Hazy Octave

Building instructions

V1.0

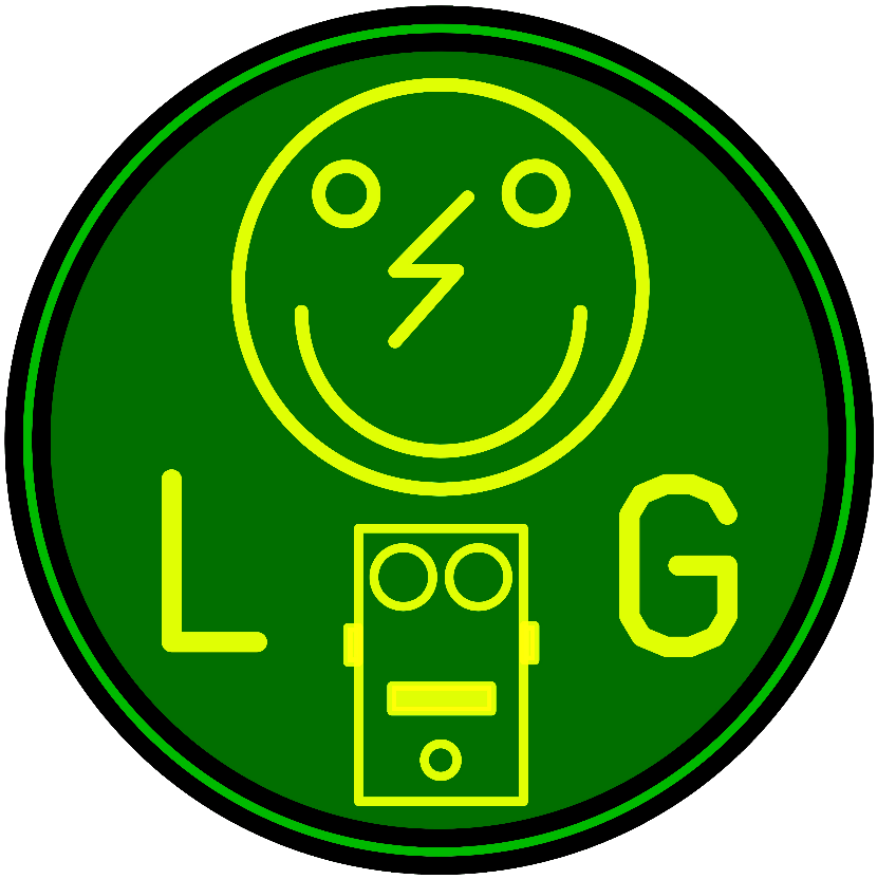


Table of contents

Read this entire manual thoroughly before you start building the effect!

Last update: 17-12-2016

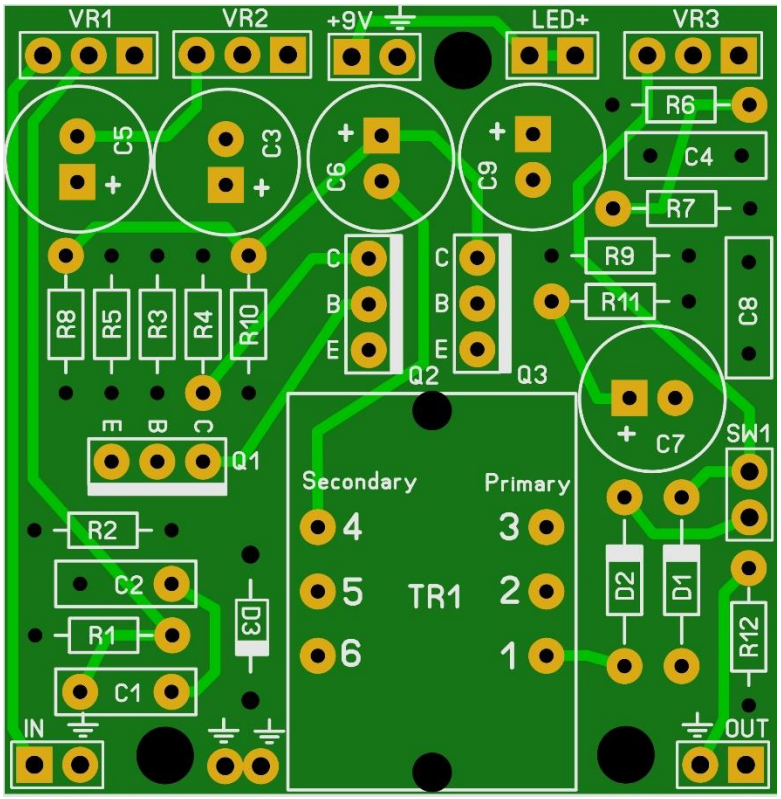
Components

Name	Value	Comment	Name	Value	Comment
C1	100n	SMF	R1	4M7	
C2	150p	Ceramic disc	R2	680k	
C3	33u	Electrolyte	R3	820k	
C4	1n	Ceramic disc	R4	180k	
C5	100u	alternative 22u	R5	1k	
C6	33u	Electrolyte	R6	220k	
C7	220u	Electrolyte	R7	220R	
C8	100n	SMF	R8	47k	
C9	100u	Electrolyte	R9	22k	
D1	1N34A	Germanium Diode or 1N270	R10	1k2	
D2	1N34A	Germanium Diode or 1N270	R11	470R	
D3	1N5817		R12	1M5	or 4M7
Q1	2N3906		SW1	DPDT/3PDT	Footswitch for Octave
Q2	2N2222A		TR1	42TM022	
Q3	2N2222A		VR1	B50k	Pre Gain
			VR2	C1k	Boost (or B1K as alternative)
			VR3	A500k	Volume

PCB layout

43 mm x 44 mm

16.9 in x 17.3 in



Bill of Materials

Capacitors		
Component	**Amount**	**Comment**
150p	1	Ceramic disc
1n	1	Ceramic disc
100n	2	SMF
33u	2	Electrolyte
100u	2	alternative 22u
220u	1	Electrolyte
Diodes		
Component	**Amount**	**Comment**
1N34A or 1N270	2	Germanium Diode
1N5817	1	
Transistors		
Component	**Amount**	**Comment**
2N3906	1	alternative 2N5087
2N2222A	2	alternative 2N5088, 2N4401
Resistors		
Component	**Amount**	**Comment**
220R	1	1% metalfilm
470R	1	1% metalfilm
1k	1	1% metalfilm
1k2	1	1% metalfilm
22k	1	1% metalfilm
47k	1	1% metalfilm
180k	1	1% metalfilm
220k	1	1% metalfilm
680k	1	1% metalfilm
820k	1	1% metalfilm
1M5	1	1% metalfilm
4M7	2	1% metalfilm
B50k	1	Linear 50k potentiometer
C1k	1	Reverse Logarithmic 1k potentiometer
A500k	1	Logarithmic 500k potentiometer
Transformer		
Component	**Amount**	**Comment**
42TM022	1	alternatives 42TU011 or 42TM011

Introduction

The Hazy Fuzz is based on the Tycobrahe Octavia™ (which itself is based on the Roger Mayer Octavia™). It has been specially adapted to fit on a small PCB and include all sorts of modifications. Mods include a pre-gain and octave switch as well as true bypass with pull down resistors. Read the modifications section on how to get as close as possible to the original.

Building 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. If you want to experiment with other transistors then you could socket them instead of soldering them to the board. You'll need a 20 SIL, break off the sockets and solder them to the board.

Start by soldering the resistors and diodes D1 and D2. Warning: Germanium diodes are fragile! Do not heat them to long or they will break.

If you want to socket the transistors then solder the sockets now. **Note:** Do not blow on your solder in an attempt to cool it down. That will possibly result in a bad join that might corrode!

Solder the ceramic capacitors next and then the small SMF capacitators, then the electrolytes, finish by soldering the transformer to the board.

Place the transistors and you are almost ready to rock.

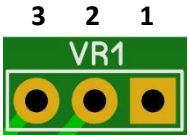
Besides the components mentioned in the Bill of Materials table, you will need:

- **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).
- **2,1mm DC jack** (isolated).
- **9v battery clip** (optional).
- **22 gage stranded hook-up wire.**
- **2 x LED.** Only 1 if you are not implementing the octave switch (SW1)
- **Hammond 1590B** case (or similar) in your favorite color.

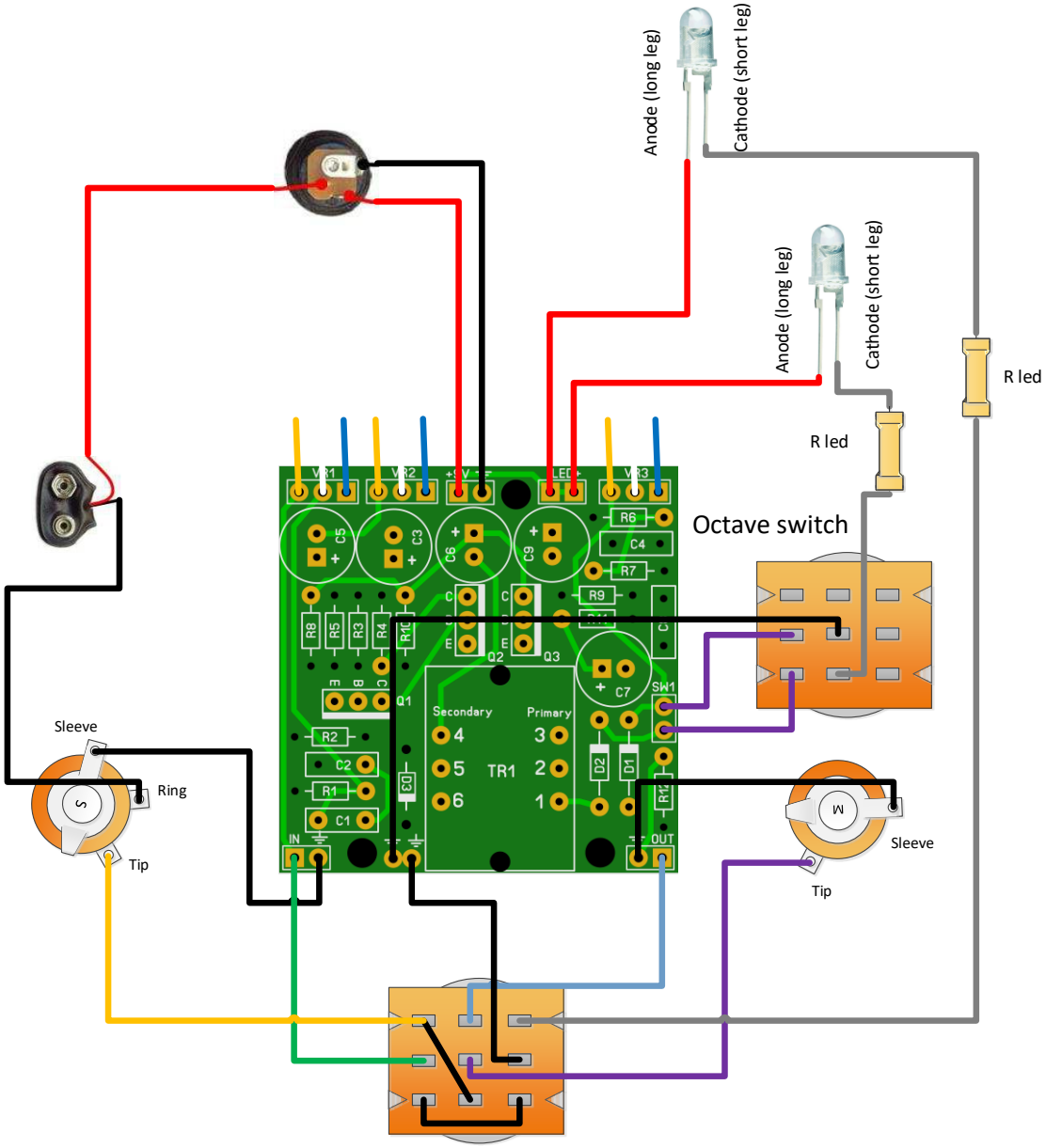
Off board wiring

Wiring the pots VR1-VR2 is very simple. The rectangle pad marks pin 1 of a potentiometer. The images below show how you can recognize which pin is which on a potentiometer.

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

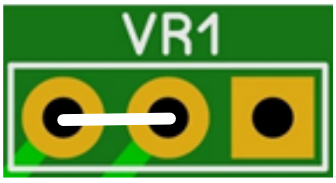


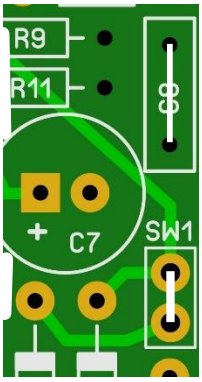
Now you can wire the rest of the board, keep the wires as short as poss ible to prevent noise. Read the modifications section on how to downsize!



Modifications

The layout of the board is made to incorporate as much modifications as possible. If you want to get as close as possible to the original than leave out VR1 (pre-gain) and short pads 2 and 3. Also do not install R1, R12, C8, C9 and the octave switch SW1 and just short both pads of SW1 and C8.





D1 and D2 need to be Germanium diodes. Preferably 1N34A or alternatively 1N270. It's been mentioned that you should try to match the forwarding voltages of both diodes for an optimal result. The Boost is reported to perform best with a 100uF in C5. Alternatively you can try a 22uF in C5. If you want to experiment with different values as C5 then solder 2 SIL pins in C5.

You can also use different mixes of transistors. I encourage you to experiment with this because it is always a taste question. Some mixed will differ more profound than others. Mixes you could try for Q1, Q2 and Q3:

- 2N3906 and 2N2222A (2x)
- 2N5087 and 2N4401 (2x)
- 2N5087 and 2N5088 (2x)

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).
- 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.
- 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, it 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, vintage diodes and transistors) so be careful that you source your parts from reliable suppliers.

Schematic