Les Boutiques

Building instructions

v1.1

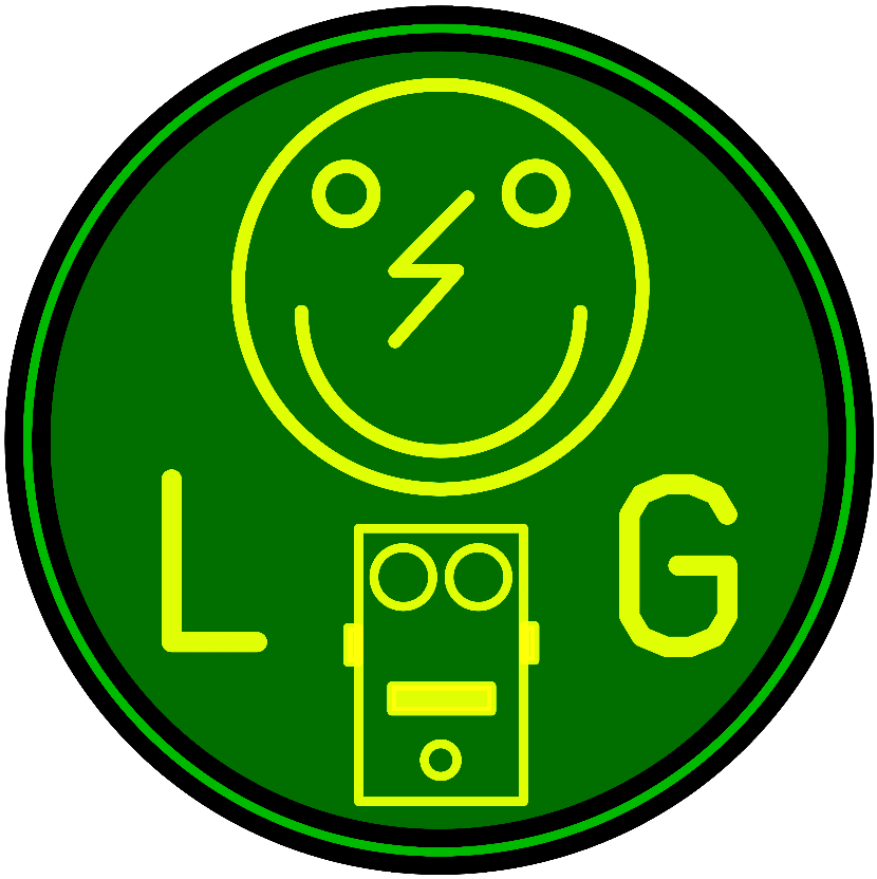
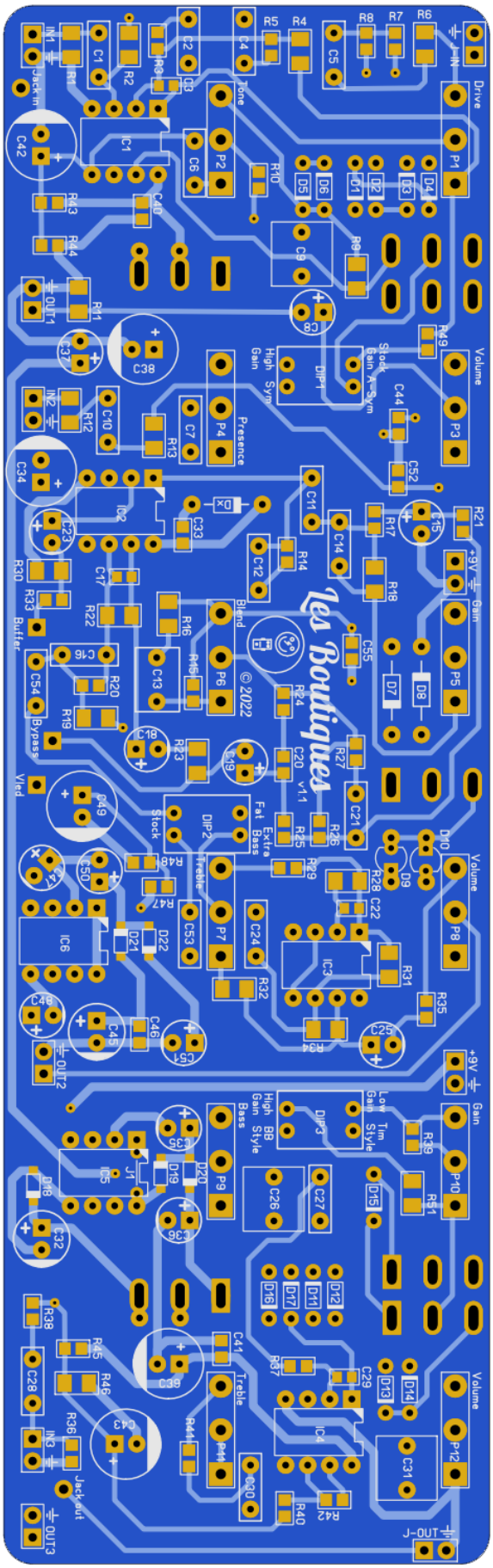


Table of contents

Read this entire manual thoroughly before you start building the effect! There are some available options and you should choose which one you want to incorporate before starting your build.

Last update: 08-09-2026

Main PCB layout



Dimensions: 56 mm x 180 mm
2.2 inch x 7.08 inch

Components Main PCB

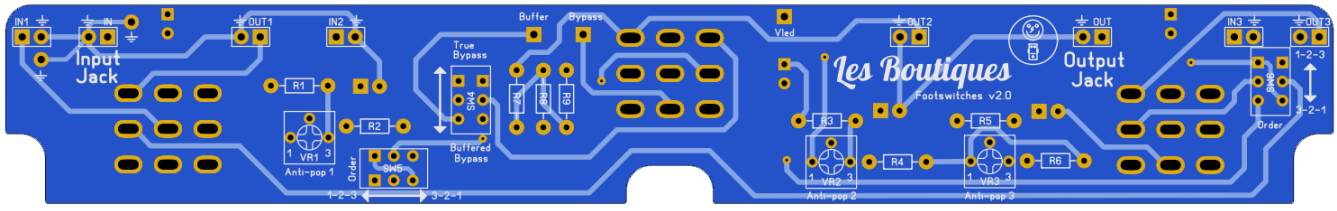
Name	Value	Comment	Package	Name	Value	Comment	Package
C1	10n		SMF/MKT/FKP2	**C48**	1u		Electrolytic 25V+
C2	10n		SMF/MKT/FKP2	**C49**	47u		Electrolytic 25V+
C4	10n		SMF/MKT/FKP2	**C50**	1u		Electrolytic 25V+
C5	100n		SMF/MKT/FKP2	**C51**	1u		Electrolytic 25V+
C6	10n		SMF/MKT/FKP2	**C53**	3n3		SMF/MKT/FKP2
C7	10n		SMF/MKT/FKP2	**C54**	68n		SMF/MKT/FKP2
C8	1u		Electrolytic 25V+	**D1**	MA858		DO-34
C9	1u		SMF/MKT/FKP2	**D2**	MA858		DO-34
C10	100n		SMF/MKT/FKP2	**D3**	MA858		DO-34
C11	100n		SMF/MKT/FKP2	**D4**	MA858		DO-34
C12	68n		SMF/MKT/FKP2	**D5**	1S1588		DO-35
C13	390n		SMF/MKT/FKP2	**D6**	1S1588		DO-35
C14	68n		SMF/MKT/FKP2	**D7**	1N34A	BAT41,OA126 V_f=0,35	DO7
C15	1u		Electrolytic 25V+	**D8**	1N34A	BAT41,OA126 V_f=0,35	DO7
C16	82n		SMF/MKT/FKP2	**D9**	1SS133	or Led 3mm	---
C18	1u		Electrolytic 25V+	**D10**	1SS133	or Led 3mm	---
C19	1u		Electrolytic 25V+	**D11**	1N4148	or 1SS133	DO-35
C21	27n		SMF/MKT/FKP2	**D12**	1N4148	or 1SS133	DO-35
C23	4u7		Electrolytic 25V+	**D13**	1N4148	or 1SS133	DO-35
C24	3n9		SMF/MKT/FKP2	**D14**	1N4148	or 1SS133	DO-35
C25	4u7		Electrolytic 25V+	**D15**	1N4148	or 1SS133	DO-35
C26	1u		SMF/MKT/FKP2	**D16**	1N4148	or 1SS133	DO-35
C27	39n		SMF/MKT/FKP2	**D17**	1N4148	or 1SS133	DO-35
C28	47n		SMF/MKT/FKP2	**DIP1**	DIP2		---
C30	10n		SMF/MKT/FKP2	**DIP2**	DIP2		---
C31	1u		SMF/MKT/FKP2	**DIP3**	DIP2		---
C32	47u		Electrolytic 25V+	**Dx**	1N4001	optional extra	DO-41
C34	47u		Electrolytic 25V+	**IC1**	JRC4580D		DIP-8
C35	10u		Electrolytic 25V+	**IC2**	TL072		DIP-8
C36	10u		Electrolytic 25V+	**IC3**	TL072		DIP-8
C37	10u		Electrolytic 25V+	**IC4**	JRC4559		DIP-8
C38	47u		Electrolytic 25V+	**IC5**	LT1054		DIP-8
C39	47u		Electrolytic 25V+	**IC6**	ICL7660S		DIP-8
C42	47u		Electrolytic 25V+				
C43	47u		Electrolytic 25V+				
C45	47u		Electrolytic 25V+				
C47	1u		Electrolytic 25V+				

All parts need to be 25V+ rated

Name	Value	Comment
P1	B250k	Drive
P2	B25k	Tone
P3	A100k	Volume
P4	B50k	Presence
P5	B100k	Gain
P6	B100k	Blend
P7	B10k	Treble
P8	B10k	Volume
P9	A50k	Bass
P10	A1M	Gain
P11	A50k	Treble
P12	A10k	Volume
SW1	DP3T	Clipping
SW2	SP3T	Clipping
SW3	DP3T	Clipping
SW4	SPDT	Headroom
SW5	SPDT	Headroom

A=Log, B=Lin, C=Rev. Log

Footswitch PCB



Part	Value			Part	Value	Comment		Part	Value	Comment
C1	10u	25V+		**R1**	390R	MF0207		**SW1**	3PDT	
C2	10u	25V+		**R2**	NC			**SW2**	3PDT	
C3	10u	25V+		**R3**	390R	MF0207		**SW3**	3PDT	
D1	Green	3mm		**R4**	1k5	MF0207		**SW4**	DPDT	
D2	Orange	3mm		**R5**	390R	MF0207		**SW5**	DPDT	
D3	Red	3mm		**R6**	4k3	MF0207		**SW6**	DPDT	
				R7	68k	MF0207		**VR1**	20k	Bourns 3362P
				R8	68k	MF0207		**VR2**	10k	Bourns 3362P
				R9	100k	MF0207		**VR3**	10k	Bourns 3362P

R1-R9, **SW4-SW6** and **VR1-VR3** should be soldered to the topside. **SW1-SW3**, **C1-C3, D1-D3** should be soldered on the bottom side.

VR1, VR2 and **VR3** vary depending on the LED you want to use for **D1, D2** and **D3**. For (ultra) bright clear 3mm LED's you can use these values:

Red : B10k (set at 6k8)
Blue : B20k (set at 15k)
Orange : B5k (set at 1k8)
Pink : B20k (set at 9k1)
White : B50k (set at 18k)
Green : B100k (set at 42k)

The brighter you turn the LED the shorter its lifespan!

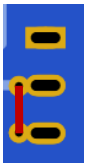
You will also need some AWG22 or AWG24 hookup wire, some shrink tubing and a **1590DD enclosure.**

SW4-SW6 are special small switches that are presoldered on the PCB.

Modifications

Headroom switches

Both the Merlin (**SW4**) and the Royal Tone (**SW5**) have an optional headroom switch. Some people do not want to incorporate the headroom switches and want to keep them the same as the stock effects. To make this easy there are 2 small extra pads added to **SW4** and **SW5**. Just leave out the desired switch and short the pads with a spare piece of leadwire.



Build sequence

Soldering this board can be very complicated for some people since the solder pads are very close together. Use a magnifying glass to make the job easier.

The trick to soldering a PCB is to work from small to big components. My building sequence suggestions in this section are based on the parts I used myself. Sometimes some components are smaller (or bigger) so always use your own common sense and change the order accordingly. Usually capacitors differ a lot in size depending on their rating and value.

Note: Do not blow on your solder in an attempt to cool it down. That can result in a bad join that might corrode! Also take extra care not to short components.

Before you start soldering, read the modifications section and decide which mod you want to incorporate.

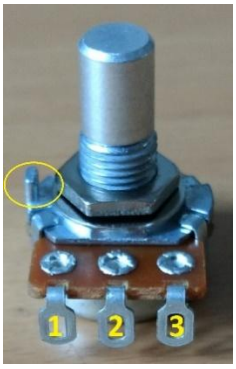
Start by soldering the resistors and jumpers (if needed). When needed you can create a jumper using a spare piece of lead from a resistor, diode or capacitor. Next come the diodes (not the LEDs).

Now is the time to solder the IC sockets on the PCB. Do not place the socketed ICs until you are finished with all soldering and off board wiring!

Now continue by soldering the internal trimpots (**VR**) then solder the SMF and MKT capacitors. Finish with soldering the LEDs and the Electrolytics.

If you are not using the drill templates, I suggest you now drill the holes in your enclosure so you can use it during the off board wiring.

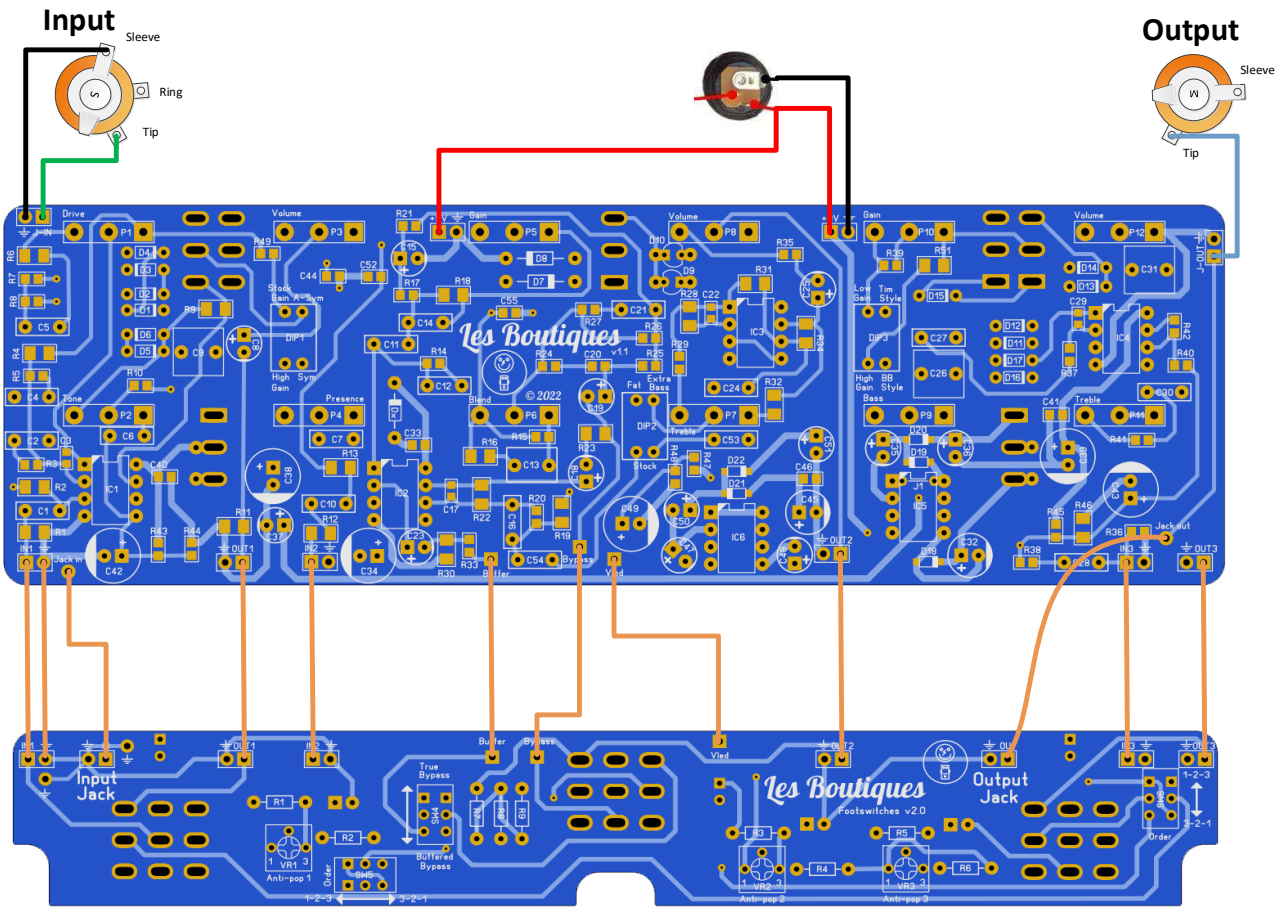
Note: Really take some time to determine where to place the pots, switches, jacks and PCB in the enclosure (1590DD) before you start drilling. Measure twice, drill once. There is a Tayda drill template if you want to use a predrilled enclosure.



The component side of the PCB is the Top side. The pots and switches need to be soldered on the bottom side of the PCB (PCB mounted pots). The rectangle pad on the PCB indicates pin 1 of the pot. You can break off the marked pin with the yellow circle using a small pair of pliers.

Off board wiring

Off board wiring is very easy using the special designed Footswitch PCB.



Note that you will need to connect both +9V pads to the + of the DC jack!

Troubleshooting

All PCB's have been 100% factory e-tested and out of every batch I receive I build an effect to double check, so there should not be a connection problem on the PCB itself.

The board is not working (at all), what now?

- Check if your 9V is plugged in correctly (and/or soldered correctly on the board). Pay special attention to the polarity.
- Check that you oriented the capacitors, IC's ,transistors and diodes the right way. SMF, MKT and ceramic capacitors as well as resistors do not need to be oriented. A likely sign of incorrect capacitors and/or orientation is when an effect is sputtering, rumbling or "motorboating".
- Check if you used the correct values of the components. For resistors you can look here: http://www.diyaudioandvideo.com/Electronics/Color/
- Double and triple check your soldering! A loose or cold solder can be really bad for your board.
- Replace the IC and/or transistors, one might be defective. Before doing that first unplug the 9V and wait for 5 seconds.
- Check that you have good/high grade components. A lot of Chinese sourced parts are fakes (especially high end opamps, audio capacitors, vintage diodes and transistors) so be careful that you source your parts from reliable suppliers.
- If you still get a lot of squealing at high gain settings, make sure you make the off board wiring as neat and as possible. Keep the wires short and do not mix the wires with high and low output signals.

Schematic