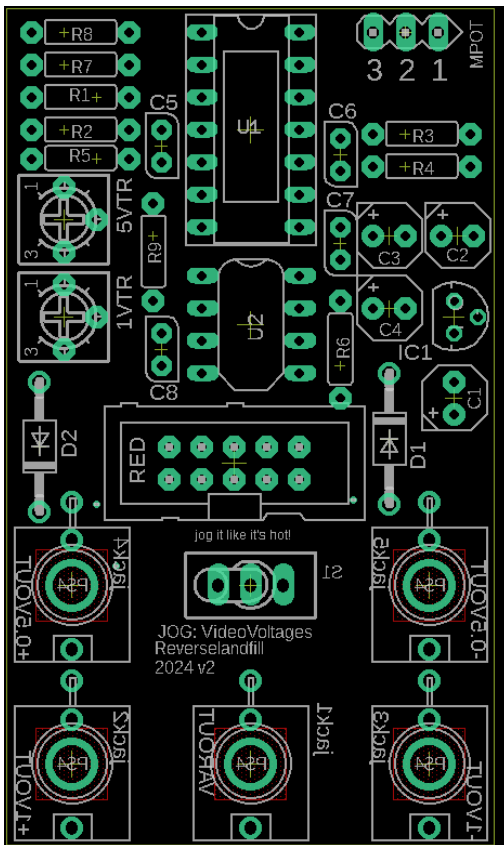


JOG Buildguide v2 2024



*Use this guide together with the JOG BOM ,
so make sure you have the latest version:*

<https://github.com/MartijnVerhallen>

Resistors

Solder all resistors!

Diodes

These parts have an orientation. the black line on the diode should match the white line on the PCB. Place the black 1N4001 diodes at D1 and D2 and solder them.

Ceramic Capacitors

Place the 4x 100nF capacitors and solder them.

IC sockets

Place the sockets (take care of the orientation

Solder one pin and check if the socket is flat to the surface of the PCB, reheat if needed.

Then solder the rest of the pins.

IC1 is the 78L05 voltage regulator.

Trimpots

Mount the two 100k trimpots.

(the ones in the kit need to be pushed in firmly, until they click a bit deeper into place)

Solder one pin and check the alignment (reheat if necessary), then solder the other pins.

Power header

Place the 10pin power header and flip the PCB around. Solder one pin and check if the header is flat to the PCB surface. Reheat the soldered pin if needed, then solder the other pins.

Electrolytic Capacitors

Place the three 10uF and the one 1uF Electrolytic Capacitors.

The Longer lead is the PLUS side, marked on the PCB with a small + sign and a triangle.

>>>> Flip over the PCB

Jacks and the Switch

Place the jack sockets on the PCB. DO NOT SOLDER YET!

Mount the switch (remove both nuts)

To align the jacks and switch correctly, we now mount the panel first.

Attach the nuts to the jacks and loosely fasten them.

Check if everything is aligned to the panel, then solder the jacks.

Potmeter

The Bourns multiturn potmeter included in the kit has a different pinout than regular potmeters.

Remove the nut and rings and mount the potmeter to the panel.

The pins should stick out 45 degrees to the right, next to the MPOT pads.

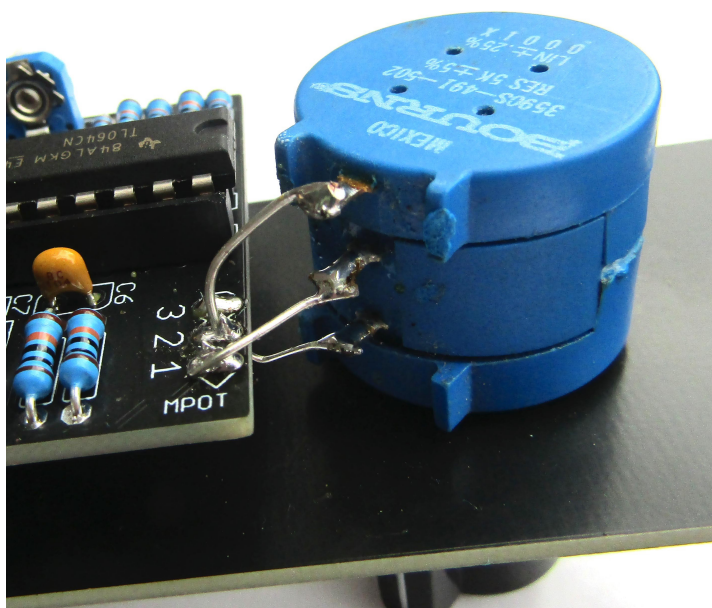
You can use snipped off resistor leads or short wire to connect the potmeter to the PCB.

The MPOT header has 3 pads, 1, 2 and 3

Pad 1: middle pin of the potmeter

Pad 2: upper pin

Pad 3: lower pin



Checking the PCB for faults:

Check the orientation of the IC's, diodes, electrolytic capacitors and the power header
Look at your solder work for shorts and missed pads.

Trimpot calibration:

the trimpot marked 5VTR sets the output of the lower 2 jacks.

Connect a multimeter or scope to the lower right jack and turn this trimpot until the voltage reads 1v

the trimpot marked 1VTR sets the output of the upper 2 jacks.

Connect a multimeter or scope to the lower right jack and turn this trimpot until the voltage reads 0.5v

The jacks on the left side put out the inverted voltage of 1v and 0.5v

Test the calibration by connecting the middle jack (variable output) and turning the potmeter all the way to the right (CW). The voltage should be 1v.

Now turn the potmeter all the left (CCW).

Depending on the switch position the voltage will be -1v or 0v.

Flip the switch to check this.

Now the module is calibrated!

Note that the two trimpots can be set to other ranges.

The 5VTR sets the main output (both the variable output range and the lower jacks)
(from 5v to 1v)

The 1VTR only sets the upper jack range, this can be any useful voltage to your system
(from the 5VTR setting to 0v)

MODS

For audio CV voltages, you can use a 5v setting of the 5VTR trimpot
(this will give you a 0 to +5v or -5v to +5v range)

If you need higher voltages, use another voltage regulator such as the 7809 or 7808
make sure the pinout of the transistor is the same.