Divine Fuzz

Building instructions

V1.0

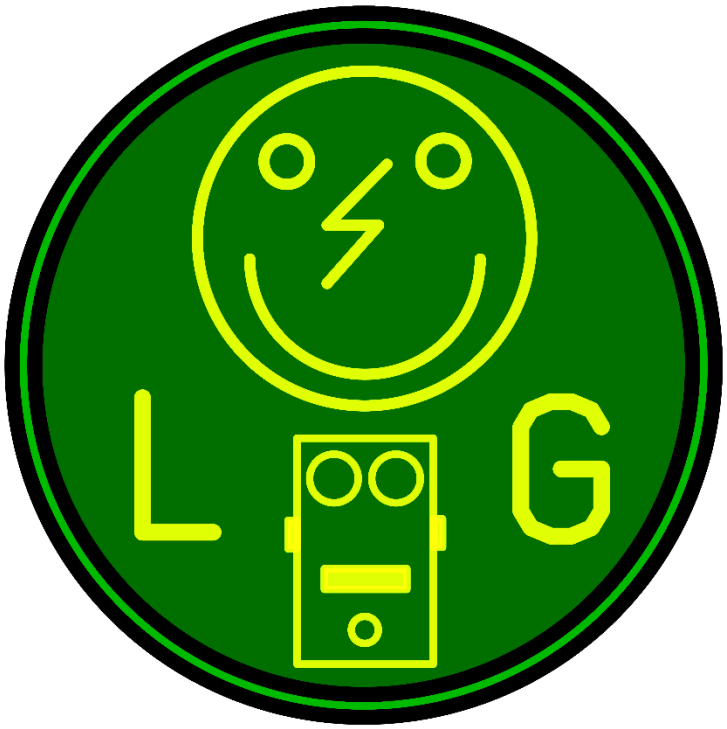


Table of contents

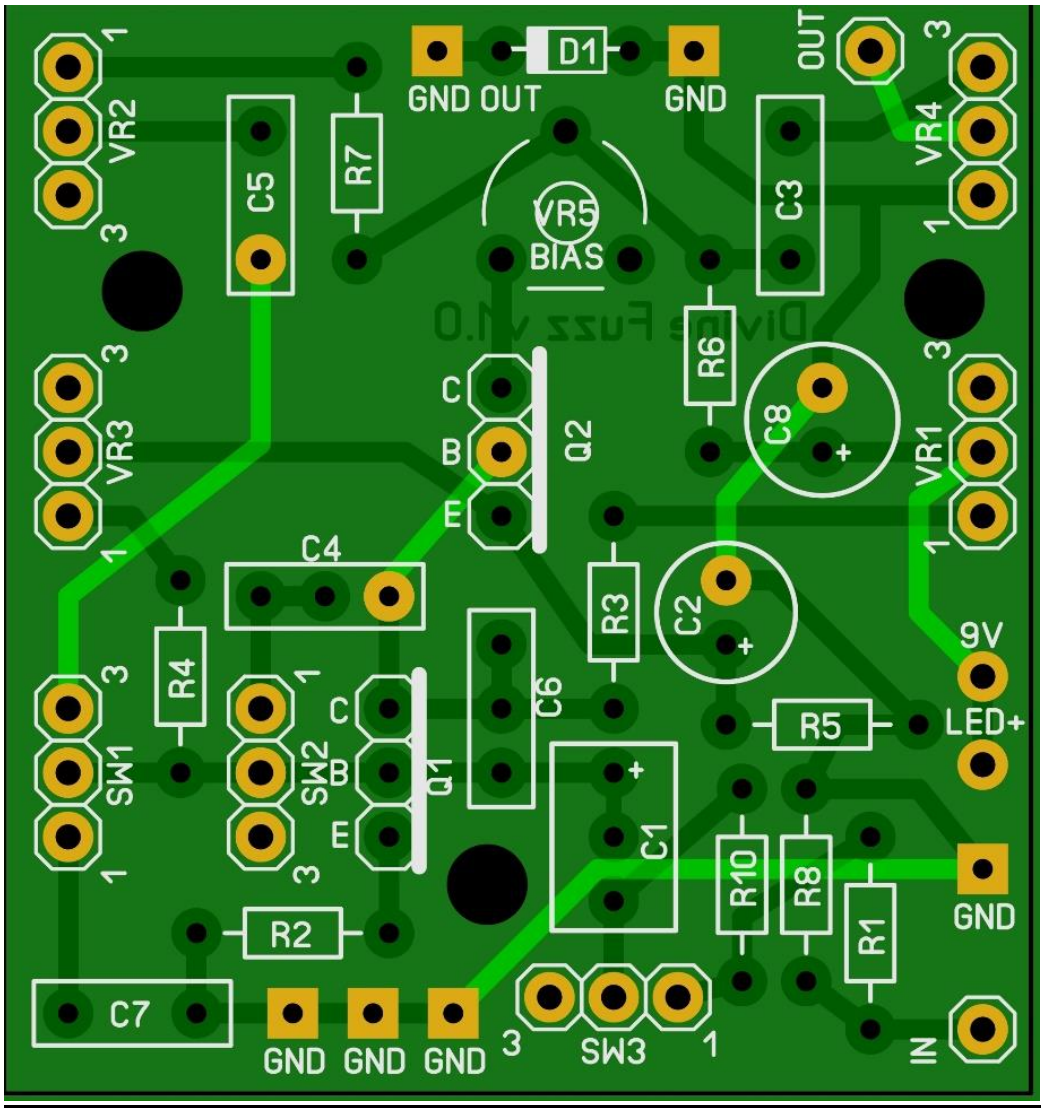
NB

Fuzz effects are known to be more sensitive to other effects in the effects chain than most other effects. So, when testing the effect, also check its placement in your effects chain.

Secondly this Fuzz effect is more sensitive to lower quality components, so you should source your components from a reliable source!

Components

Part	Value	Comment	Part	Value	Comment
C1	2u2		R4	180R	
C2	22u		R5	1K	
C3	100n	MKT	R6	470R	
C4	2n2	MKT	R7	100K	
C5	100n	MKT	R8	1M5	
C6	56p	Ceramic	R10	100K	
C7	1n	MKT	SW1	DPDT	Wrath (footswitch)
C8	100u		SW2	SPST	Treble boost
D1	1N4001		SW3	SPST	Gain boost
Q1	BC109B	Socket	VR1	A25k	Sputter (log)
Q2	BC109C	Socket	VR2	B1M	Wrath (linear)
R1	1K		VR3	A100K	Fuzz (log)
R2	100R		VR4	A100K	Volume (log)
R3	1K		VR5	10k	BIAS Trimpot



General guideline for components

- Capacitors: All values under 1nF should be ceramic disks. From 1nF up to 1uF should be MKT (foil/metalfilm capacitors) and over 1uF use electrolyte caps (or tantalum) 16V+ rated and watch out for polarity!
- Resistors: use 1% metalfilm for the best results.
- Socket all transistors. This way you can easily mod them or replace them if they break.

General building tips

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

Start by soldering the resistors (but not yet VR5)and diode. D1 is optional so if you do not want it, short it with a lead. Solder the small ceramic capacitator C6 next. C6 fits both ceramic as well as MKT. Always insert one leg of the capacitator in the bottom mounting hole and the other leg in one of the other two mounting holes. If you want to experiment with other transistors, you should now solder the sockets for Q1 and Q2.

Now solder the MKT capacitors and the electrolyte capacitors. C1 is made to fit both MKT as well as electrolyte. The polarity is (bottom to top) - + + for the three mounting holes (see page 3). Solder the Bias trimpot VR5.

Place the transistors and you are almost ready to rock. The BC109's are metal can transistors. Please consult the datasheets for correct pinning.

SW1 should be a **footswitch**, SW2 and SW3 are **mini switches** and can also be SPDT. The remaining 3 legs of SW1 can be used to add a LED to the Wrath switch. To do that just look at how the status LED is connected in the off board wiring on page 5 and copy that. If you prefer to let the LED turn on in the other position then just move the grey cable on pin 3 of the switch to pin 1.

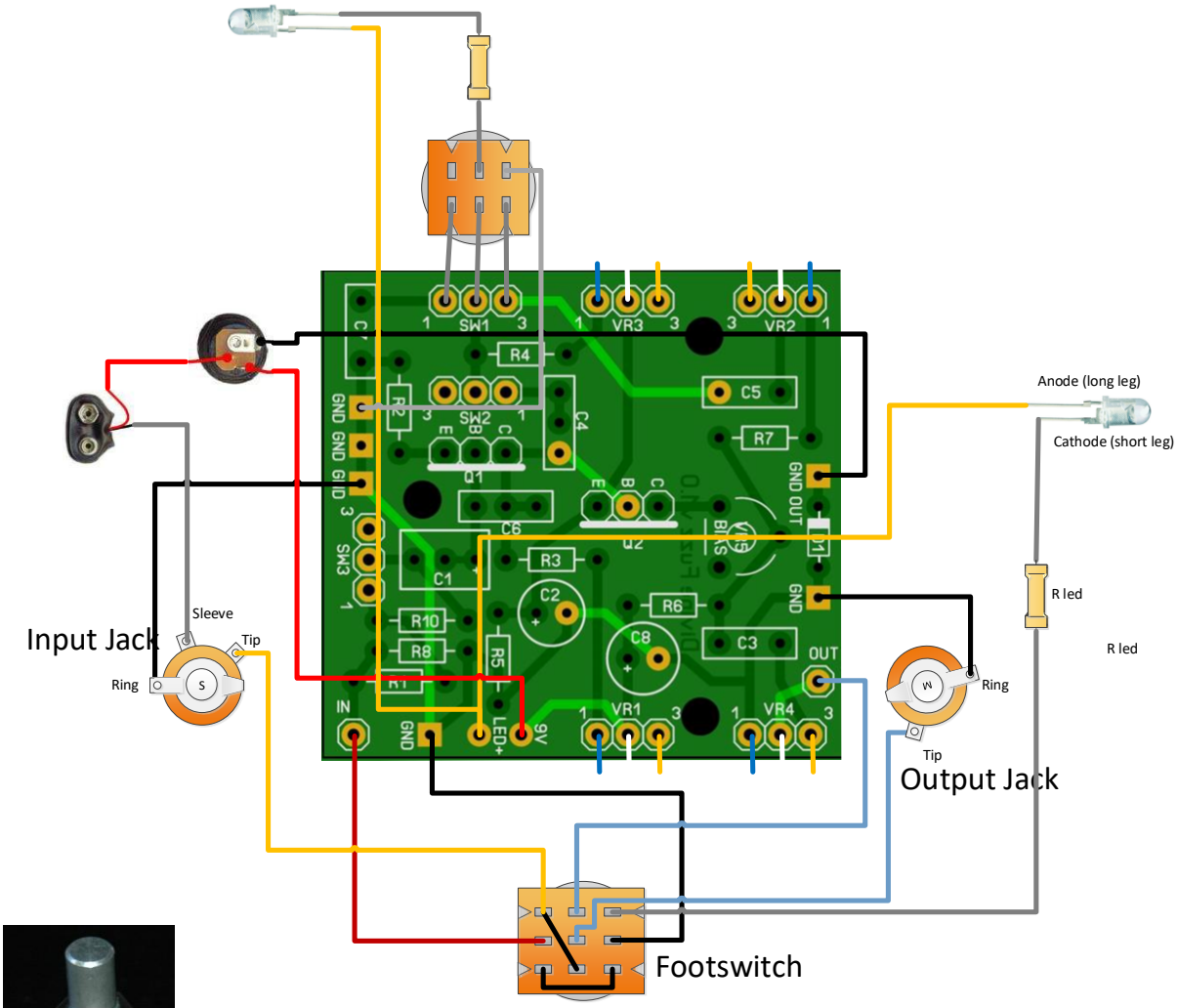
Besides the components mentioned in the table on the first page, you will need:

- **2 input jacks**. 2 mono jacks if you are not going to use a battery but only the 9V adapter. 1 mono (for output) and 1 stereo jack (for input) if you will be using both a 9V battery and the 9V adapter.
- **3PDT footswitch** (9 pins). I also carry an easy off board circuit for true bypass.
- **2,1mm DC jack** (isolated).
- **9v battery clip** (optional).
- **2 LED's**.
- **22 gage stranded hook-up wire**.
- **Hammond 1590BB** case (or similar) in your favourite colour.

Modifications

- It is reported that changing the value of R10 to 12K makes is sound better.
- Instead of the BC109B you can experiment with other transistors like the 2N2222A.
- Originally the values of C4 and C6 are switched (see schematic on p. 7). If you want that you can switch them back, but most prefer the current setup.

Off board wiring



Blue = pin 1
White = pin 2
Yellow = pin 3

NB. If you do not want to use the D1 diode than remember to short it by soldering a lead instead.

NB 2. You are free to use any ground connection you want on the PCB (GND) if the sequence as mentioned in the picture above does not match your build.

Notice that in the “off” position the effect input is connected to ground to prevent possible oscillation.

The LED requires a resistor (R led in the diagram) depending on the type of LED you are using. An ultra-bright red or blue LED requires a 4k7 resistor, Green requires 680R.

Biasing

This layout requires high grade potmeters, capacitators and trimpot to work as expected. The trimpot is used to bias the transistors Q1 and Q2. Start in the middle position and experiment which setting suits your tonal taste best. Also test it with **Wrath** on and off. If you only get squealing then adjust the trimpot. Remember that the squealing/oscillating is part of the effect with the Wrath setting engaged, just the amount of squealing should be set to your own liking! This effect takes time to get to know it and finding your tone. Sputter and Wrath also influence each other's behaviour so also experiment with that.

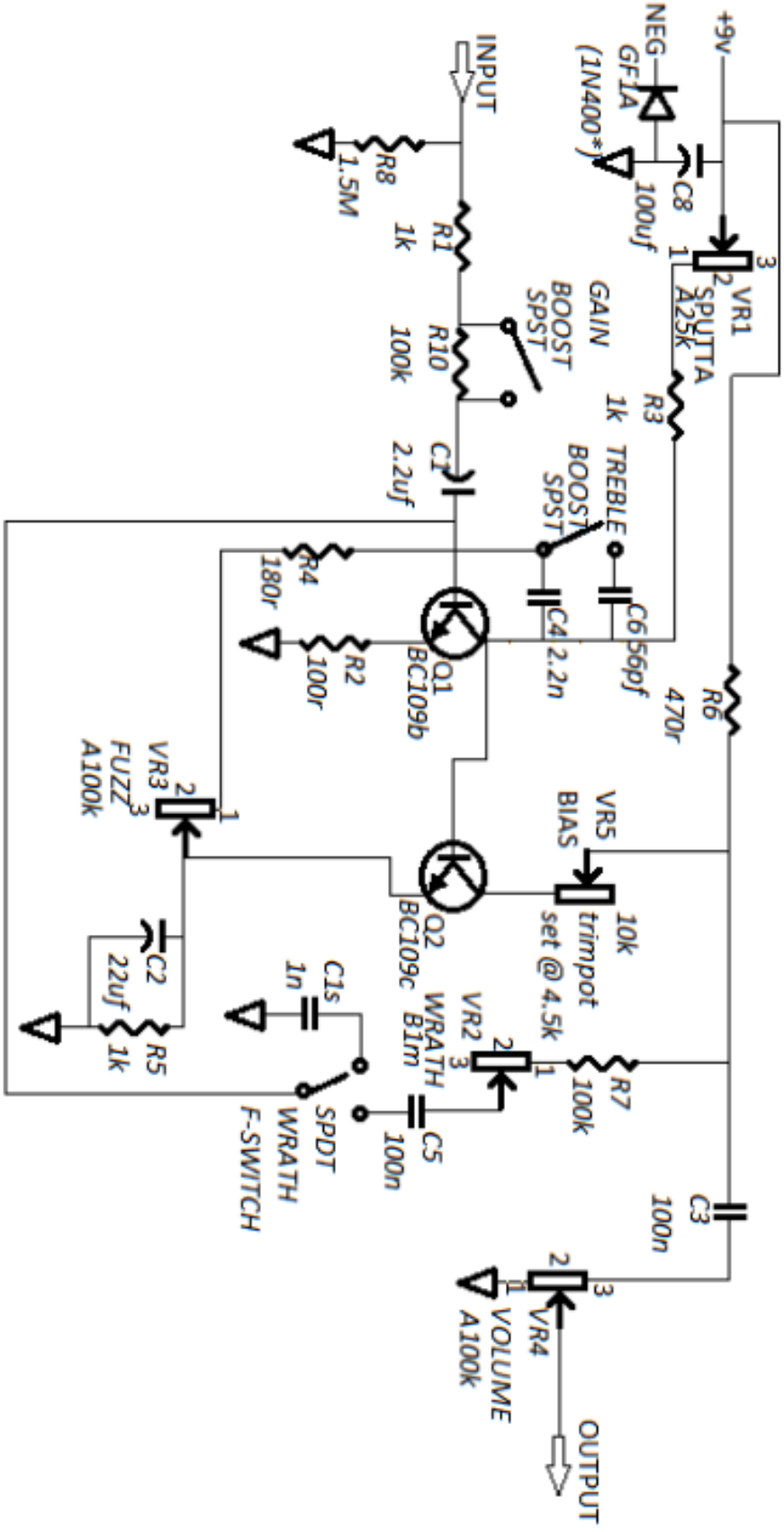
Troubleshooting

All PCB's have been e-tested 100% in the factory, 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).
- Check that you oriented the capacitors, IC's ,transistors and diodes the right way. MKT capacitors and resistors do not need to be oriented.
- 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 lose or cold solder can be really bad for your board.
- Replace the IC's and transistors, one might be defective. Before doing that first unplug the 9V and wait 5 seconds.
- Check that you have good/high grade components. A lot of Chinese sourced parts are fakes (especially high end opamps and transistors) so be careful that you source your parts from reliable suppliers.

Schematic


+9v
NEG
GF1A
(1N400*)
INPUT
R8
1.5M
R1
1k
R10
100k
C1
2.2uf
C8
100uf
VR1
SPUTTA
A25k
GAIN
BOOST
SPST
R3
1k
TREBLE
BOOST
SPST
C6
56pf
C4
2.2n
R4
180r
R2
100r
Q1
BC109b
R6
470r
VR5
BIAS
10k
trimpot
set @ 4.5k
VR3
FUZZ
A100k
Q2
BC109c
VR2
WRATH
B1m
R7
100k
C3
100n
C2
22uf
R5
1k
C1s
1n
SPDT
WRATH
F-SWITCH
C5
100n
VR4
VOLUME
A100k
OUTPUT