

Have Mercy

Building instructions

v2.0

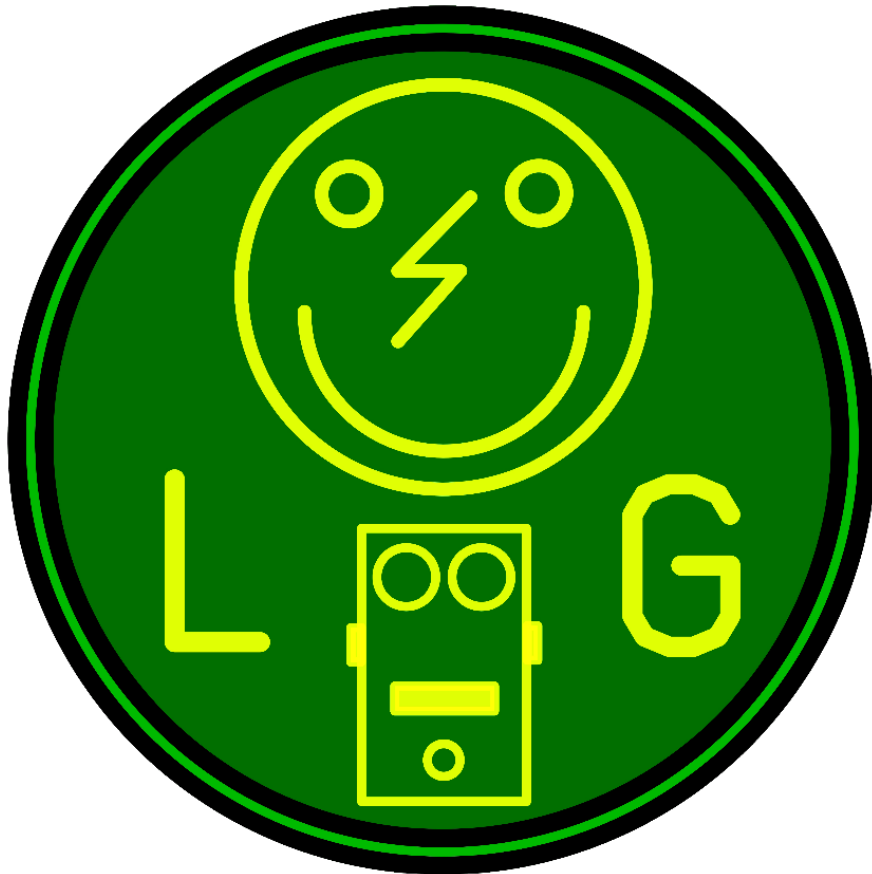




Table of contents

.....	2
PCB layout	3
Components main PCB	4
Bill of Materials.....	5
Power section	6
Build sequence	6
Calibrating to stock factory settings.....	8
Switch settings.....	8
Off board wiring	9
Modifications.....	10
Transistors	10
Op amp and charge pump.....	10
Pot values and resistors.....	10
Hissing and squealing	10
Extra DC noise filters	10
Change gain switch to potentiometer.....	10
COT high frequencies filter.....	10
Boost section	11
Conversion to Box of Rock™.....	11
Incorporating an expression pedal.....	11
Troubleshooting	12
Schematic	13

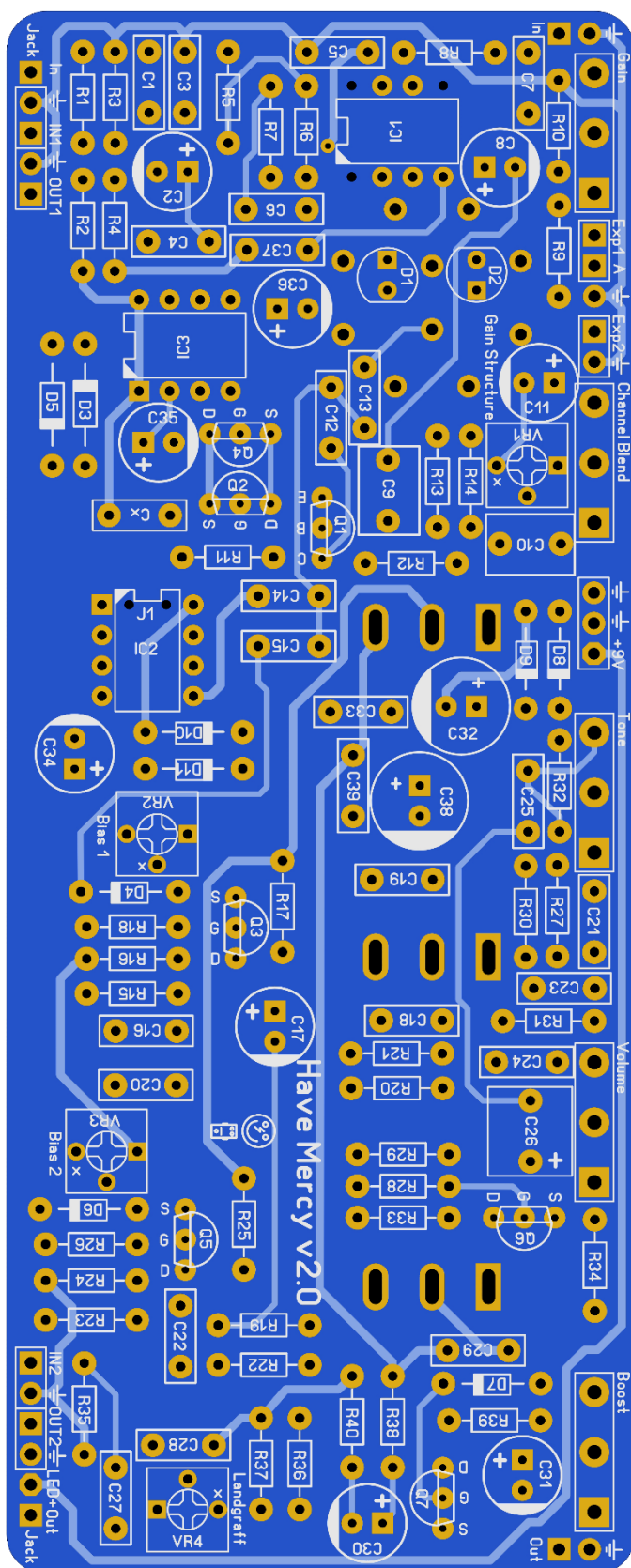
Read this entire manual thoroughly before you start building the effect! There are some available options and you should choose which one you want to incorporate before starting your build.

Last update: 06-05-2023



Have Mercy v2.0

PCB layout



Dimensions: 52,5 mm x 130 mm
2.1 inch x 5.15 inch

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Components main PCB

Name	Value	Comment
C1	100n	SMF/MKT/FKP2
C2	33u	Electrolytic 25V+
C3	100p	SMF/MKT/FKP2
C4	100n	SMF/MKT/FKP2
C5	100n	SMF/MKT/FKP2
C6	22p	SMF/MKT/FKP2
C7	100n	SMF/MKT/FKP2
C8	33u	Electrolytic 25V+
C9	470n	SMF/MKT/FKP2
C10	470n	SMF/MKT/FKP2
C11	47u	Electrolytic 25V+
C12	100n	SMF/MKT/FKP2
C13	47n	SMF/MKT/FKP2
C14	47n	SMF/MKT/FKP2
C15	22n	SMF/MKT/FKP2
C16	22n	SMF/MKT/FKP2
C17	22u	Electrolytic 25V+
C18	150p	SMF/MKT/FKP2
C19	470p	SMF/MKT/FKP2
C20	22n	SMF/MKT/FKP2
C21	22n	SMF/MKT/FKP2
C22	22n	SMF/MKT/FKP2
C23	2n2	SMF/MKT/FKP2
C24	2n2	SMF/MKT/FKP2
C25	10n	SMF/MKT/FKP2
C26	1u	SMF/MKT/FKP2
C27	100n	SMF/MKT/FKP2
C28	47p	SMF/MKT/FKP2
C29	100n	SMF/MKT/FKP2
C30	10u	Electrolytic 25V+
C31	4u7	Electrolytic 25V+
C32	100u	Electrolytic 25V+
C33	100n	SMF/MKT/FKP2
C34	10u	Electrolytic 25V+
C35	10u	Electrolytic 25V+
C36	47u	Electrolytic 25V+
C37	100n	SMF/MKT/FKP2
C38	47u	Electrolytic 25V+
C39	100n	SMF/MKT/FKP2
Cx	100n	SMF/MKT/FKP2

Name	Value	Comment
D1	LED	5mm red
D2	LED	5mm red
D3	1N270	DO-7
D4	Zener 10V	
D5	1N270	DO-7
D6	Zener 10V	
D7	Zener 10V	
D8	1N5817	
D9	Zener 12V	
D10	1N5817	
D11	1N5817	
IC1	OPA132	
IC2	LT1054	
IC3	ICL7660S	
P1	B25k	Alpha 16 mm pots
P2	B1k	Alpha 16 mm pots
P3	B100k	Alpha 16 mm pots
P4	A100k	Alpha 16 mm pots
P5	C25k	Alpha 16 mm pots
Q1	2N5088	
Q2	2N7000	
Q3	BS170p	
Q4	2N7000	
Q5	BS170p	
Q6	BS170p	
Q7	BS170p	
SW1	2P4T	Rotary switch
SW2	SP3T	on-off-on
SW3	SP1T	on-off
SW4	SPDT	on-on
VR1	B1k	---
VR2	B10k	---
VR3	B1k	---
VR4	B10k	---

Name	Value	Comment
R1	68k	MF0207
R2	47k	MF0207
R3	2M2	MF0207
R4	1k	MF0207
R5	8M2	MF0207
R6	10k	MF0207
R7	15k	MF0207
R8	10k	MF0207
R9	47R	MF0207
R10	2M2	MF0207
R11	2M2	MF0207
R12	3k3	MF0207
R13	47R	MF0207
R14	390R	MF0207
R15	1M	MF0207
R16	1M	MF0207
R17	5k1	MF0207
R18	100R	MF0207
R19	1k	MF0207
R20	470k	MF0207
R21	22k	MF0207
R22	100R	MF0207
R23	1M	MF0207
R24	1M	MF0207
R25	5k1	MF0207
R26	47R	MF0207
R27	47k	MF0207
R28	1M	MF0207
R29	1M	MF0207
R30	10k	MF0207
R31	10k	MF0207
R32	82k	MF0207
R33	5k1	MF0207
R34	330R	MF0207
R35	1M	MF0207
R36	1M	MF0207
R37	1M	MF0207
R38	5k1	MF0207
R39	5k1	MF0207
R40	100k	MF0207

Bill of materials
All parts need to be 25V+ rated

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

Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.



Bill of Materials

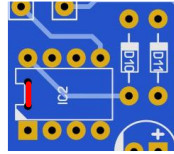
Value	Amount	Package
Capacitors		
100n	10	SMF/MKT/FKP2
100p	1	SMF/MKT/FKP2
100u	1	Electrolytic 25V+
10n	1	SMF/MKT/FKP2
10u	5	Electrolytic 25V+
150p	1	SMF/MKT/FKP2
1u	1	SMF/MKT/FKP2
22n	5	SMF/MKT/FKP2
22p	1	SMF/MKT/FKP2
22u	1	Electrolytic 25V+
2n2	2	SMF/MKT/FKP2
33u	2	Electrolytic 25V+
470n	2	SMF/MKT/FKP2
470p	1	SMF/MKT/FKP2
47n	2	SMF/MKT/FKP2
47p	1	SMF/MKT/FKP2
47u	3	Electrolytic 25V+
4u7	1	Electrolytic 25V+
100n	1	SMF/MKT/FKP2
Diodes		
1N270	2	
1N5817	3	
LED	2	5 mm Red for clipping
LED	2	3mm for footswitch
Zener 10V	3	
Zener 12V	1	
Ics		
ICL7660S	1	
LT1054	1	
OPA132	1	
Pots		
A100k	1	Alpha PCB 16mm pot
B100k	1	Alpha PCB 16mm pot
B1k	1	Alpha PCB 16mm pot
B25k	1	Alpha PCB 16mm pot
C25k	1	Alpha PCB 16mm pot

Value	Amount	Package
Transistors		
2N5088	1	
2N7000	2	
BS170p	4	BS170 has reversed pinout
Resistors		
100k	1	MF0207
100R	2	MF0207
10k	4	MF0207
15k	1	MF0207
1k	2	MF0207
1M	9	MF0207
22k	1	MF0207
2M2	3	MF0207
330R	1	MF0207
390R	3	MF0207
3k3	1	MF0207
470k	1	MF0207
47k	2	MF0207
47R	3	MF0207
5k1	6	MF0207
68k	1	MF0207
82k	1	MF0207
8M2	1	MF0207
Switches		
2P4T	1	Tayda RS16211
SP1T	1	Dailywell 1M series
SP3T	1	Dailywell 1M series
SPDT	1	Dailywell 1M series
3PDT	2	Footswitch
Trim pots		
B10k	4	Bourns 3362P
B1k	2	Bourns 3362P



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 depleting.

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. You must read the part about incorporating the expression pedal else you might get into trouble at the end of your build!

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 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. Do not place the transistors and IC 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. 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. 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 indicates pin 1 of the pot. You can break off the pin I marked with the yellow circle using a small pair of pliers.



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

- **1 x mono jacks**
- **1 x stereo jack**
- **2,1mm DC jack** (isolated).
- **22 gage stranded hook-up wire**
- **2 x LED holders for 3mm LEDs.** This enables you to mount the LEDs in the enclosure.
- **Hammond 1590XX** case in your favorite color.

If you are planning on incorporating the expression pedal, you will need an extra jack as mentioned in that section.



Calibrating to stock factory settings

Once everything is connected, you should calibrate **VR1 – VR4** to give you your optimum gain.

VR1 (Gain Structure) : **100%** (full clockwise/right)

VR2 (Bias 1) : **50%** (12 o'clock/middle)

VR3 (Bias 2) : **100%** (full clockwise/right)

VR4 (Landgraff) : **100%** (full clockwise/right)

These are the stock factory setting for the original pedal.

VR1 will subtract some gain when lowered on the COT part of the pedal. **VR2** can add or subtract some gain on the 1st stage and **VR3** will subtract some gain of the 2nd stage when lowered.

VR4 is a sort of pre gain for the boost part as used in the original Landgraff design.

This is a very loud effect and prone to squeal at high gain. Adjust the settings of especially **VR1-VR3** as you desire to tame the beast. If you think you are lost in the calibration, just return the settings to the default settings and start over.

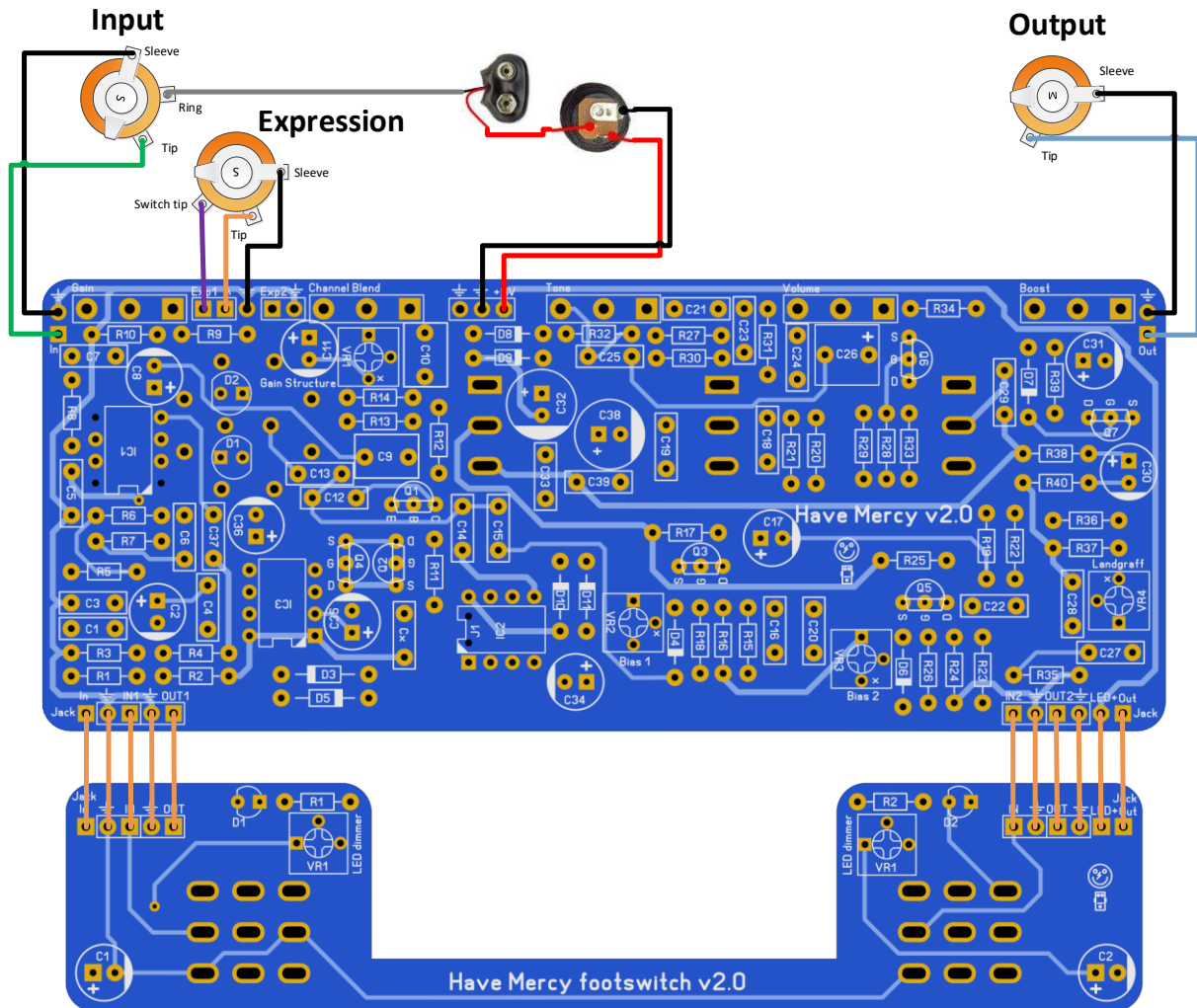
Switch settings

Setting	Structure	Setting	Gain	Variac	Presence
1	None	Top	High	Off	High
2	LEDs	Middle	Low		
3	Mosfet	Bottom	Middle	On	Low
4	Germanium				



Off board wiring

Off board wiring is very easy using the special designed Footswitch PCB. It fits a 9V battery however be warned that a beeping battery will not be able to deliver the 9V needed for the voltage doubler to work and produce 18V. I would recommend to use a good 9V adapter instead.



The footswitch PCB requires 2x 3PDT footswitches (**SW1** and **SW2**), 2x 390R resistors (**R1** and **R2**), 2x 3mm led of your choice for the status indicators (**D1** and **D2**), 2x 10uF caps (10V+) (**C1** and **C2**) and 2x 10k trimpots (**VR1** and **VR2**). All required parts are included in the BoM.

Using the trimpots you can dim the LEDs or brighten them as much as you like. Start by setting the trimpots **VR1** and **VR2** on the footswitch PCB to about half way (50%) and work from there. A bright Blue LED might even need a higher valued trimpot of 50k instead of 10k.

The brighter you turn the LED the shorter its lifespan!

The Footswitch PCB also incorporates the [AMZfx anti LED popping](#) circuit by means of C1 and C2.



Modifications

Transistors

Q3, Q5, Q6 and 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 in reverse! 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 difference. Note that TI does not make OPA132 in DIP8 so if you find them on Ebay or AliExpress, they are fakes! you can use almost any pin compatible single opamp chip as long as it is able to take -9V (Vcc-) and +18V (Vcc+). You could also buy a OPA132 (SMD) and a converter board to keep true to the original.

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, however it has very little effect as it is in parallel with the lower resistance gain pot and thus not very effective.

Hissing and squealing

As this is a very high gain effect, using the gain switch will introduce some extra hissing and squealing. I added an optional filter capacitor **C17** and it is reported that using a 22uF in there will lessen the squealing. If you do not use it then solder a jumper in **C17**! You can also consider changing the value of **R22** to something between 200R to 500R which will still give you high gain.

C28 is an optional cap for radiofrequency filtering in the boost section. ([Jack Orman Mosfet Booster](#))

Also not that the quality of the charge pumps is vital. Low cost knock offs will introduce more noise!

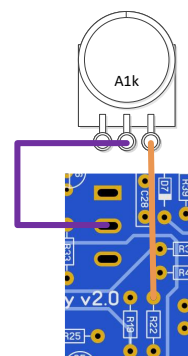
Extra DC noise filters

The optional **Cx** and **C29** are added if you experience too much hiss due to the power supply noise.

Change gain switch to potentiometer

The gain switch lets you switch either **R19** (1k) or **R22** (100R) to ground. If you want more control and options you could use a A1k instead and leave out **R19**, **R22** and **SW2**.

Connect pin 1 and 2 together to the middle pad of **SW2** and pin 3 to the right side of **R22**. 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.



COT high frequencies filter

Some people do not like the highs in the COT. That is why it has an extra 47n cap in the church of Tone™. In the original La Grange™ this cap is moved in series with the diodes (**C13**). If you perceive glassy highs, add **C14** to the board and optionally jumper **C13**.



Boost section

P5	R39	C31	R35	R36	R37	Configuration
C25k	5k1	4u7	1M	1M	1M	Original default
C20k	5k1	4u7	1M	1M	1M	Simpler pot sourcing
B25k	5k1	4u7	1M	1M	1M	Even simpler pot sourcing
C20k	5k1	jump	Omit	10M	10M	Simpler pot sourcing tuned to SHO™
C5k	omit	jump	Omit	10M	10M	SHO™

Altering the value of **P5** has little influence on the amount of boost. Since it is in parallel to R39 the equivalent resistance fluctuates between 4,63k using a 50k, 4,24k using a 25k down to 4,06k using a 20k pot.

Conversion to Box of Rock™

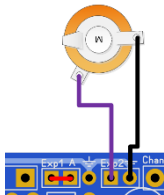
The original Bogner La Grange is a combination of multiple modified Zvex™ and Lovepedal™ effects (Super Hard On™ Boost, Box of Rock™ and Clock of Tone 50™).

1. Leave out **C17**, **C19**, **C20**, **C39**, **D6**, **R18**, **R19**, **R21**, **R22**, **SW2**, **SW3**, **VR2**, **VR3**
2. Substitute **VR2** for an external Alpha Pot **C5K**
3. Change **R26** to 100R, **C15** to 100n, **C18** to 470p, **C26** to a 1uF electrolytic (+ marked on PCB)
4. Short pad 3 and 2 on **VR3**
5. Short the pads **C20**, **R18** and **R21**

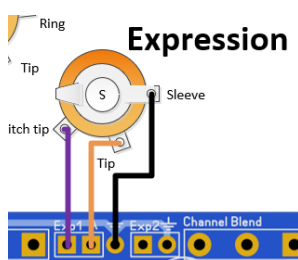
Incorporating an expression pedal

There are 2 ways to incorporate an expression pedal. The first being as mentioned in the original pedal. The original adds a expressionpedal parallel to the gainpot in the pedal. This results in around 12.5k ($25k // 25k // 2M2 = \text{approx. } 12.5k$) and so makes the gain act a bit different.

To wire this configuration you should jumper Exp1 and A and add a monojack to Exp2 and ground like this:



The other way is to replace the internal gain pot when inserting the expressionpedal. This gives you more usable gain options when using different resistor values as it will use the full extent of the pot taper. This can be wired like this with a switchable jack ([for example this](#)):



When you do not want to incorporate an expression pedal, you will need to jumper the Exp1 and A pads.



Troubleshooting

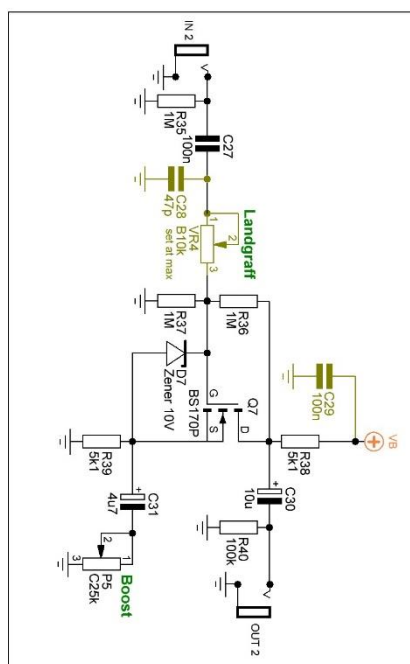
All PCB's have been 100% factory e-tested and out of every batch I receive I build an effect to double check, so there should not be a connection problem on the PCB itself.

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

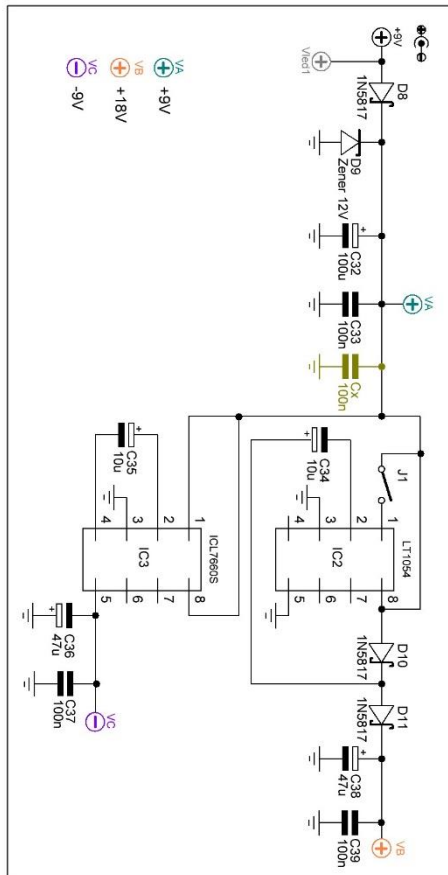
- Check if your 9V is plugged in correctly (and/or soldered correctly on the board). Pay special attention to the polarity.
- Check that you oriented the capacitors, IC's ,transistors and diodes the right way. SMF, MKT and ceramic capacitors as well as resistors do not need to be oriented. A likely sign 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

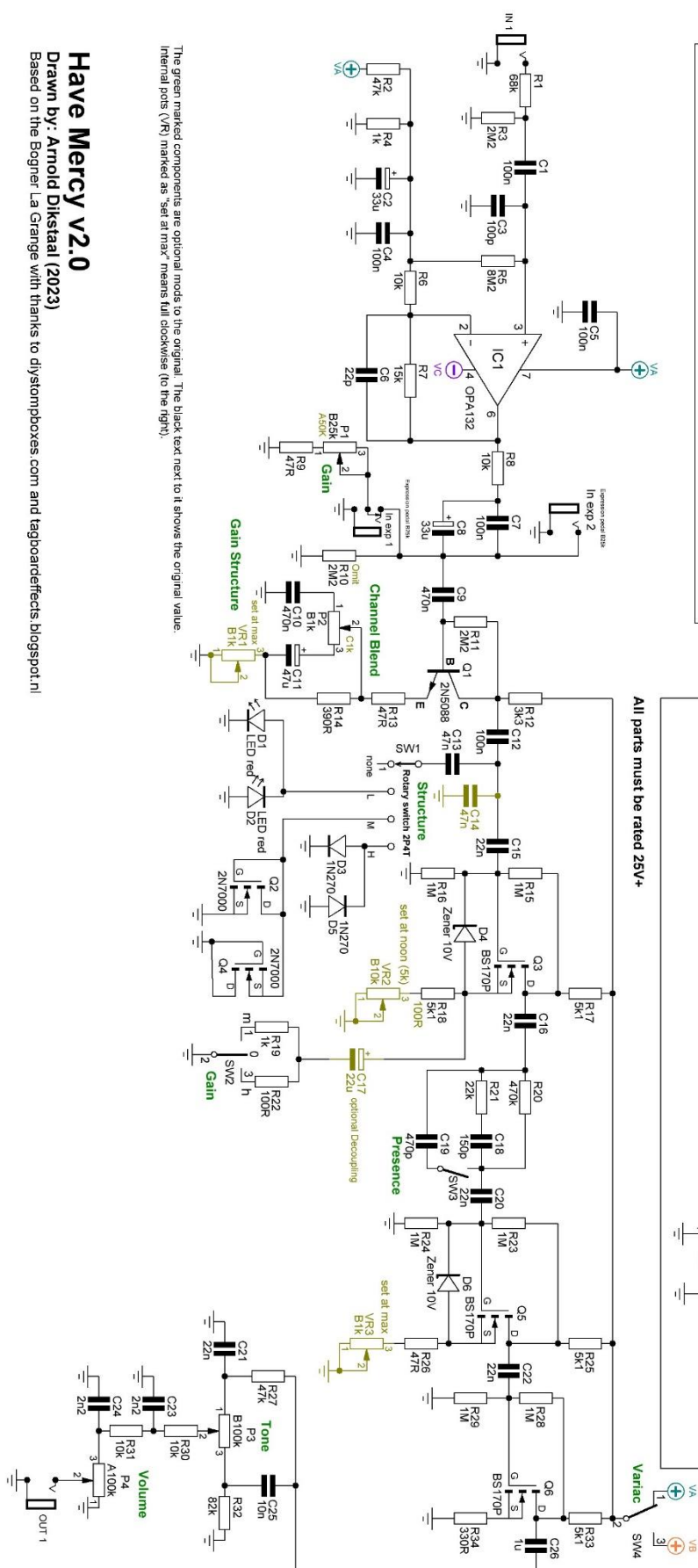
Clean Boost (Post OD)



Power



All parts must be rated 25V+



The green marked components are optional mods to the original. The black text next to it shows the original value. Internal pots (VR) marked as "set at max" means full clockwise (to the right).

Have Mercy v2.0

Drawn by: Arnold Dikstaal (2023)

Based on the Bogner La Grange with thanks to diystompboxes.com and tagboardeffects.blogspot.nl