Conspiracy to Commit Electronics – Steddi Go

Steddi Go is a simple, no-frills, great sounding tube preamp/D.I. that works well on any instrument.

DISCLAIMER AND WARNING

This circuit contains high voltages exceeding 200V and is EXTREMELY DANGEROUS. Conspiracy to Commit Electronics is not responsible for any damage or injury caused by improper use or assembly. I encourage you to use the utmost care when building, testing, and using this pedal. If high voltages make you uncomfortable, DO NOT BUILD THIS. Just don't. This is not a beginner project and should not be treated as such. It was designed to be as easy as possible to assemble and make it work, but **you have to be careful**.

Normally I would recommend testing a circuit before putting it into the box, but in this case I recommend fully boxing the unit before testing for the sake of safety. If for any reason you need to probe voltages inside the box, do so with extreme caution, and only keep one hand near the box at a time, do not allow both hands to touch the box/circuit at the same time.

Recommended Build Instructions

Since this build is not only more complicated but also very different than typical DIY pedal builds, I've provided detailed build instructions at the end of this document. You're welcome to try your own method, but the boards were designed to be assembled using the methods explained in this document.

Bill of Materials

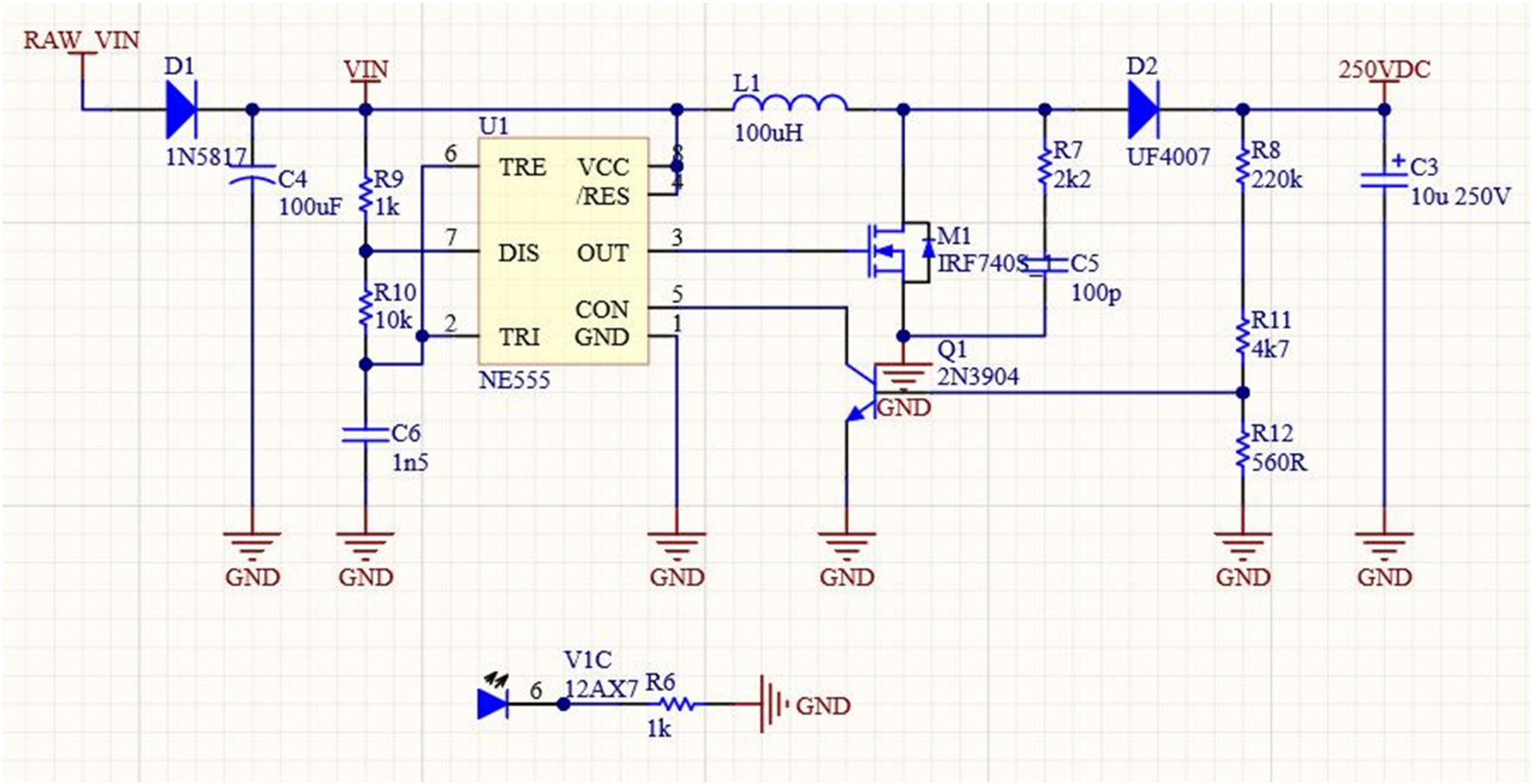
The links below are recommendations and suitable replacements can be used as needed.

Comment	Description	Designator	Qty	Link	Alternate
51R	1/4W Resistor	R1 (XLR PCB)	1	Tayda Link	
10n 100V	Film capacitor	C1 (XLR PCB)	1	Tayda Link	
AC3MAV2	XLR jack, PCB mount	J4 (XLR PCB)	1	Mouser Link	
NC3MAAV	XLR Jack (alternate)	J4 (XLR PCB)	1	Mouser Link	
SPDT	LONG BAT switch	GND LIFT (XLR PCB)	1	Tayda Link	
Screw	M3x12mm self-tapping	M3, M4 (XLR PCB)	2	Amazon	
JT-DB-EPC	D.I. Transformer	X1 (XLR PCB)	1	Jensen	
MC15 (alt)	D.I. Transformer (alternate)	X1 (XLR PCB)	1	Lightning Boy Audio	
E4305 (alt	D.I. Transformer (alternate)	X1 (XLR PCB)	1	C2C Electronics	

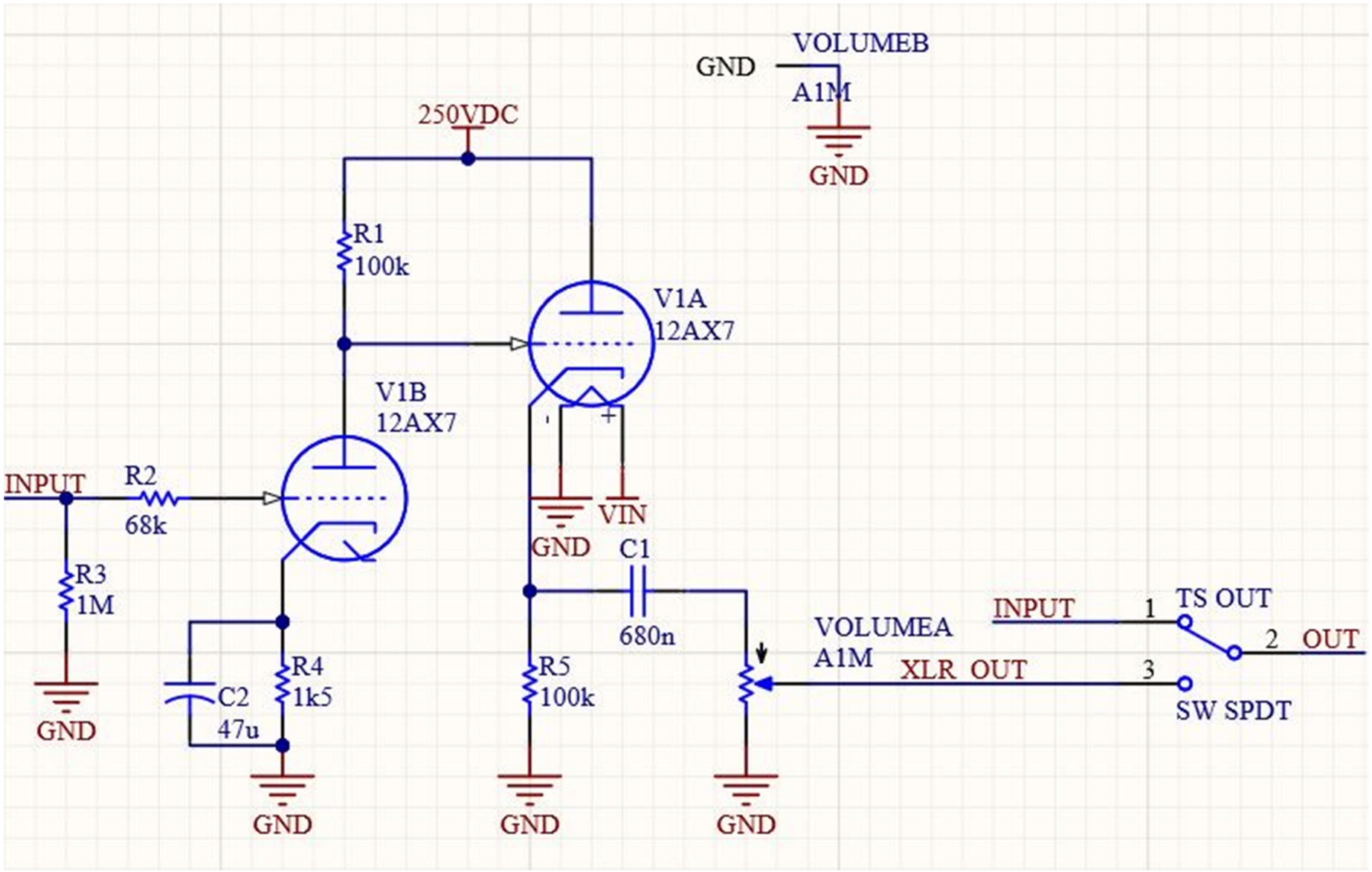
Comment	Description	Designator	Qty	Link	Alternate
100k	1/4W Resistor	R1, R5	2		
68k	1/4W Resistor	R2	1		
1M	1/4W Resistor	R3	1		
1k5	1/4W Resistor	R4	1		
1k	1/4W Resistor	R6, R9	2		
2k2	1/4W Resistor	R7	1		
220k	1/4W Resistor	R8	1		
10k	1/4W Resistor	R10	1		
4k7	1/4W Resistor	R11	1		
560R	1/4W Resistor	R12	1		
680n	Film capacitor	C1	1		
47u	Electrolytic capacitor	C2	1		
10u 250V	HV Electrolytic capacitor	C3	1		
100u	Electrolytic capacitor	C4			
100p	Ceramic capacitor	C5	1		
1n5	Film capacitor	C6	1		
A1M	9mm Potentiometer	Volume	1		SBP Link
SPDT	SPDT ON/ON Toggle switch	1/4” OUT	1		
1N5817	Schottky diode	D1	1		
UF4007	Ultra-fast rectifier	D2	1		
1N4007	Generic rectifier	D3	1		
100uH	100uH 2A Inductor	L1	1		
IRF740	N-Channel MOSFET	M1	1		
2N3904	NPN Transistor	Q1	1		
NE555	Timer	U1	1		
12AT7 /12AX7	Dual Triode	V1	1		
9-pin socket	Tube socket	V1	1	AP Link	

Comment	Description	Designator	Qty	Link	Alternate
2x5 header	2x5 right angle	V1	1	Tayda Link	
Standoff	M3x10mm standoff	M1, M2	2	Tayda Link	
Nut	M3 nut	M1, M2	2	Tayda Link	
Screw	M3x8mm machine screw	M1, M2	2	Tayda Link	
1/4"	1/4" audio jack	J1, J2	2	AP Link	LMS Link
2.1mm	2.1mm DC jack	J3	1	AP Link	LMS Link
Knob	(your choice)	External	1	Tayda Link	
LED	(your choice)	Tube daughter PCB	2	Tayda Link	

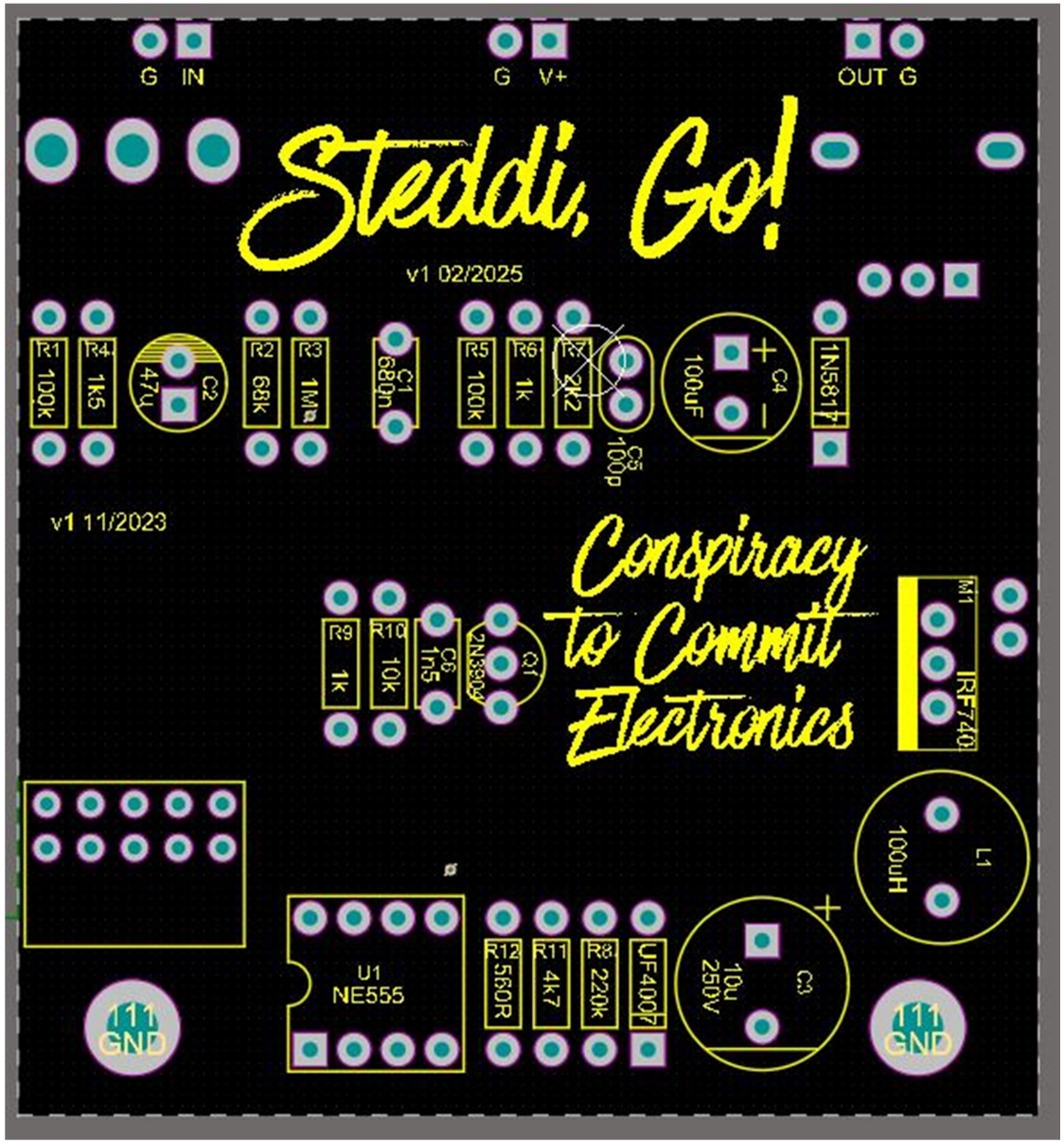
Schematic: Power Supply



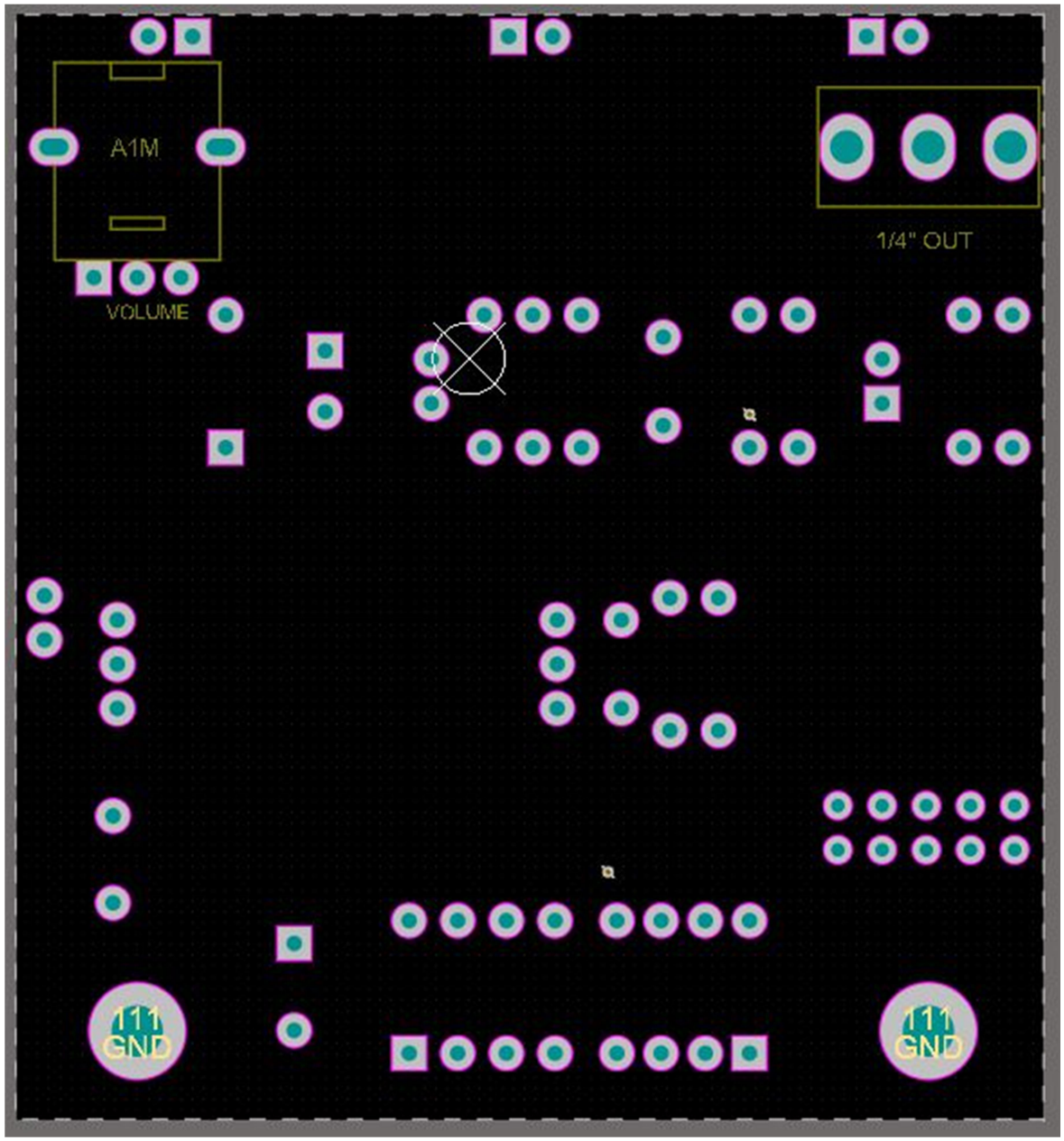
Schematic: Audio


GND
VOLUMEB
A1M
GND
250VDC
R1
100k
V1A
12AX7
V1B
12AX7
INPUT
R2
68k
GND
VIN
GND
C1
680n
R3
1M
GND
C2
47u
R4
1k5
GND
R5
100k
GND
VOLUMEA
A1M
GND
INPUT
1
TS OUT
2
OUT
XLR OUT
3
SW SPDT

Board Layout - Top



Board Layout - Bottom



Drill Template – 1590BB2

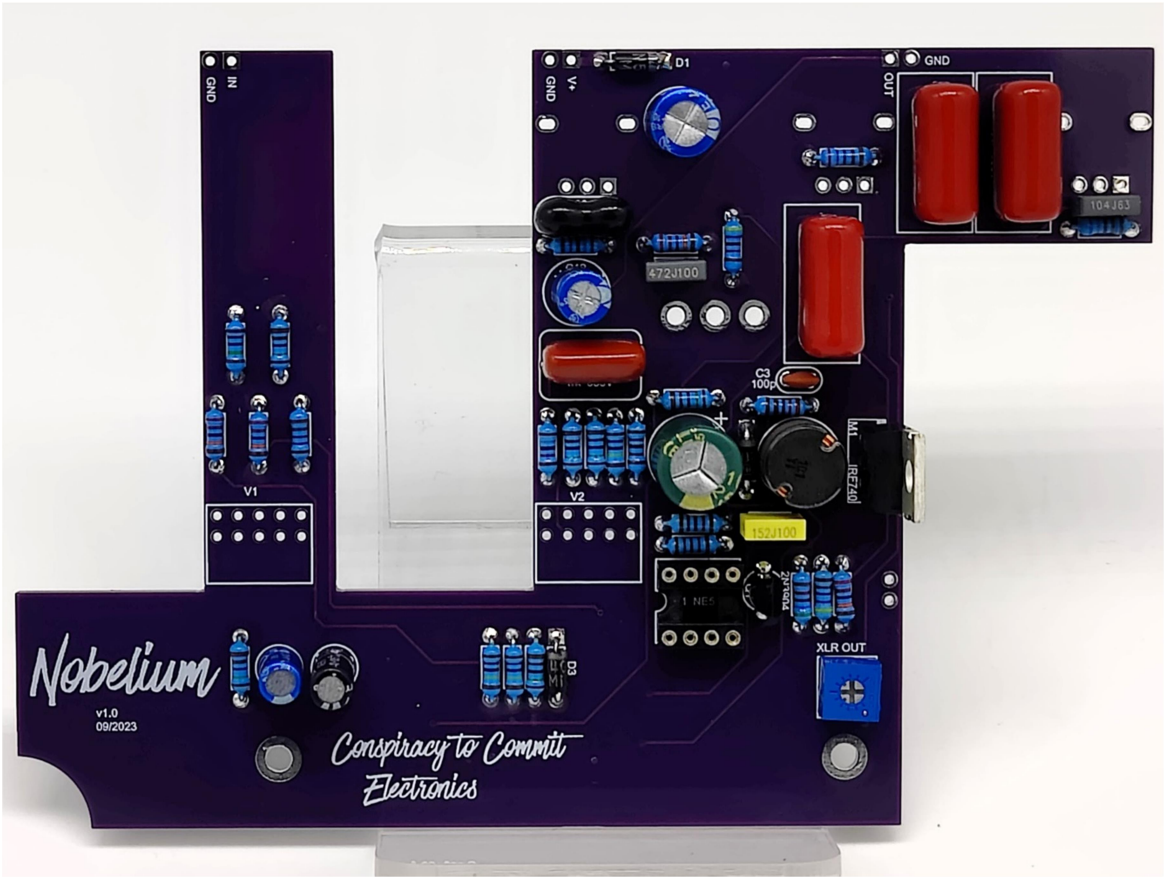
Print to 100% scale; provided as a reference with no guarantees. The small holes to the right side of the enclosure face are optional for tube ventilation. Feel free to use as many (or few) of them as you want.

Drill Template coming soon

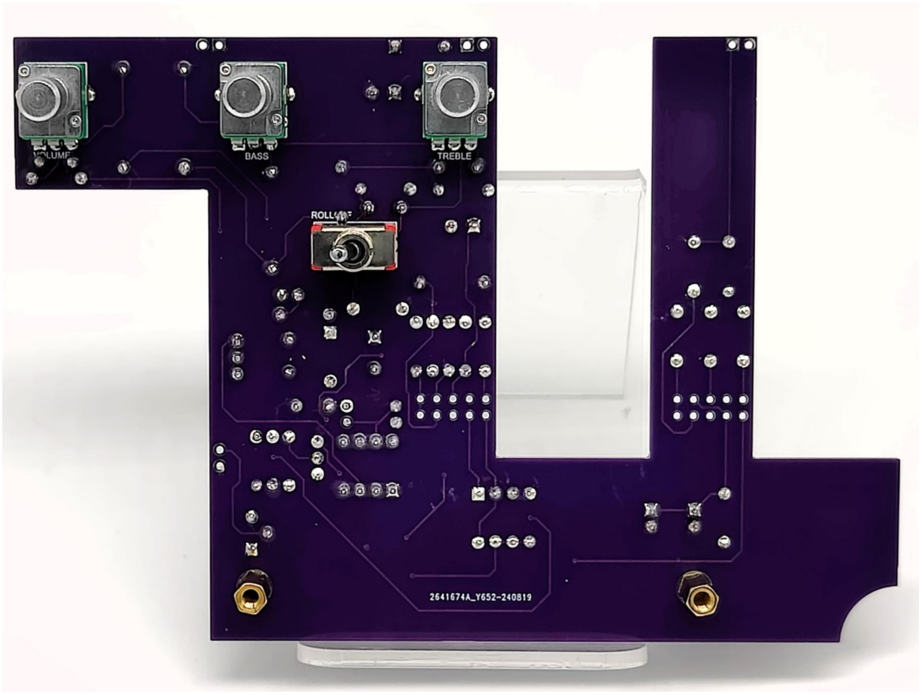
Detailed Build Instructions

Due to the complexity of the build, please reference the following instructions. Due to similarities to the Nobelium build process, Nobelium images have been used here.

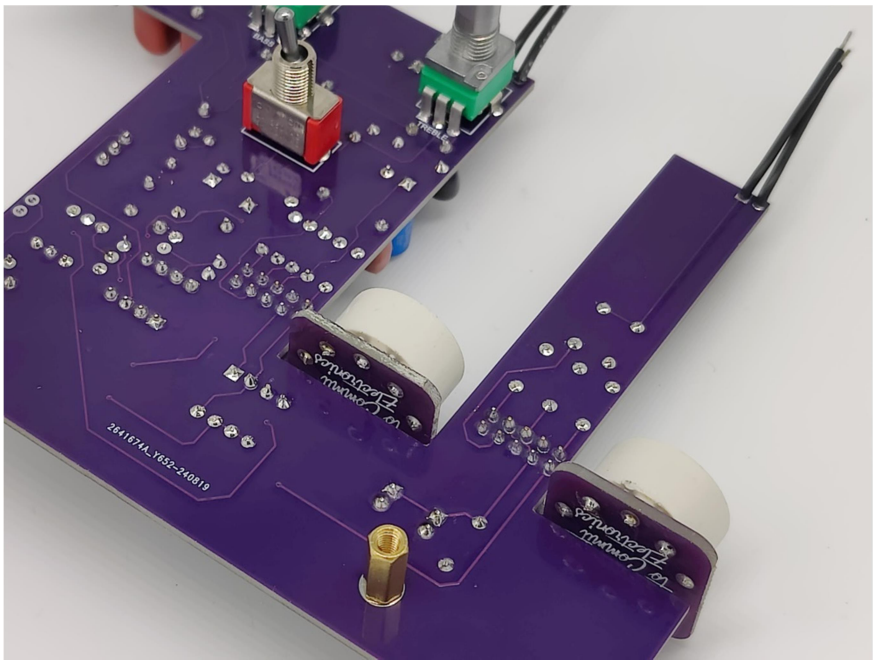
1. Assemble the main PCB. Solder the components to the top of the main PCB and clip the excess leads from the bottom.



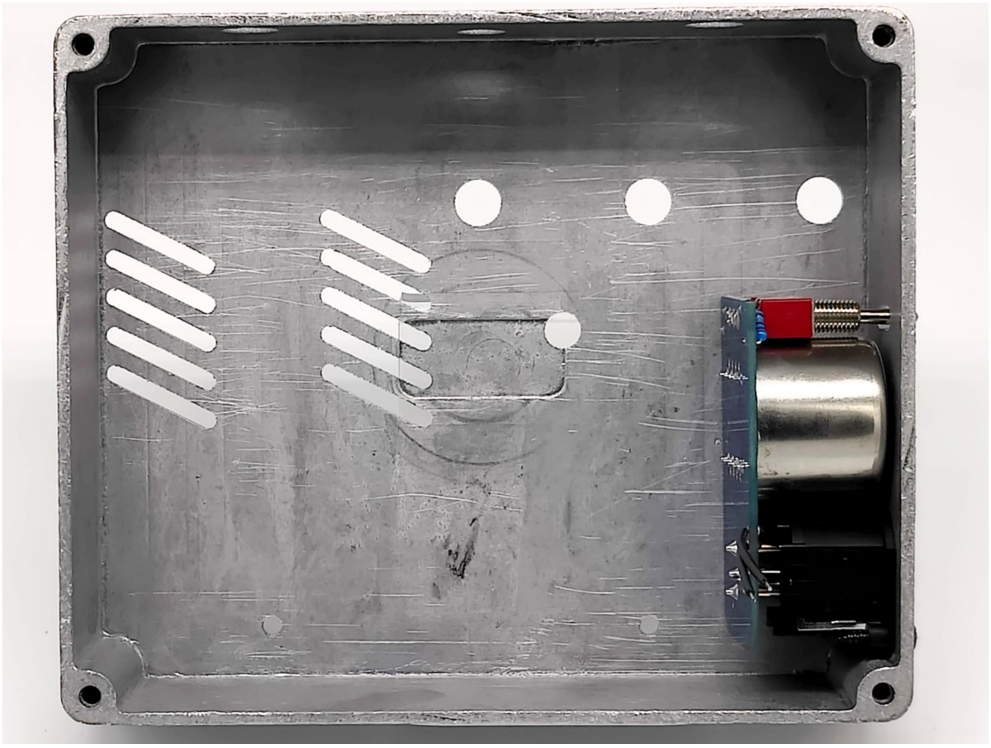
2. Attach the pots. Connect the pots to the bottom side of the main PCB and solder them in place. The two pins on the side of the body are for stability and are also used as a ground connection to the chassis and it is recommended that you solder them in addition to the 3 main signal pins. Secure the standoffs to the main PCB with the M3 nuts.

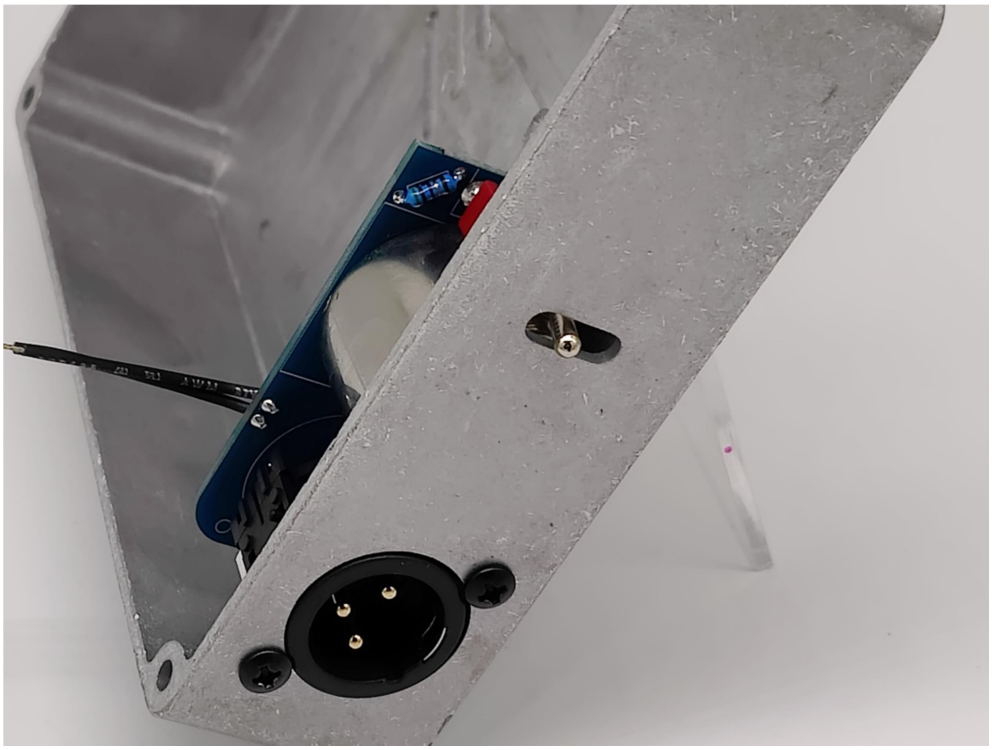


3. Attach the Tube Daughter Boards. After assembling the tube daughter board per these instructions, slide the pins into the pads on the board until the daughter board sits flush with the main PCB, then solder the pins in place from the bottom of the main PCB. Solder the wires to the pins on top of the PCB.

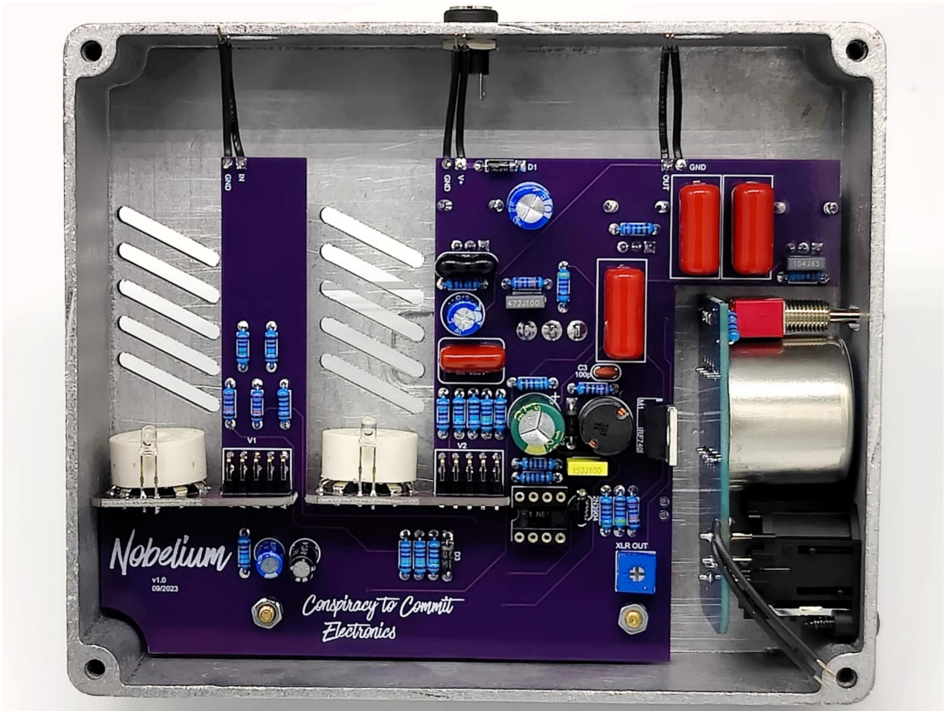


4. Secure the XLR daughter PCB into the enclosure by the M3 self-tapping screws. Ensure that the toggle switch for the ground lift can move freely I the slot.



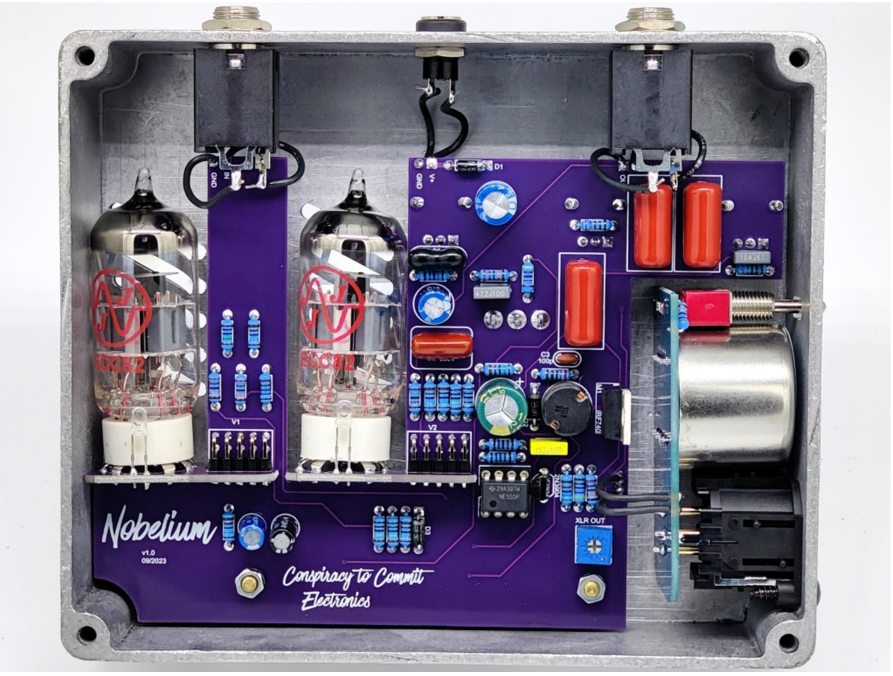


5. Insert the main PCB into the enclosure. If using a faceplate, place the faceplate on the outside of the enclosure, fitting over the pots and switch. Secure the faceplate through the enclosure to the main PCB using the nuts on the pots, the nut on the toggle switch, and the M3 machine screws into the standoffs of the main PCB.



6. Insert the jacks and secure them with their respective hardware. Solder the wires from the PCB to the jacks. Solder the wires from the XLR daughter to the respective pads of the main PCB.

7. Insert the tube and NE555. I'm specifically mentioning the NE555 because if you plug in the pedal without the NE555 then your MOSFET will get very hot and potentially damage itself, so make sure it gets inserted before being plugged in. The way I like to do this is to put the tube in angled away from the pedal and line up pins 1 and 9 with the socket, then tip the tube backwards toward the enclosure to seat the remaining pins, then push down until the tube sits flush in the socket.



8. Add knobs on the outside and you're good to go! Plug it in and try it out! If you have any issues or questions regarding assembly or troubleshooting if it doesn't work, please reach out on the [PedalPCB forum] for assistance.