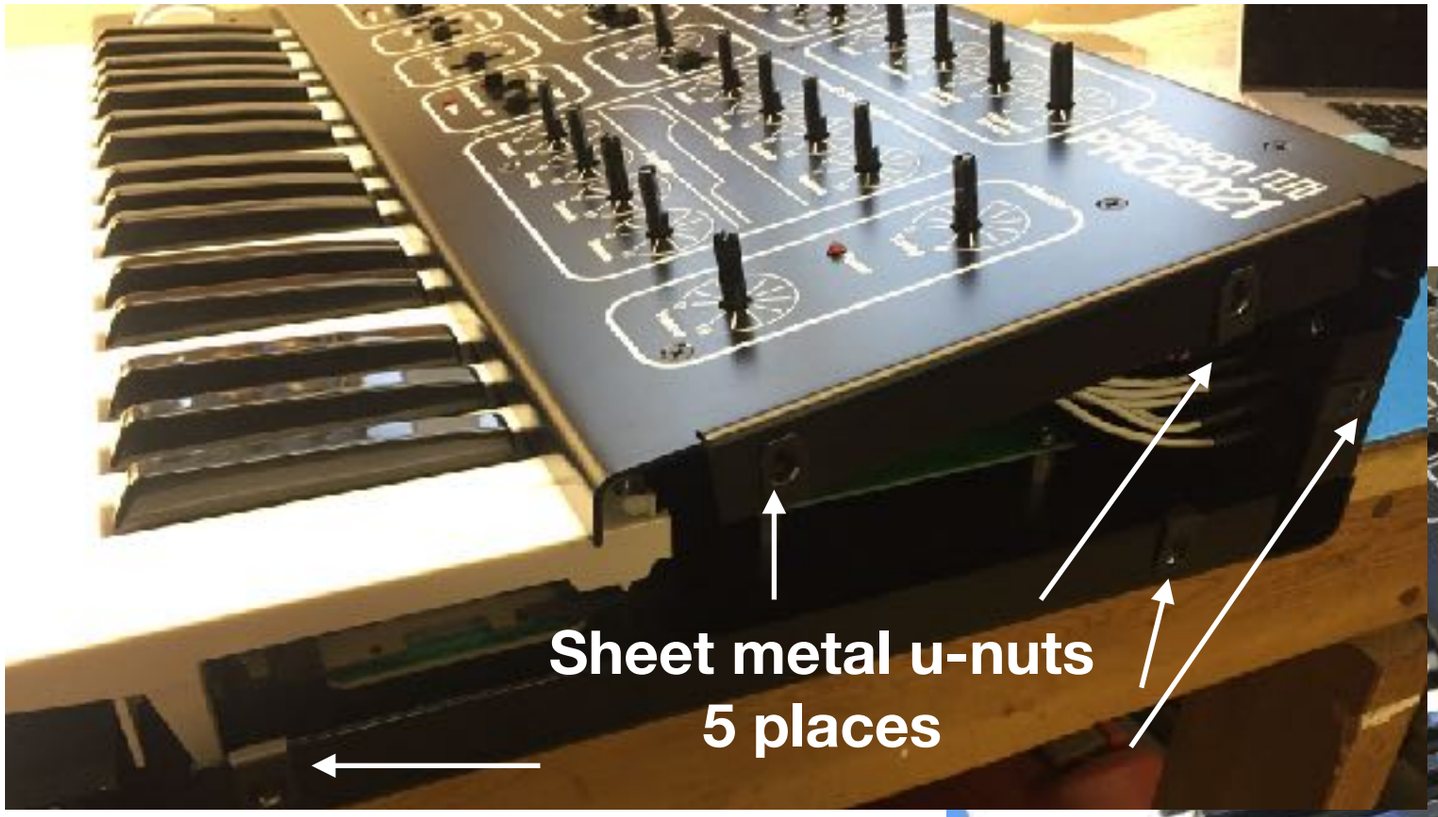
1. Install and tighten 2 screws from **BAG C** below the pitch/mod wheels as shown in the first picture.
2. Turn the unit up on it's side and tighten the 4 chassis brace bottom screws that you left slightly loose in the beginning of the build as shown in the second picture.





Final Assembly

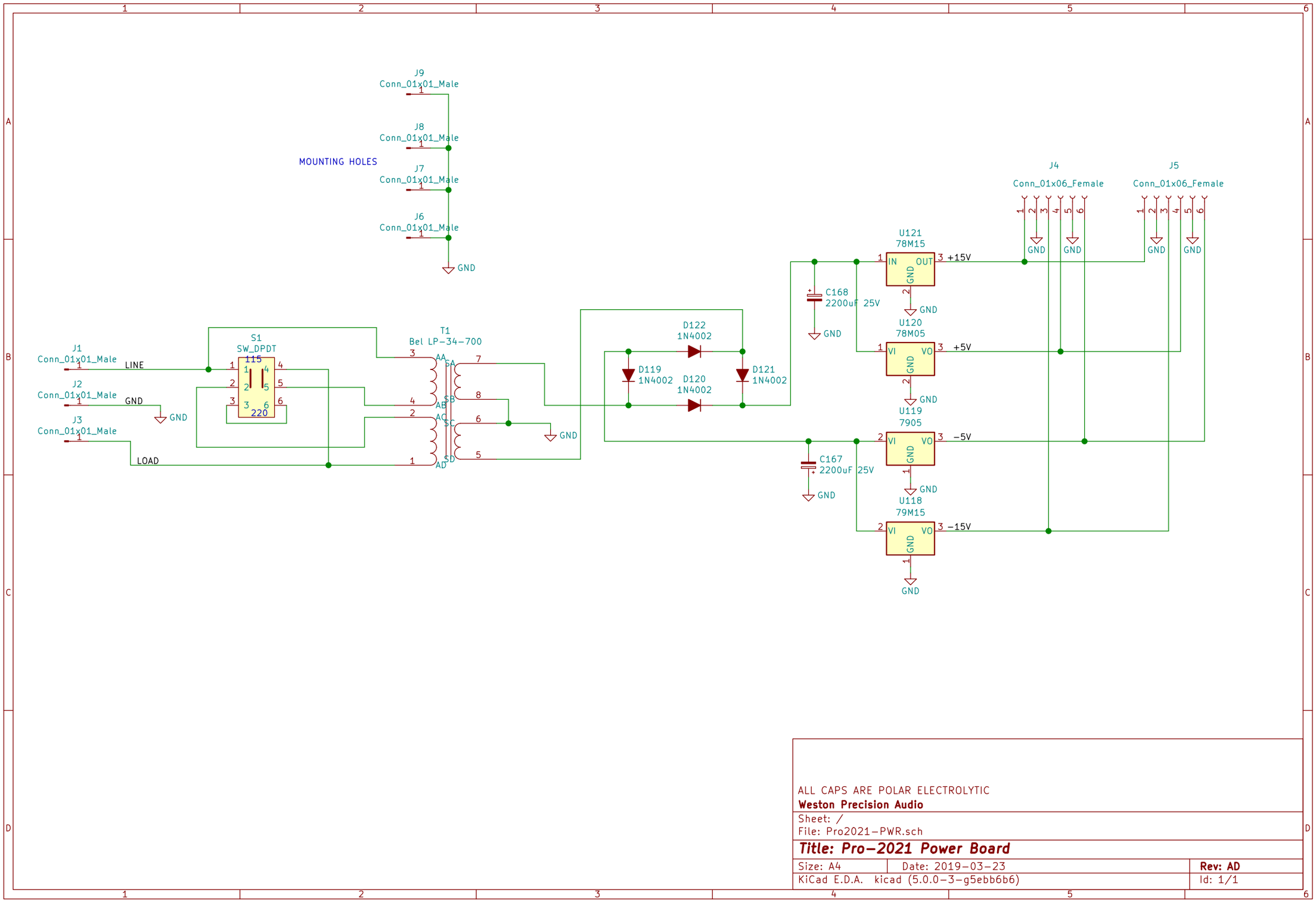
1. Install 5 sheet metal u-nuts from **BAG H** onto their holes on the right side as shown. You should feel/here the front tab of each nut *click* into the whole meaning it is positioned correctly and sitting flush!.
2. Install the right end cheek with 5 flat head socket head screws from **BAG H** as shown. Incrementally tighten them as you would lugs on a car wheel until the cheek is pulled up flush and in position.
3. Repeat for the left side of the PRO2021.



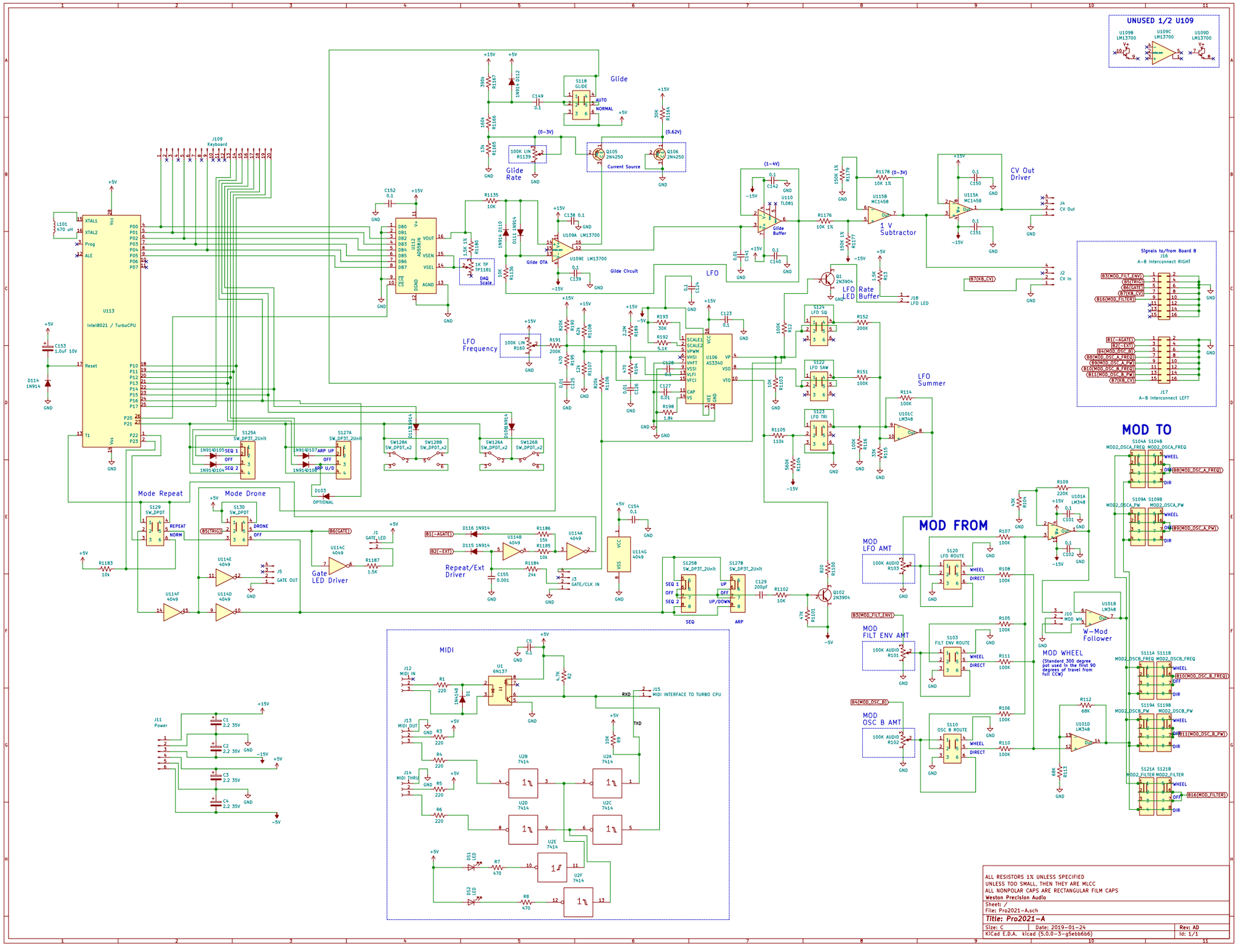
Final Assembly

1. Install your knobs.
2. Congratulations! You are the proud owner of a PRO2021 synthesizer that YOU built! Go have a beer, but don't spill any on your new instrument! Then, PLAY THAT THING!

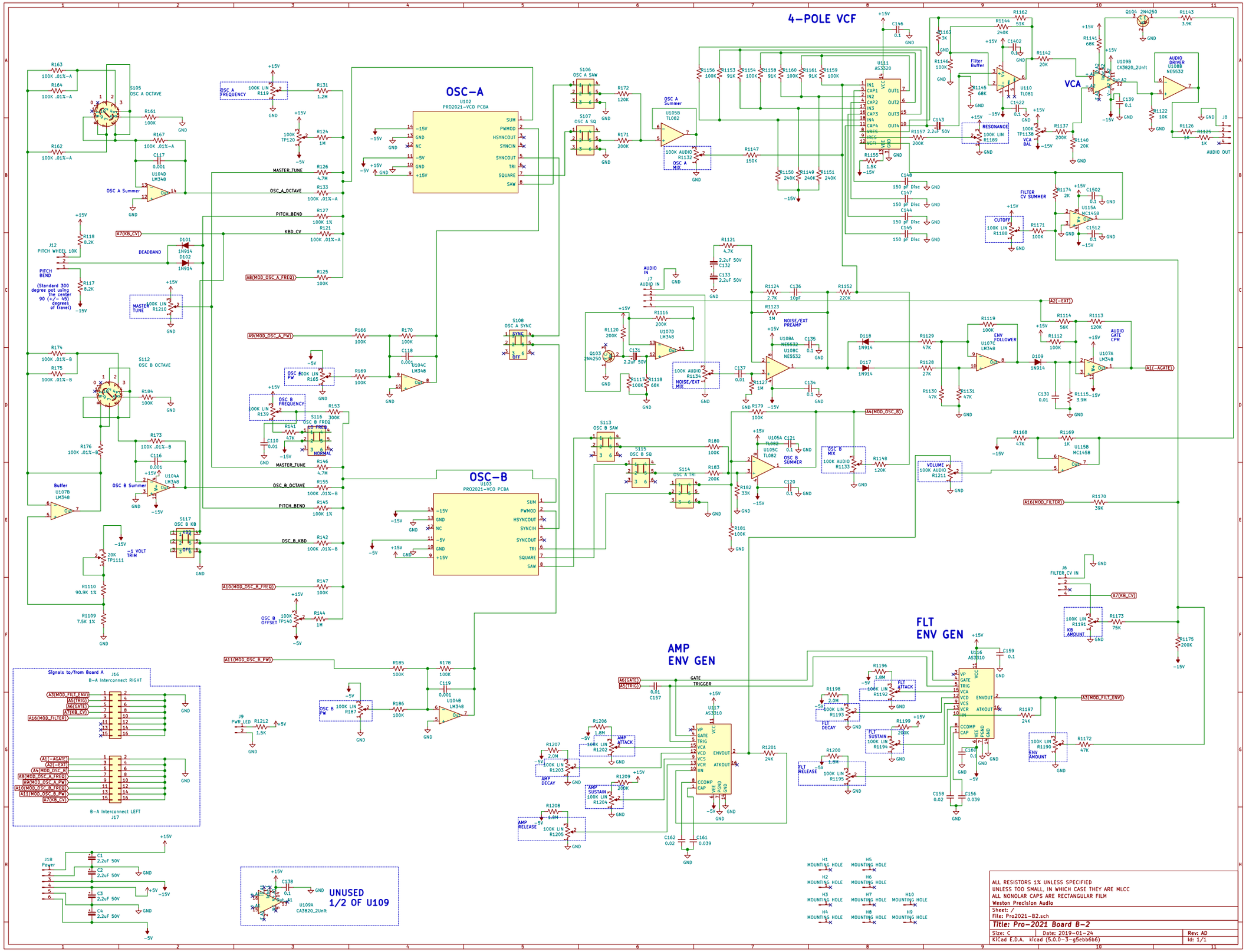
Appendix - Schematics / Power Supply



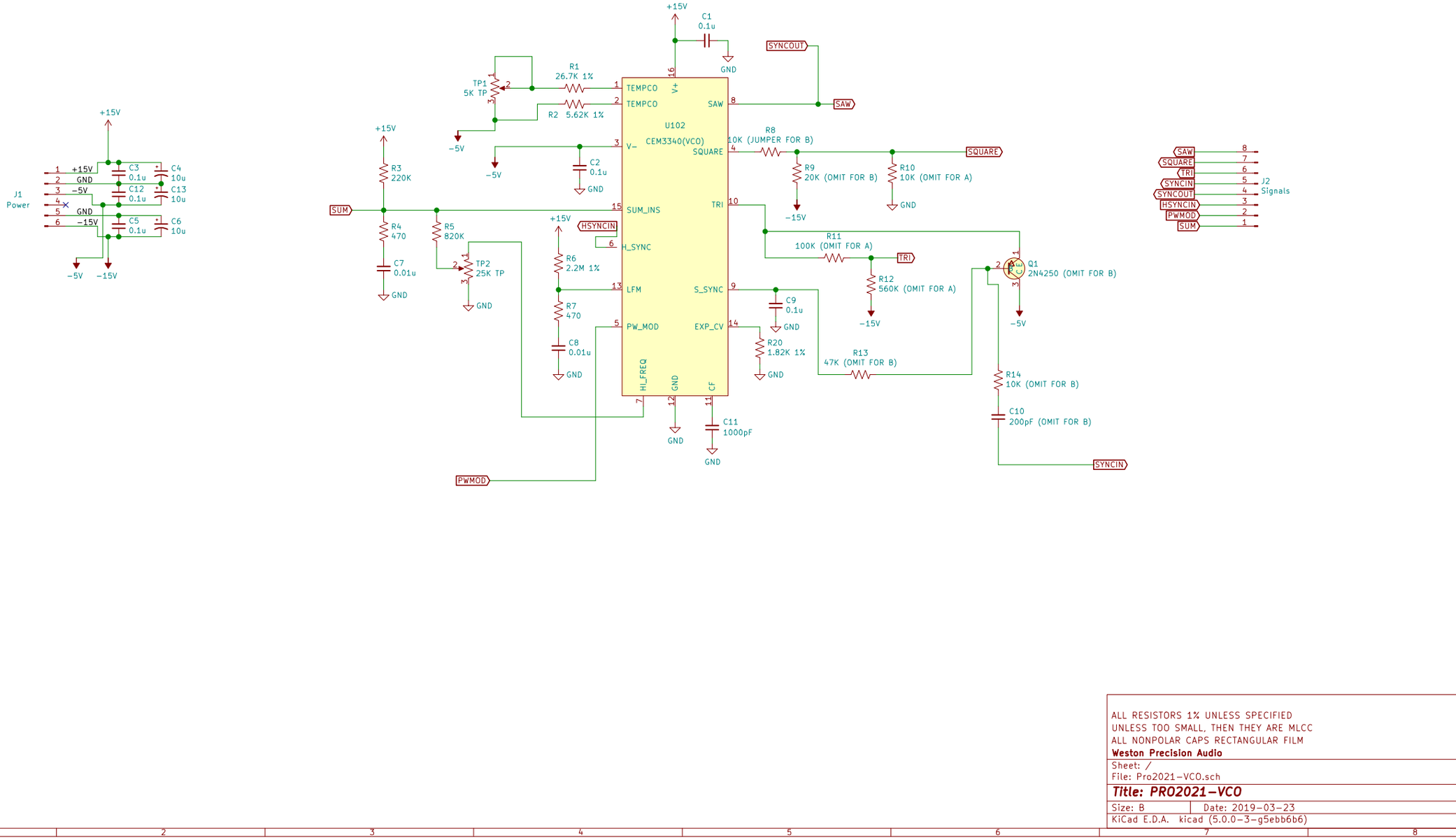
Appendix - Schematics / PCB “A”



Appendix - Schematics / PCB “B”



Appendix - Schematics / PCB VCO



ALL RESISTORS 1% UNLESS SPECIFIED
UNLESS TOO SMALL, THEN THEY ARE MLCC
ALL NONPOLAR CAPS RECTANGULAR FILM
Weston Precision Audio

Sheet: /
File: Pro2021-VCO.sch

Title: PRO2021-VCO

Size: B	Date: 2019-03-23	**Rev: AD**
KiCad E.D.A. kicad (5.0.0-3-g5ebb6b6)		Id: 1/1

Appendix - Resources

1. Original SCI Pro-One schematics: http://www.synthfool.com/docs/SequentialCircuits/Pro_One/
2. Original SCI Pro-One Technical Manual: https://www.manualslib.com/manual/1324788/Sequential-Pro-One.l
3. Music Technologies Group Turbo CPU: http://www.musictechnologiesgroup.com/
4. Modular Addict “Jove” knobs: https://modularaddict.com/knurled-knob-18mm
5. Tips for finishing Cherry wood: https://www.popularwoodworking.com/projects/tips-for-finishing-cherry/
6. Original PRO2021 MuffWiggler thread: https://www.muffwiggler.com/forum/viewtopic.php?t=209075