Orange Alpaca

Building instructions v1.0

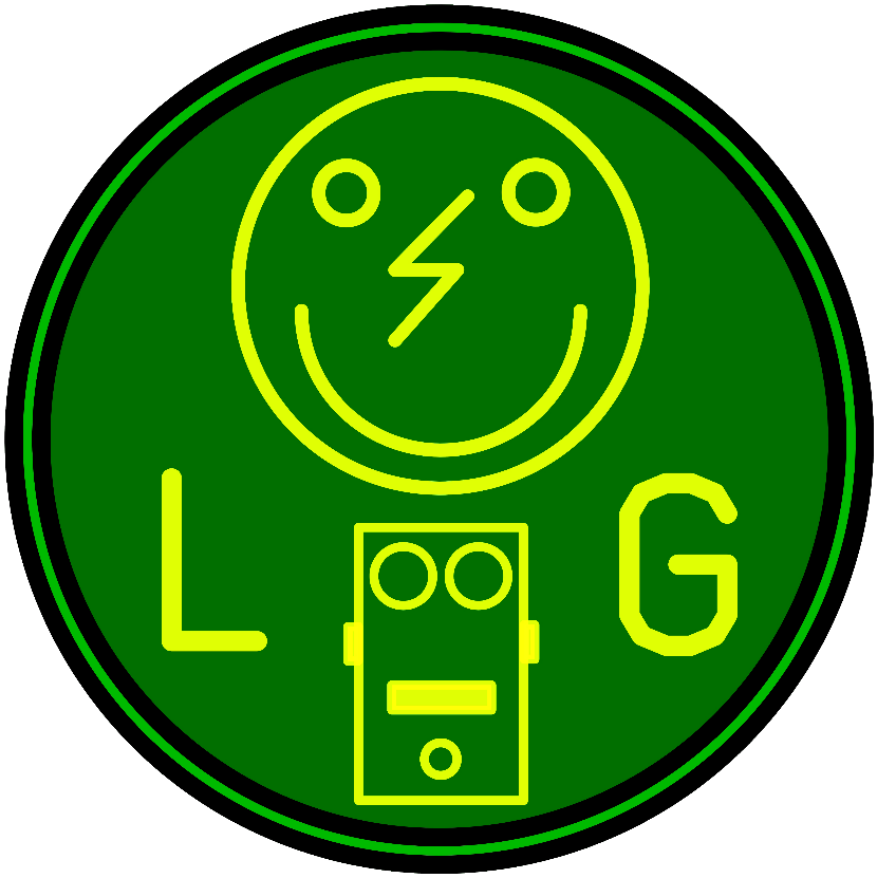


Table of contents

Read this entire manual thoroughly before you start building the effect! Especially the modification part. Decide before building the effect which mods you want to try so that you do not need to desolder parts later.

Last update: 17-10-2021

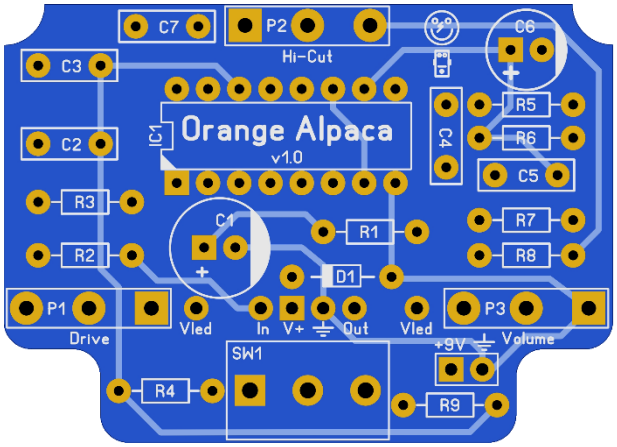
Components

Name	RL	RL 25th	TSF	OA	Comment
C1	100u	330u	100u	100u	Electrolytic
C2	68n	68n	100n	68n	SMF/MKT/Wima
C3	51p	51p	10p	51p	MLCC/Ceramic
C4	33n	33n	50n	33n	SMF/MKT/Wima
C5	100p	100p	10p	100p	MLCC/Ceramic
C6	10u	10u	10u	10u	Electrolytic
C7	x	33n	x	33n	SMF/MKT/Wima
D1	x	1N4001	x	1N4001	
IC1	CD4049UBE	CD4049UBE	CD4049UBE	CD4049UBE	
P1	B1M	B1M	B1M	B1M	Drive
P2	x	B100k	X	B100k	Hi-Cut
P3	A10k	B10k	B10k	B10k	Volume
R1	1k	1k	100R	1k	1% metalfilm
R2	1M	1M	x	1M	1% metalfilm
R3	100k	100k	100k	100k	1% metalfilm
R4	x	x	10M	10M	1% metalfilm
R5	1M	1M	1M	1M	1% metalfilm
R6	x	1M	x	1M	1% metalfilm
R7	Jump	27k	Jump	Jump	1% metalfilm
R8	x	1M	x	1M	1% metalfilm
R9	100k	91k	100k	91k	1% metalfilm
SW1	Jump 2-3	Jump 2-3	SPDT	SPDT	Mode switch

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

RL= Red Lama™, **RL 25th** = Red Lama 25th anniversary Edition™
TSF = Tube Sound Fuzz™, **OA** = Orange Alpaca

PCB layout



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.

Do not blow on your solder in an attempt to cool it down. That will possibly result in a bad join that might corrode!

First you need to decide which version you are going to build, then you can start by soldering the resistors and then the diode (if applicable). Solder the IC socket next and then the small SMF/MKT/MLCC capacitators, transistors (if not socketed) and finish with the electrolytes.

Do not yet place the IC (and diodes/transistors if socketed). Wait until after you soldered the potentiometers and switch.

The Mode switch (**SW1**) controls the range Drive (**P1**). You can either use a PCB mounted switch SPDT (on-on) or a regular SPDT and solder it using wires (see off board wiring)

Besides the components mentioned in the table on the previous 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)
- **2,1mm DC jack** (isolated).
- **9v battery clip** (optional).
- **22 gage stranded hook-up wire.**
- **LED and LED flange.**
- **Hammond 1590B** case (or similar) in your favorite color. If you need more space, use a Hammond 1590BB.

Modifications

Mode switch

You can make the switch **SW1** into a footswitch. Use a DPDT so you can use the second row of pins to connect an LED to so you can see if the lead mode is engaged.
If R4 (lead resistor) is 10M the drive pot will become almost useless. You might consider a lower value like 4M7.

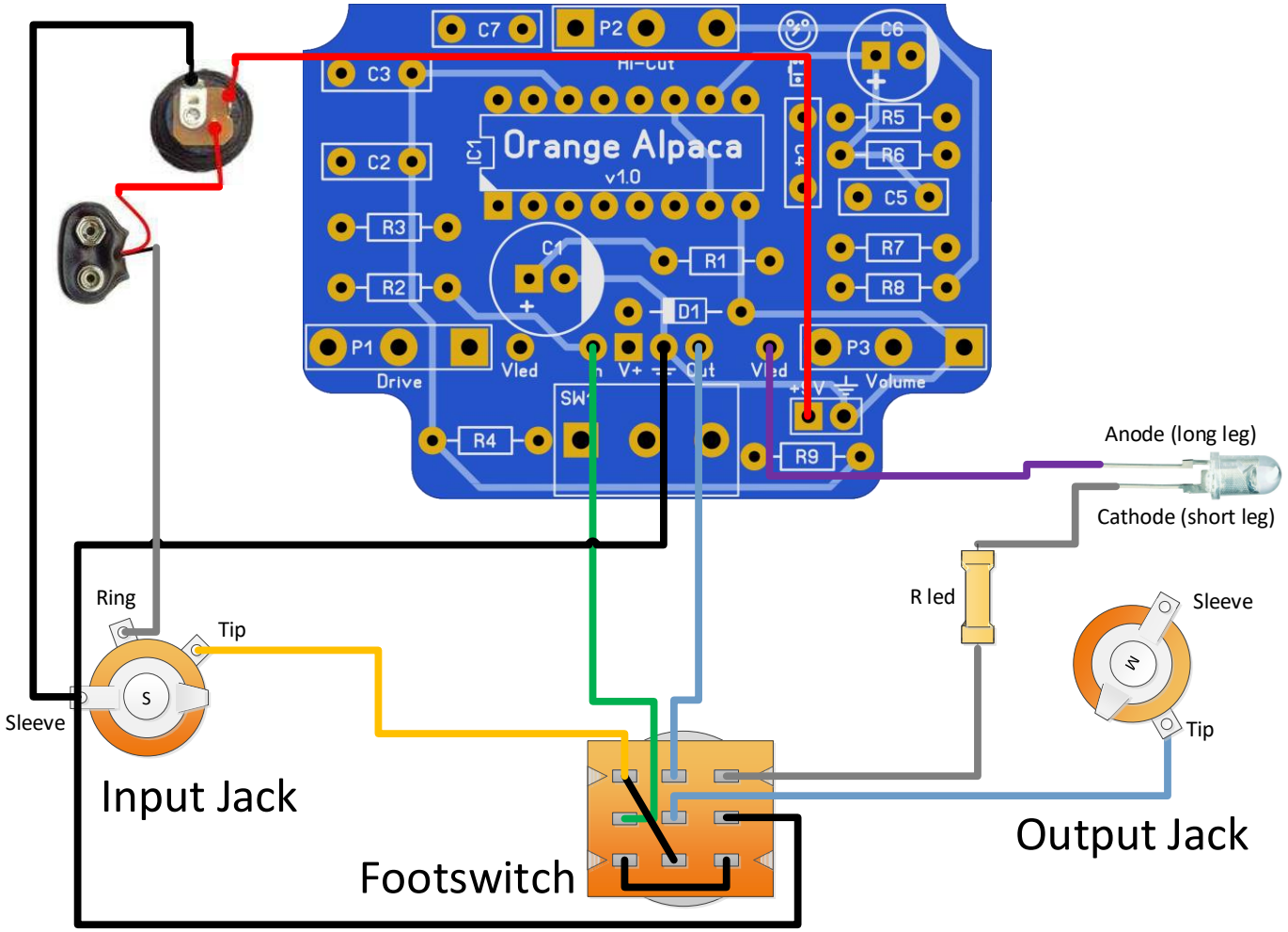
Off board wiring

The biggest challenge of this build is to get all the offboard wiring correct and fit it in a box. Take your time measuring and testing before you start fitting everything in the box.

P1-**P3** are PCB mounted potentiometers (Alpha). The rectangle pad marks pin 1 of a potentiometer. NB the pots need to be soldered ont the bottom side not the component side (top side). The image to the left shows how you can recognize which pin is which on a potentiometer. You can break off the pin I marked with the *yellow circle* with a small pair of pliers.

SW1 is a regular PCB mounted SPDT (on-on) switch or you can use a switch with solder lugs for external use.

Place the potentiometers in the predrilled enclosure and then insert the PCB. Once the PCB fits nicely and is not touching the enclosure, solder the potentiometers to the PCB. Now you will need to wire the footswitch, LED and DC jack:



The LED requires a resistor (R led in the diagram) depending on the type of LED you are using. An ultra-bright red, green or blue LED requires a 4k7 resistor, diffuse green requires 680R. If you want to be more exact then go here: http://www.muzique.com/schem/led.htm

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