The Senator

Building instructions

v1.0

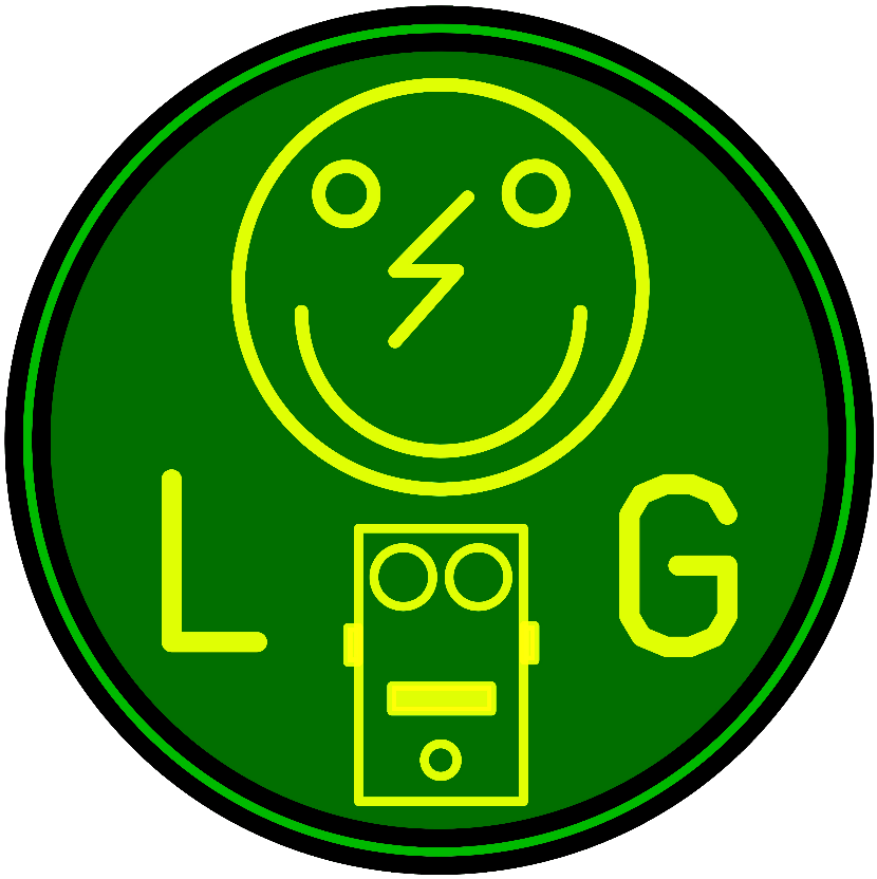
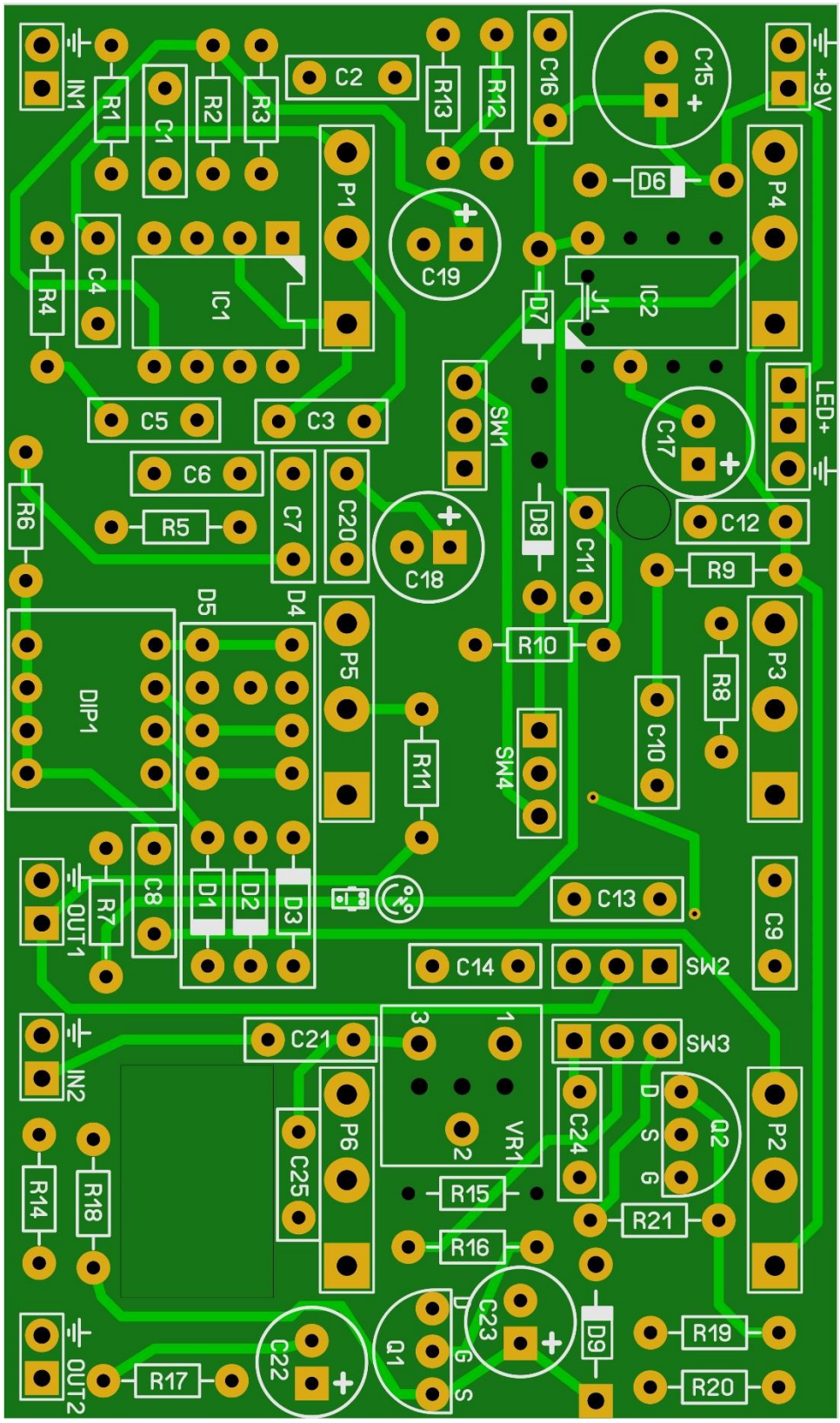


Table of contents

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: 27-03-2018

PCB layout



Dimensions: **84 mm x 49,6 mm**
3.31 inch x 1.95 inch

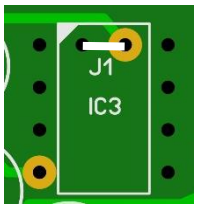
Components

Name	Value	Comment	Name	Value	Comment
C1	10n	SMF/MKT/Wima	**IC1**	TL072	see modifications
C2	100n	SMF/MKT/Wima	**IC2**	LT1054	
C3	120p	MLCC	**P1**	B100k	Gain
C4	220n	SMF/MKT/Wima	**P2**	A10k	Treble
C5	100n	SMF/MKT/Wima	**P3**	A10k	Mid
C6	220p	MLCC	**P4**	A10k	Bass
C7	220n	SMF/MKT/Wima	**P5**	B100k	Level
C8	4n7	SMF/MKT/Wima	**P6**	C25k	Boost
C9	10n	SMF/MKT/Wima	**Q1**	BS170p	
C10	220n	SMF/MKT/Wima	**Q2**	2N5457	
C11	100n	SMF/MKT/Wima	**R1**	2M2	1% metalfilm
C12	68n	SMF/MKT/Wima	**R2**	1M	1% metalfilm
C13	47p	MLCC	**R3**	2k2	1% metalfilm
C14	470p	MLCC	**R4**	10k	1% metalfilm
C15	220u	Electrolytic 25V+	**R5**	680k	1% metalfilm
C16	100n	SMF/MKT/Wima	**R6**	1k	1% metalfilm
C17	10u	Electrolytic 25V+	**R7**	1k5	1% metalfilm
C18	47u	Electrolytic 25V+	**R8**	100R	1% metalfilm
C19	100u	Electrolytic 25V+	**R9**	680R	1% metalfilm
C20	100n	SMF/MKT/Wima	**R10**	680R	1% metalfilm
C21	100n	SMF/MKT/Wima	**R11**	1k	1% metalfilm
C22	10u	Electrolytic 25V+	**R12**	47k	1% metalfilm
C23	4u7	Electrolytic 25V+	**R13**	47k	1% metalfilm
C24	100n	SMF/MKT/Wima	**R14**	2M2	1% metalfilm
C25	330p	(optional) MLCC	**R15**	10M	1% metalfilm
D1	1N4148		**R16**	10M	1% metalfilm
D2	1N4148		**R17**	100k	1% metalfilm
D3	1N4148		**R18**	5k1	1% metalfilm
D4	LED RGB	common anode	**R19**	1M	1% metalfilm
D5	LED RGB	common cathode	**R20**	1M	1% metalfilm
D6	Zener	12V	**R21**	5k1	1% metalfilm
D7	1N5817		**SW1**	see modifications	Variac1
D8	1N5817		**SW2**	SP3T	Caps
D9	Zener	9,1V or 10V	**SW3**	see modifications	Push
DIP1	Clipping	DIP4	**SW4**	see modifications	Variac2
			VR1	B100k	Pre-Gain trimpot

All parts need to be 25V+ rated **A=Log, B=Lin, C=Rev. Log**

Power section

IC2 can be either a LT1054 or a (cheaper) ICL7660S. If you want to use the 7660S then you'll need to connect both pads of J1



I do not advise to use a battery in this build as the charge pumps will do strange things when the battery is depleted. This is why it is also left out in the off board wiring section.

If you do not want to make the voltages of the effects switchable, then you have several options.

Hardwire voltages

SW1 determines the voltage for the distortion side (guvnor) and **SW4** determines the voltage of the boost side (SHO). If you connect pads 1 to pad 2 on either switch, you will feed that side with 18V. Connect pad 2 to 3 and you will feed it 9V. Pads are marked like this:

9V only

For this you can leave out **C17, C18, C20, D7, D8, IC1** and hardwire **SW4** pads 2-3 and **SW4** pads 2-3.

Build sequence

Before starting with this section, make sure you have read the modifications section first!

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 can 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.

Start by soldering the diodes **D1-D3** (if applicable), resistors and jumpers (if applicable). If needed you can create a jumper using a spare piece of lead from a resistor or diode. Next come the diodes (not the LEDs).

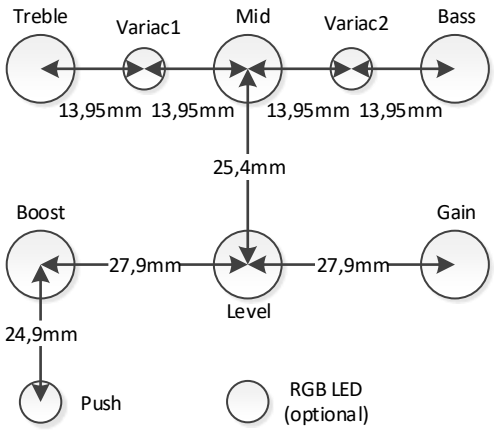
If you want to experiment with other transistors then you could socket them instead of soldering them to the board. You'll need a some 20 SIL sockets, break off the sockets and solder them to the board. Now is the time to solder these sockets on the PCB as well as the socket for the IC. Place the transistors and IC once you are finished with all soldering and off board wiring!

Now continue by soldering the MLCC, SMF/MKT/Wima capacitors then solder the internal trim pot (**VR**). Now finish with soldering the transistors (if not socketed), LEDs and the Electrolytics.

Enclosure (drilling)

I suggest you now drill the holes in your enclosure so you can use it during the off board wiring.

Here is a suggestion for drilling:



Pots need a **7 mm** hole and switches (Caps, Push, Variac) need **6 mm** holes. Diameters might differ with different brands. The optional LED hole is **5mm**. Read the modifications section to learn more about the LEDs

You are free to choose where to drill the holes for the input/output and DC jacks and the footswitches as it is very taste dependent.

Note: Really take some time to determine where and how to place the pots, switches, jacks and PCB in the enclosure before you start drilling. Measure twice, drill once.

Besides the components mentioned in the components table, you will need:

- **2 mono input jacks**.
- **2,1mm DC jack** (isolated).
- **22 gage stranded hook-up wire.**
- **2 x LED holders.** This enables you to mount the LEDs in the enclosure.
- **2 x Footswitch 3PDT** (9 pins)
- **2 x LED (3mm or 5mm depending on your taste).** These are the status LEDs
- **Hammond 1590BB** case (or similar) in your favorite color. This case will fit very tight and leaves little room for error. If you need more room you could consider using a **Hammond 1590XX**.

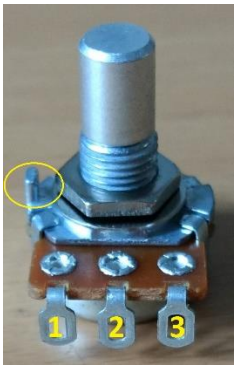
Off board wiring

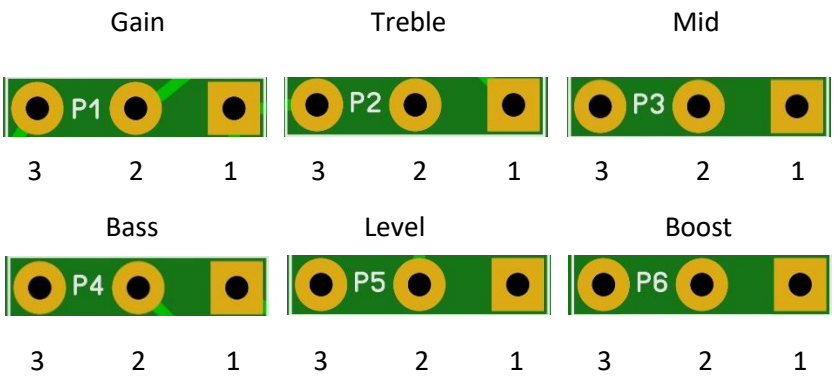
You can either mount the potentiometers directly to the PCB with special potentiometers or use the more traditional solderlug potentiometers.

Potentiometers non PCB mounted

In the pictures below you see the correct pin numbering of the pots (Alpha 16mm style).
Solder the wires accordingly and it is always a good idea to twist the wires together to create some extra shielding against external noise. The rectangle pad marks the pad for **pin 1**.

You can break off the pin I marked with the yellow circle with a small pair of pliers.

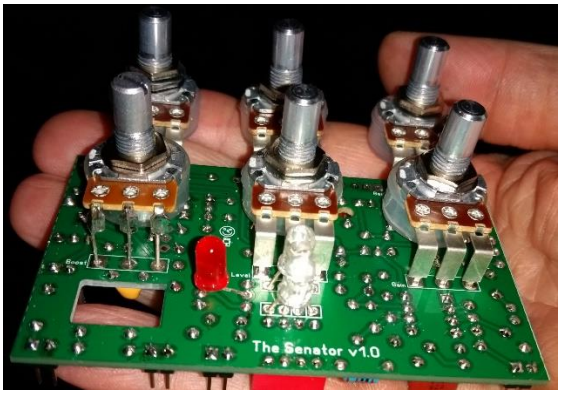




The wiring for the switches is the same, marking position 1 with a rectangle pad.

Potentiometers PCB mounted

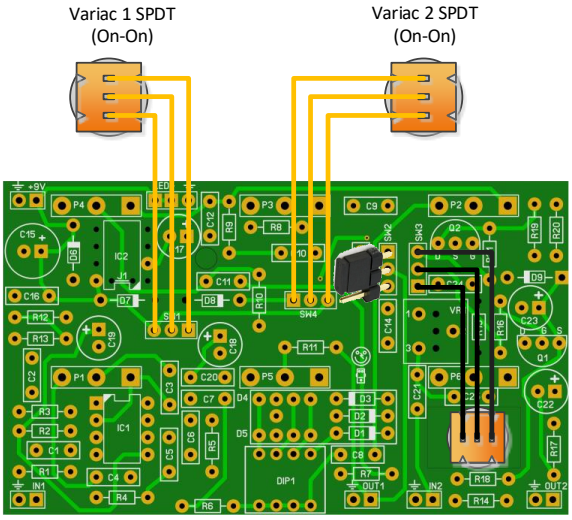
Start by mounting the potentiometers to the PCB (bottom side), but do not solder them yet! Make sure you removed the pin with a small pair of pliers and that you screwed off the nuts. Also check if the component leads on the bottom side are cut as short as possible so they will not stick out too much. While holding the PCB move it in the enclosure like this:



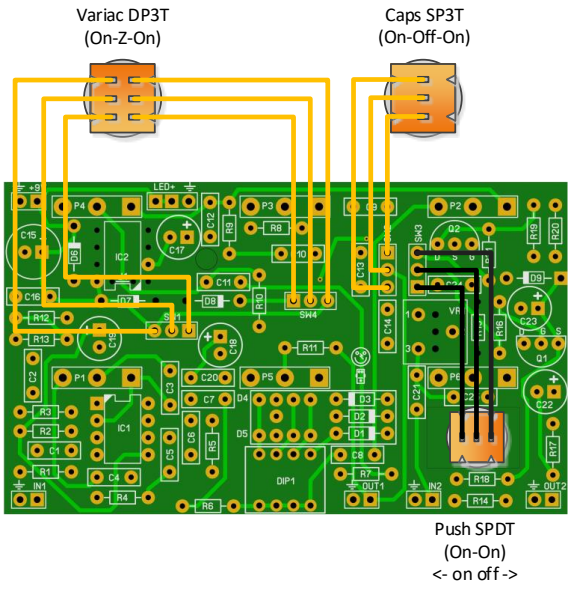
Now rotate the PCB and attach all nuts to the potentiometers and tighten them. If all fits well, solder the pots to the PCB, else adjust where needed. Make sure the bottom of the pots do not contact the PCB in any way as this might result in short circuit or other strange behavior! You can use the standard plastic caps that you can buy for the alpha pots, or tape the PCB under the pots of with some duct tape.

Switches

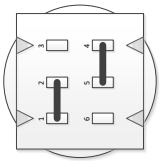
The PCB was designed so that 3 single pole switches will fit easily in the enclosure. There is a special gap made in the PCB which will fit almost all types of miniature single pole switches. There are a lot of configurations possible, and you are free to experiment. By default you can use this configuration:



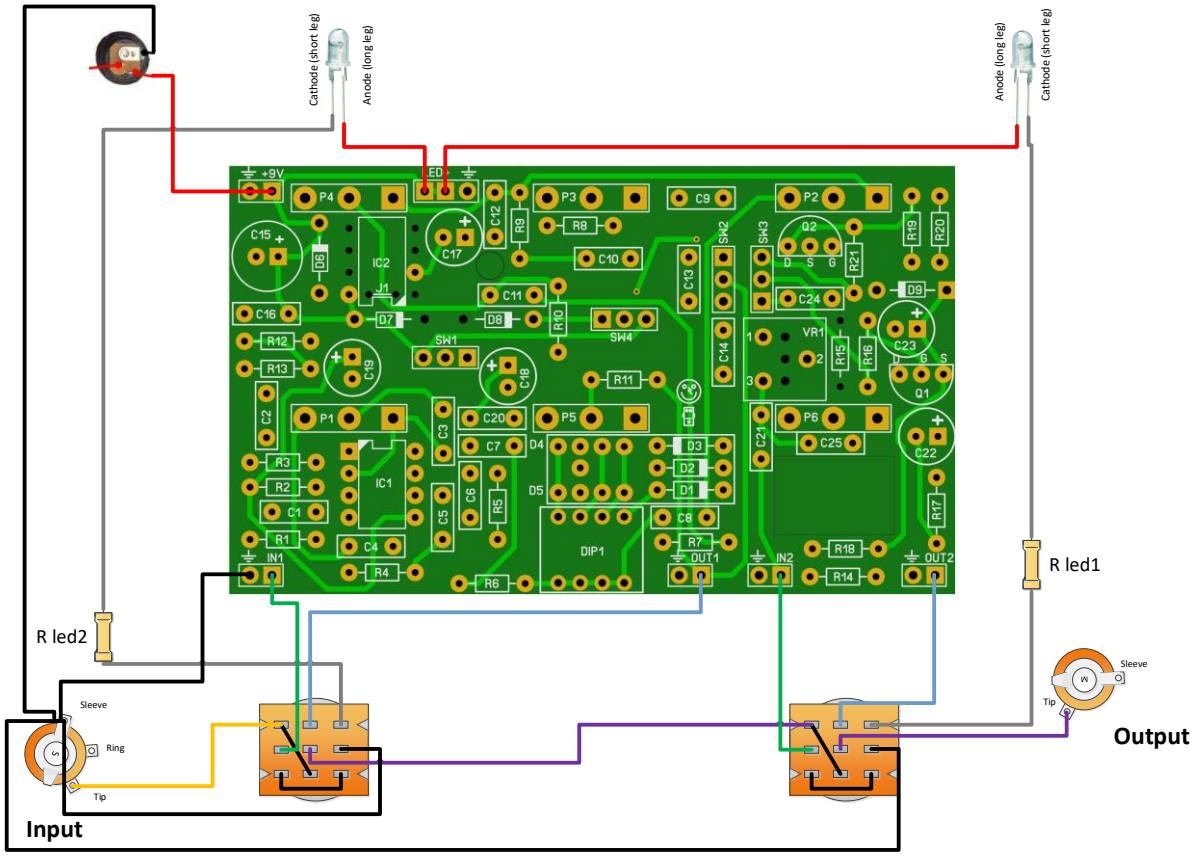
In this configuration, **SW2** (caps) is not installed. You could use a small internal SP3DT switch (ON-OFF-ON), make a 3pin header with a shunt like in the picture or hardwire it. There are other configurations possible like this one:



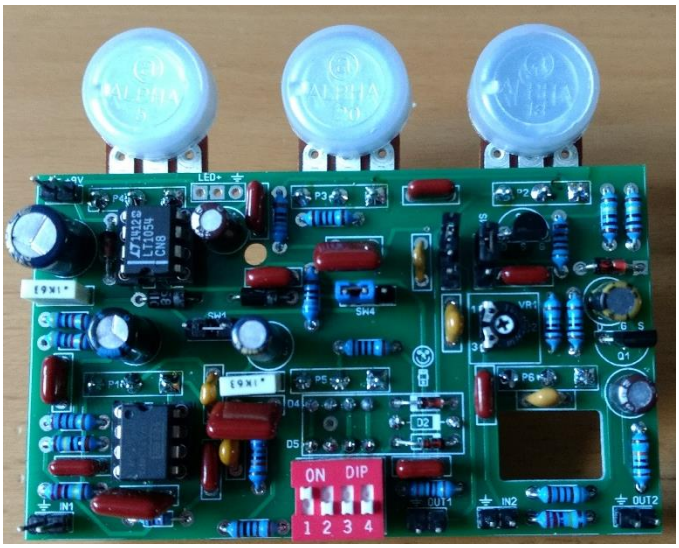
Note that there is little room between the potentiometers and when trying to fit a DP3T between them you cannot use the optional plastic cases on the potentiometers. Fit before you decide!
There are different types of DP3T On-Z-On switches. Check which contacts connect in the middle position and change the wiring if it is not to your liking!



Switch in middle position



Note that **R led1** and **R led2** are **4k7** resistors. You can change these values depending on the type of LED you use but 4k7 is safe enough for almost all LEDs @9V.



Modifications

Transistors

Q1 is based on the BS170P as used in a lot of Zvex™ pedals. You could use the regular BS170 instead of BS170P but note that the pinout is different (reversed)!

Op amp

You could use a OPA2134 instead of the TL072. Actually you can use almost any dual opamp as long as the pinout is similar to the TL072 and can handle up to +18V. Some suggestions NE5532, TL082, JRC4558P or JRC4580D. Feel free to experiment!

Clipping section

Diode section has a lot of options. Let's walk through the standard layout.

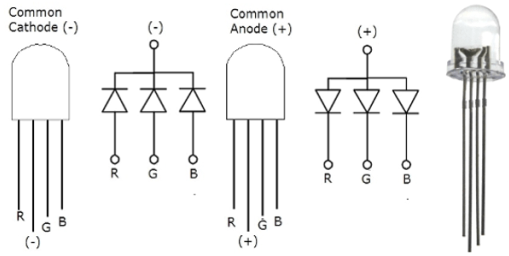
The clipping section consists of **D1-D5** and can be switched with the DIP switch **DIP1**. Switch 1 switches the red LEDs of **D4** and **D5**, Switch 2 switches the green LEDs of **D4** and **D5** and Switch 3 switches the blue LEDs of **D4** and **D5**. All options of switch 1-3 are symmetrical clipping. Switch 4 is a special one as it switches **D1**,**D2** and **D3** and uses a-symmetrical clipping.

By default **D1-D3** are 1N4148. You could consider changing **D2** to a red diffuse LED, which will make it a bit smoother. You might have to bend it a bit so it will not stick out to much.

D4 and **D5** are RGB LEDs and you need to take some time and determine how you want to place them on the PCB. The LEDs light up really nice and you might consider making them visible from the outside of the enclosure. To do this, I would advise you to insert **D5** from the bottom side of the PCB and **D4** from the top side. Make sure you do not solder **DIP1** and **D5** until you have drilled the enclosure and soldered the pots because the hole is off by 0,4mm. First solder **D4** on the component side as close as possible to the PCB with the rest of the components and when ready insert **D5** in the PCB (do not solder!!) and make it fit both the PCB and enclosure. When it fits, solder it to the PCB while it is in the enclosure. Yes, that can be difficult for some and make sure the soldering is clean as the pads are close together. Finish by soldering **DIP1**.

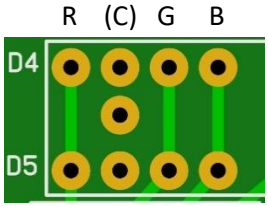
If you do not want the RGB to be visible then still solder **D4** on the topside and **D5** to the bottom side but as close as possible to the PCB. When ready, bend the LEDs (especially **D5**) so it will not stick out to much and touch the enclosure.

This is the pinout of RGB Leds:



source: arduino.cc

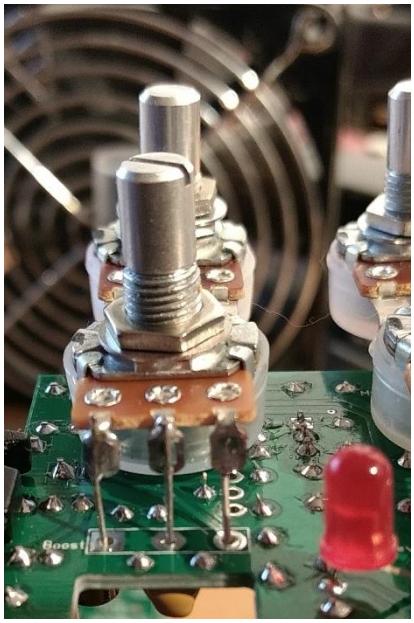
And this is the way it fits on the PCB:



C is the common, either + or –
The extra center hole is a common which goes to ground

Pots

C25k PCB mounted pots are very hard to find. You could use a regular C25k alpha pot and mount it to the PCB with some spare leads like this:



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.

Schematic