Have Mercy

Building instructions

v1.2

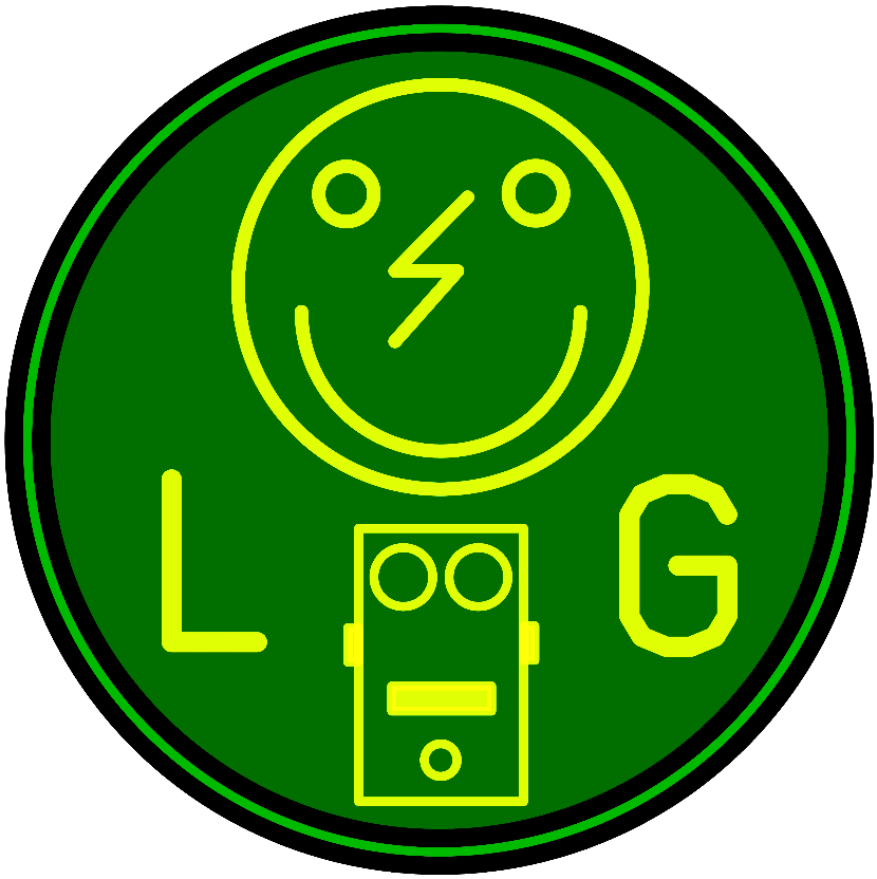


Table of contents

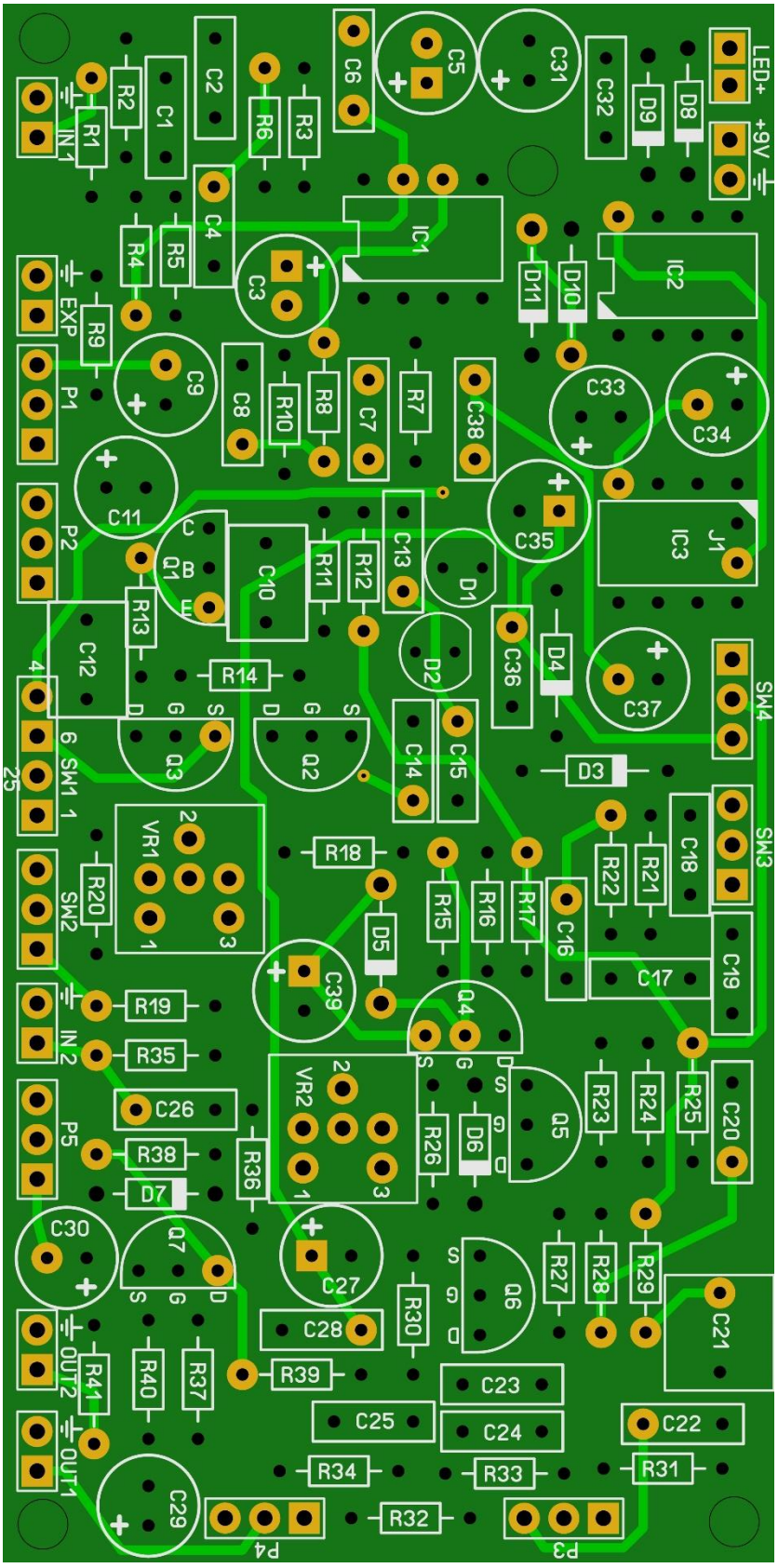
Changelog v1.2 vs v1.1:

- Minor orientation changes to parts
- Minor PCB layout optimizations

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: 18-02-2019

PCB layout



Dimensions: 50 mm x 100 mm
1.97 inch x 3.94 inch

Components

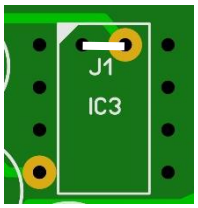
Name	Value	Comment	Name	Value	Comment	Name	Value
C1	100n	SMF	**D1**	LED Red	5mm	**R1**	68k
C2	100p	MLCC	**D2**	LED Red	5mm	**R2**	2M2
C3	33u	Electrolytic	**D3**	1N270	Germanium	**R3**	8M2
C4	100n	SMF	**D4**	1N270	Germanium	**R4**	47k
C5	100u	Electrolytic	**D5**	Zener 10V	1W	**R5**	1k
C6	100n	MKT	**D6**	Zener 10V	1W	**R6**	10k
C7	22p	MLCC	**D7**	Zener 10V	1W	**R7**	15k
C8	100n	SMF	**D8**	1N5817		**R8**	10k
C9	33u	Electrolytic	**D9**	Zener 12V	1W	**R9**	47R
C10	470n	SMF	**D10**	1N5817		**R10**	2M2
C11	47u	Electrolytic	**D11**	1N5817		**R11**	2M2
C12	470n	SMF				**R12**	3k3
C13	100n	SMF	**IC1**	OPA132		**R13**	47R
C14	47n	SMF	**IC2**	LT1054		**R14**	390R
C15	22n	SMF	**IC3**	LT1054/ICL7660S		**R15**	1M
C16	22n	SMF				**R16**	1M
C17	150p	MLCC	**P1**	B25k	Gain	**R17**	5k1
C18	470p	MLCC	**P2**	B1k	Channel Blend	**R18**	100R
C19	22n	SMF	**P3**	B100k	Tone	**R19**	1k
C20	22n	SMF	**P4**	A100k	Volume	**R20**	100R
C21	1u	SMF	**P5**	C25k	Boost	**R21**	470k
C22	10n	SMF				**R22**	22k
C23	22n	SMF	**Q1**	2N5088		**R23**	1M
C24	2n2	SMF	**Q2**	2N7000		**R24**	1M
C25	2n2	SMF	**Q3**	2N7000		**R25**	5k1
C26	100n	SMF	**Q4**	BS170P		**R26**	47R
C27*	100u	Electrolytic	**Q5**	BS170P		**R27**	1M
C28*	100n	MKT	**Q6**	BS170P		**R28**	1M
C29	10u	Electrolytic	**Q7**	BS170P		**R29**	5k1
C30	4u7	Electrolytic				**R30**	330R
C31	100u	Electrolytic	**SW1**	DP3T (ON-ON-ON)	Structure	**R31**	47k
C32	100n	MKT	**SW2**	SP3T (ON-OFF-ON)	Gain	**R32**	82k
C33	10u	Electrolytic	**SW3**	SPDT (ON-OFF)	Presence	**R33**	10k
C34	10u	Electrolytic	**SW4**	SPDT (ON-ON)	Voltage	**R34**	10k
C35	47u	Electrolytic				**R35**	1M
C36	100n	MKT	**VR1**	B10k	Set 5k	**R36***	10k
C37	47u	Electrolytic	**VR2**	B5k	Set 0k	**R37**	1M
C38	100n	MKT				**R38**	1M
C39*	22u	Electrolytic				**R39**	5k1
						R40	5k1
						R41	100k

* Optional parts.

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

Power section

IC3 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 depleting. This is why it is also left out in the off board wiring section.

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 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 resistors and jumpers (if needed). 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 SMF and MKT capacitors then solder the internal trimpots (**VR**). Now finish with soldering the transistors (if not socketed), LEDs and the Electrolytics.

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 before you start drilling. Measure twice, drill once.

You are almost ready to rock, well… not really. The difficult part starts now.

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

- **3 mono input jacks**. **2 x 3PDT footswitch** (9 pins)
- **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 LED (3mm or 5mm depending on your taste).** These are the status LEDs
- **Hammond 1590XX** case (or similar) in your favorite color.

Calibration

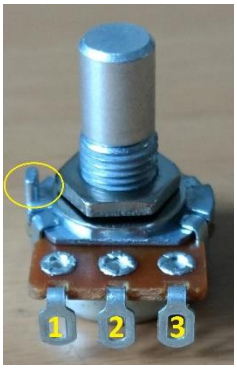
Once everything is connected, you should calibrate **VR1** and **VR2** to give you your optimum gain. Start by setting **VR1** to 12 o'clock (50%) and VR2 all the way to the left (0%). Set **P1** (Gain) to about 2 o'clock (70%), the gain switch **SW2** in the middle position (low gain) and the voltage switch **SW4** to 9V. This should give you a nice crunch sound. Now test setting **P1** (Gain) to max and try all settings on the gain switch **SW2** to see if you like all positions. If not, you can add more gain by turning **VR1** to the right, or subtract gain by turning VR1 to the left. You can also lower the gain by turning **VR2** to the left. Please note that **VR1** and **VR2** communicate. Changing one will affect the function of the other.

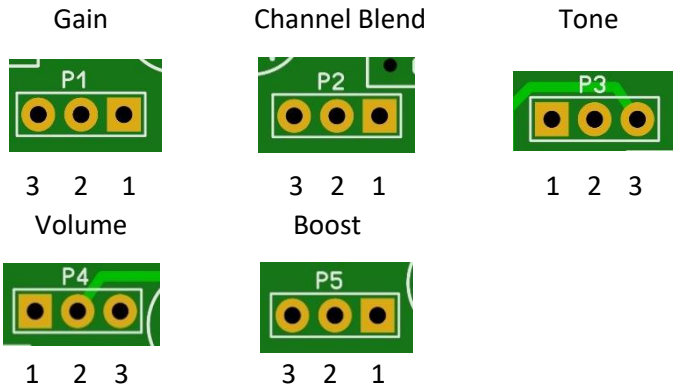
Off board wiring

Potentiometers

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**.

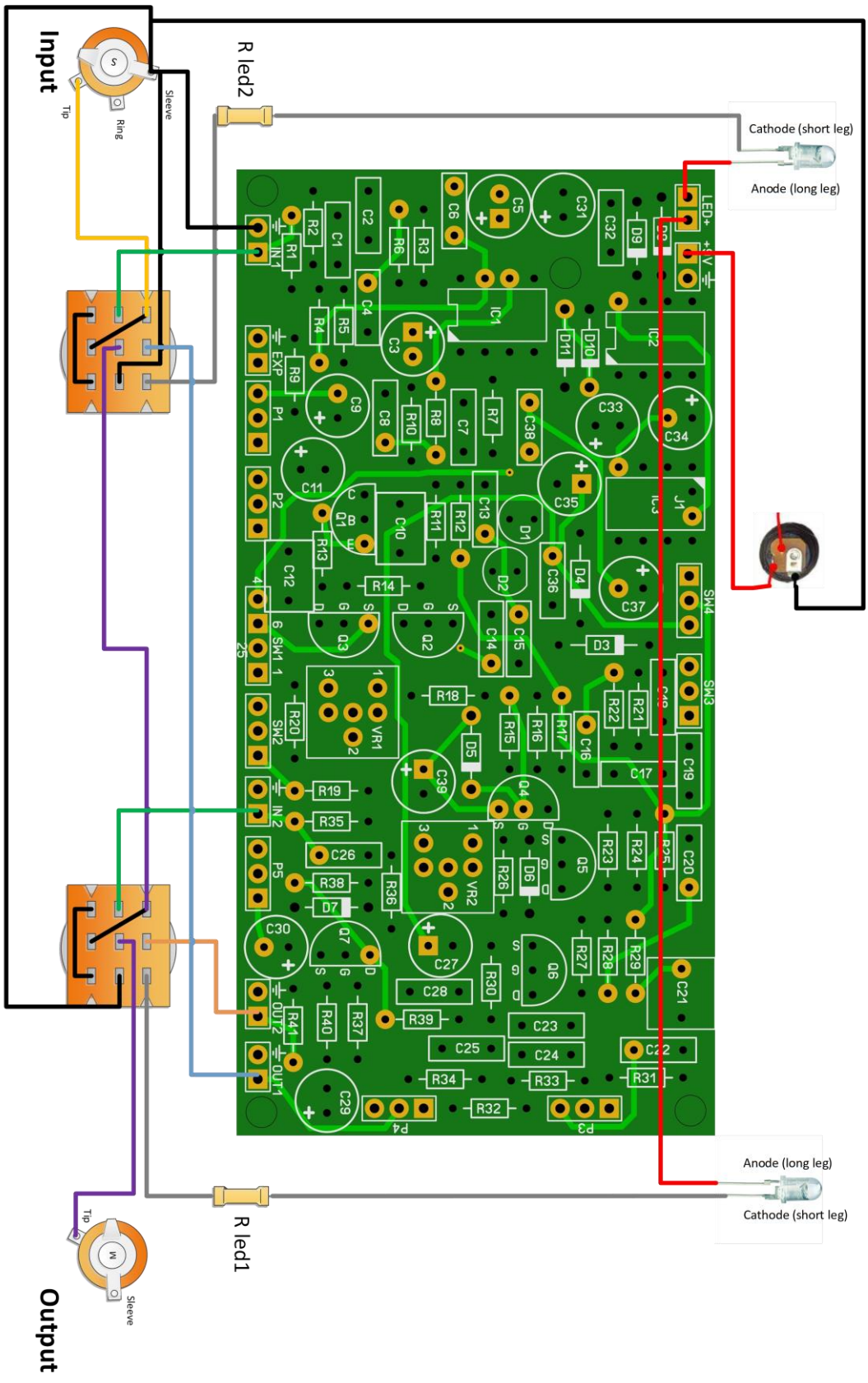
You can break off the pin I marked with the yellow circle with a small pair of pliers.



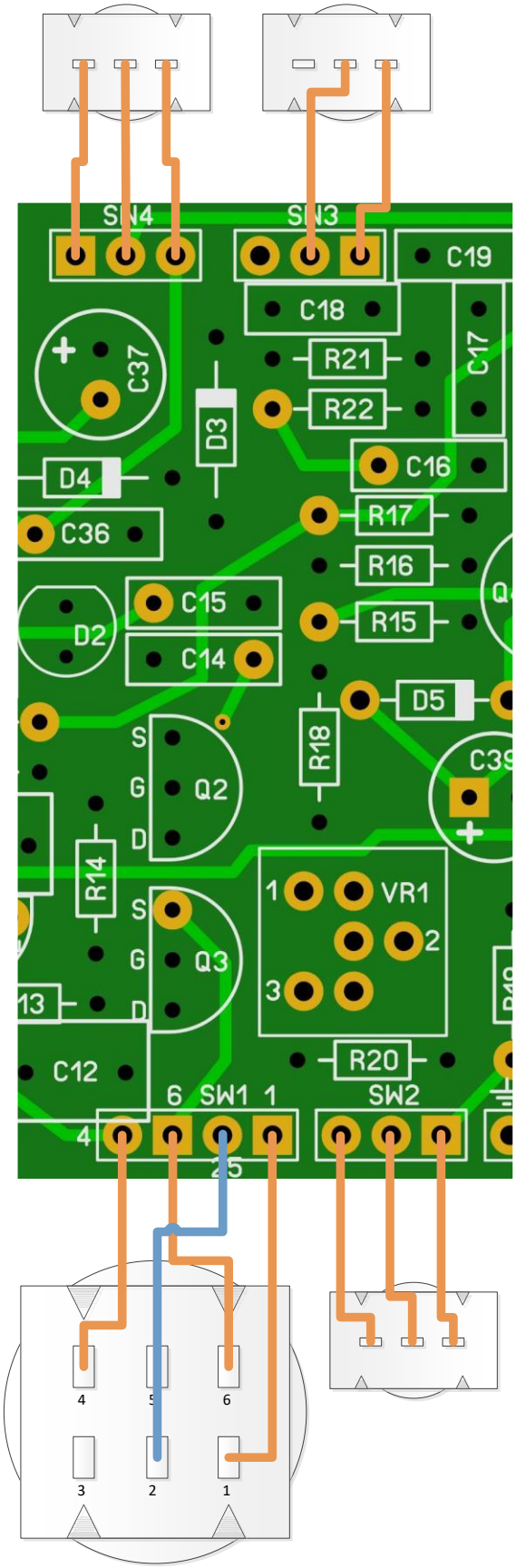


The wiring for the switches is the same, marking position 1 with a rectangle pad.

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.



Switches



Modifications

Transistors

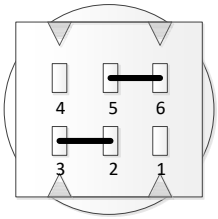
Q4 to **Q7** are based on the BS170P as used in a lot of Zvex pedals. You can of course use the regular BS170 instead of BS170P but note that the pinout is different! Soundwise they are exactly the same.

Op amp and charge pump

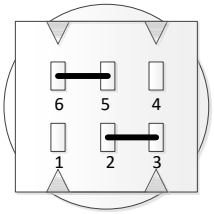
You could use a OPA134 instead of the OPA132. It is available in DIP8 and is often cheaper while it makes no audible tonal difference. Note that TI does not make OPA132 in DIP8 so if you find them on Ebay or AliExpress, they are fake! If you come to think of it, you could use almost any pin compatible single op amp chip although I can't guarantee there will not be tonal differences and it must be able to take -9V (Vcc-) and +9V (Vcc+). You could also buy a OPA132 (SMD) and a converter board to keep true to the original.

Clipping section

Diode section can be wired in a lot of different ways. Note that the designated pins of a ON-Z-ON are marked on the PCB and correspond to the following switches depicted in the middle position (so **do not** think you need to wire pin 2 to 3 and 6 to 5) :



Or:
(depending on the type of switch)



You can connect pin 2 and 5 together on the switch and then connect one of those pins with a wire to the pad marked 25. The rest is straight forward. Pin 1 to the pad marked 1, pin 4 to the pad marked 4 and pin 6 to the pad marked 6 (pin 3 is not connected). You can always connect it another way if you want different options or use different diodes/transistors, just remember that the pad marked 25 is for the common pins (middle pins) of the switch. Remember to check the forwarding voltages of the clipping components when making your choice.

Pot values and resistors

The Gain pot (**P1**) is reportedly better off using a A50k instead of a B25k. Same goes for Channel Blend (**P2**) using a C1k instead of a B1k. An A25K for Boost (**P5**) is also ok.

It is also reported that leaving out R10 will do the gain some good.

Hissing and squealing

As this is a very high gain effect, using the gain switch will introduce some extra hissing and possibly squealing. For this I added an optional filter capacitor **C39** and it is reported that using a 22uF in there will lessen the squealing. If you do not use it then place a jumper in **C39**! For this I also added **VR1** and **VR2**. With those trim pots you can lower the gain on both stages. The original only uses a 5k1 in **R18** and **R26** is a 47R. If you want to use it as in the original then leave out **VR1** and **VR2** and use a jumper between pad 1 and 2 of both **VR1** and **VR2**. Else set both **VR1** and **VR2** to 5k. Another option is to replace **R20** with a 330R or 470R. A little less high gain but also less squealing.

Extra noise and pop protection in boost section

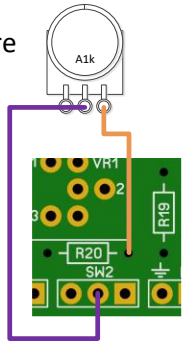
The effect originally does not have **R36**. It was added to prevent popping and hissing when turning down/off the volume on your guitar. You could leave it out and solder a simple jumper instead.

You can also experiment with the values of almost all components in the boost. For example you could lower **R36** to 1k or 100R or **R37** and **R38** to 10M like in the Zvex Super Hard On™. You could also make **R37** and **R38** non symmetrical values. For example **R37** 4M7 and **R38** 10M. You will notice the tone structure will change.

The optional **C27** and **C28** are added if you experience to much hiss on the Boost side due to the power supply.

Change gain switch to potentiometer

The gain switch lets you switch either **R19** (1k) or **R20** (100R) to ground. If you want more control and options you could use a A1k instead and leave out **R19**,**R20** and **SW2**.



Connect pin 2 and 3 together to the middle pad of **SW2** and pin 1 to the right side of **R20**. Note that with this configuration, you will be missing the middle position of the original switch. You could again add a SPDT switch that connects/disconnects the purple wire.

Conversion

The original Bogner La Grange is a combination of multiple modified Zvex™ and Lovepedal™ effects (Super Hard On™ Boost, Box of Rock™ and Clock of Tone 50™). You can leave out and change some components to convert to those effects.

SHO™

1. Leave out **C30**, **R35**, **R40**
2. Short both pads on the **C30**
3. Change **P5** to C5K and both **R37** and **R38** to 10M

Box of Rock™

1. Leave out **C17**, **C19**, **C39**, **D6**, **R18**, **R19**, **R20**, **R22**, **SW2**, **SW3**, **VR1**, **VR2**
2. Substitute **VR1** for an external Alpha Pot **C5K**
3. Change **R26** to 100R
4. Short pad 1 and 2 on **VR2**, pad 1 and 2 on **SW3**
5. Short the pads of **C19** and **R18**

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