OmniScreamer

Building instructions

v1.0

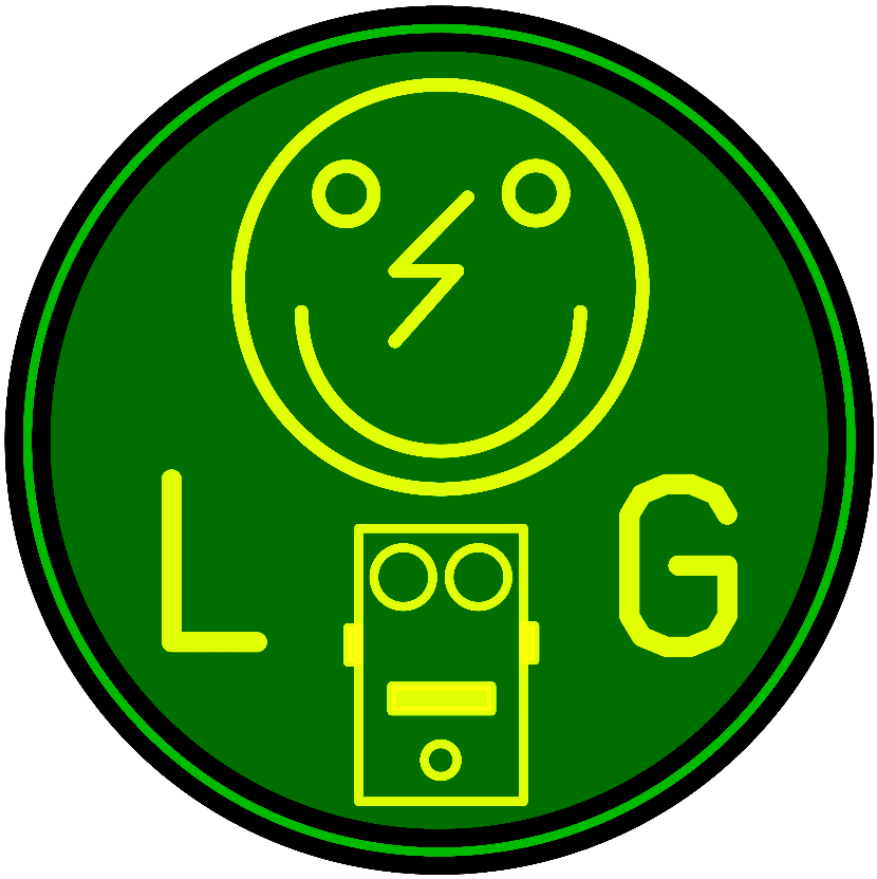
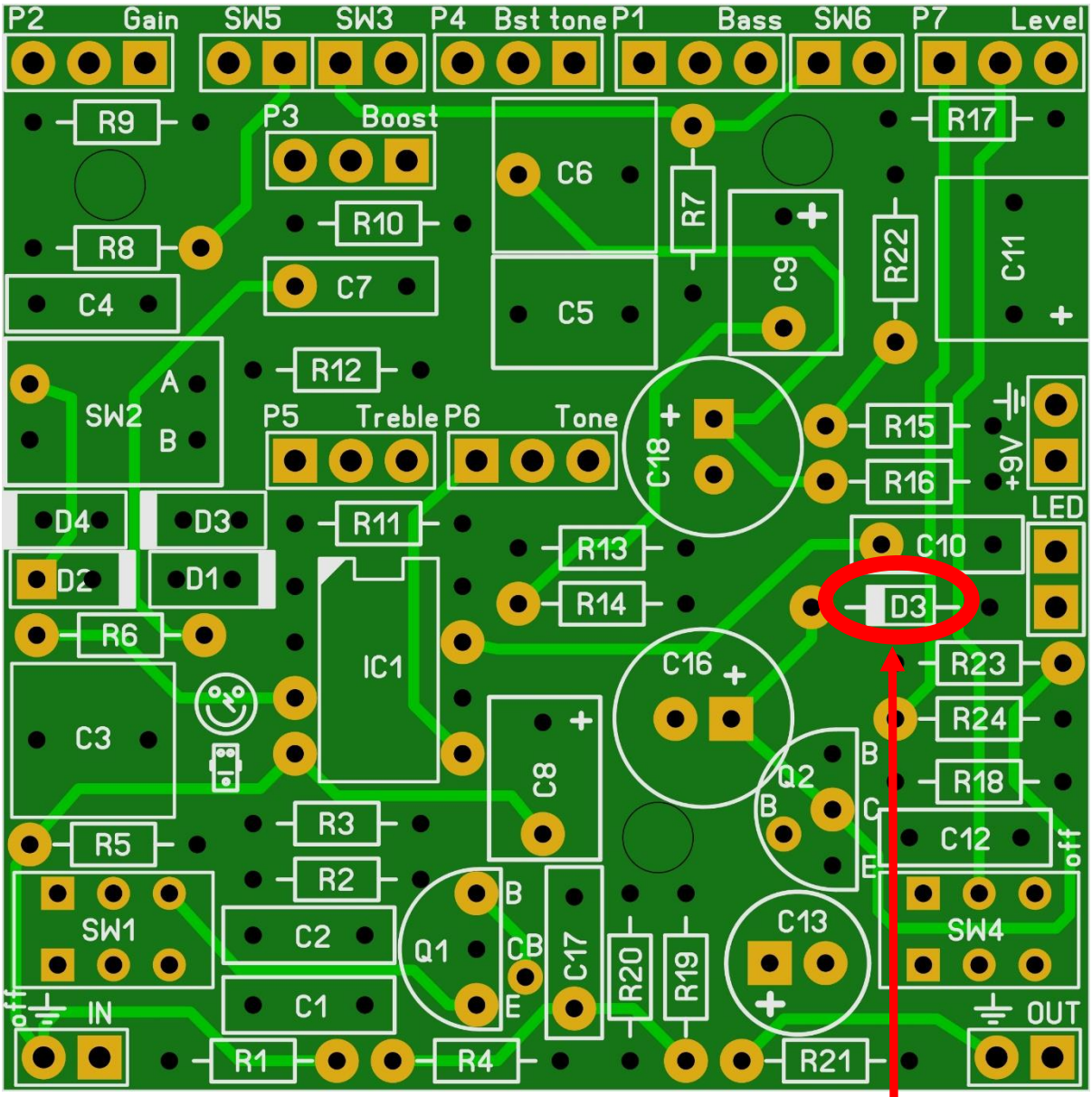


Table of contents

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: 17-07-2019

PCB layout



Dimensions: **49,5 mm x 49,5 mm**
1,95 inch x 1,95 inch

Soldermask mistake! Should be D5

Introduction

Before we start with the component list, it might be interesting to know how this PCB was designed.

Strictly speaking, you can break down this tubescreamer into 5 main sections and 3 optional mods.

The main sections are:

1. Input buffer
2. Output buffer
3. Main clipping stage
4. Tone/Volume stage
5. Power section

Both the input (1) and output (2) buffers are made switchable, so I would suggest to add them and if you do not like them, you can just switch them off.

The main clipping stage (3) is where the most fun is at. Most tubescreamer based effects will have a wide variety of clippingdiode configurations. In this manual you will find a lot of these configurations will be explained and you can always decide to “mix and match”. A tubescreamer is an extremely “modable” effect.

The tone and volume stage is built to fit almost all types of tone controls as found in a lot of tubescreamer based effects.

The power section is not that interesting, but some versions use an a-symmetrical voltage divider to form V_{ref}. The only non-standard is that the polarity protection is changed to a (safer) serial connection. This has no tonal effect what so ever!

The optional mods:

1. Boost
2. Offboard Clipping diodes
3. Feedback loop

All these mods can be added to any version you want to build! In the separate configurations sheet you will find which mod is standard for some of the configurations and which are not. Again you are free to add these mods or leave them out.

The boost mod (**SW3**) is a mod found in the Tim™. The off board clipping diodes switch (**SW5**) can be used to add extra clipping diode configurations instead of (or in addition to) the onboard clipping diode section. The feedback loop (**SW6** and **R22**) will reduce the overall gain and brightness depending on the capacitor that is switched in the circuit.

You should also take a look at the Electrosmash Tubescreamer dissection. It has a more in-depth analysis (note that it also covers the JFET foot switching that is intentionally left out of this PCB)

The components needed for the different type of effects can be found in the separate configuration sheet (pdf).

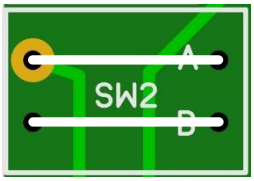
Configurations and components

This PCB can be configured to become a wide range of Tubescreamers. The components needed to build these configurations can be found in the separate configuration sheet.

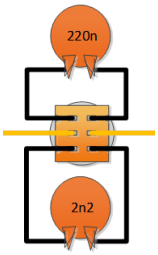
If you are not sure which type of components you need to buy, you can follow this list:

1. All resistors are metal film 1% 0,6W
2. Capacitors smaller than 1nF are MLCC unless stated otherwise in the sheet.
3. Capacitors 1nF – 1uF are MKT/Wima/SMF which ever you prefer unless stated otherwise in the sheet.
4. Capacitors 1u and bigger are Electrolytic and rated 25V+ unless stated otherwise in the sheet.
5. **SW1** and **SW4** (buffers) can best be miniature PCB mounted DPDT (on-on) switches :

6. **SW2** (onboard diode selection) is normally a DIP 2. If the sheet says “jump” you need to short both poles like this:



7. **SW3** (Boost) is a standard DPDT On-On footswitch
8. **SW5** (External Diode section)can best be DPDT (On-Off-On). You can find all types of diode configurations in the diode section of these instructions. **You can also buy a separate clipping PCB in the shop!!**
9. **SW6** (feedback loop) is a DP3T On-Off-On switch and uses a 220nF and 2nF capacitor on each side. You will also need to put a 47k in **R22** on the PCB. The switch can be wired like this:



Pots

P1 Bass
P2 Gain
P3 Boost
P4 Boost Tone
P5 Treble
P6 Tone (or Glass)
P7 Level

Build sequence

Before starting with this section, make sure you have read the configurations section first!

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 jumpers, resistors and small diodes (not LEDs if used). If needed you can create a jumper using a spare piece of lead from a resistor or diode.

If you want to experiment with other transistors for **Q1** and **Q2** then you could socket them instead of soldering them to the board. You'll need a some 20 SIL sockets, break off the 3 pin sockets and solder them to the board. Now is the time to solder 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 MLCC, small SMF/MKT/Wima capacitors. Now finish by soldering the transistors (if not socketed), the bigger SMF/MKT/WIMA and the electrolytic capacitors.

I suggest you now drill the holes in your enclosure so you can use it during the off board wiring. This PCB is very sensitive to noise! Prevent crossing input and output wires and keep the wires as short as possible (and/or use shielding on the input and output).

Besides the components mentioned in the configuration sheet, you will need:

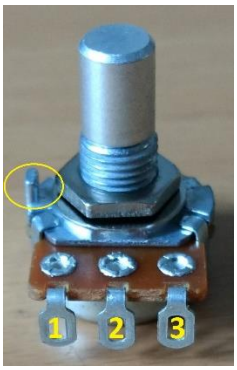
- **1 stereo input jack and 1 mono output jack**.
- **2,1mm DC jack** (isolated).
- **22 gage stranded hook-up wire.**
- **LED holder.** This enables you to mount the LEDs in the enclosure.
- **Footswitch 3PDT** (9 pins) and also a **DPDT** if you are going to build the Boost switch (**SW3**)
- **Hammond 1590B/125B** style case (or similar) in your favorite color. If you are going to use all the switches and pots, I suggest you use a **1590BB** enclosure.

Off board wiring

Potentiometers

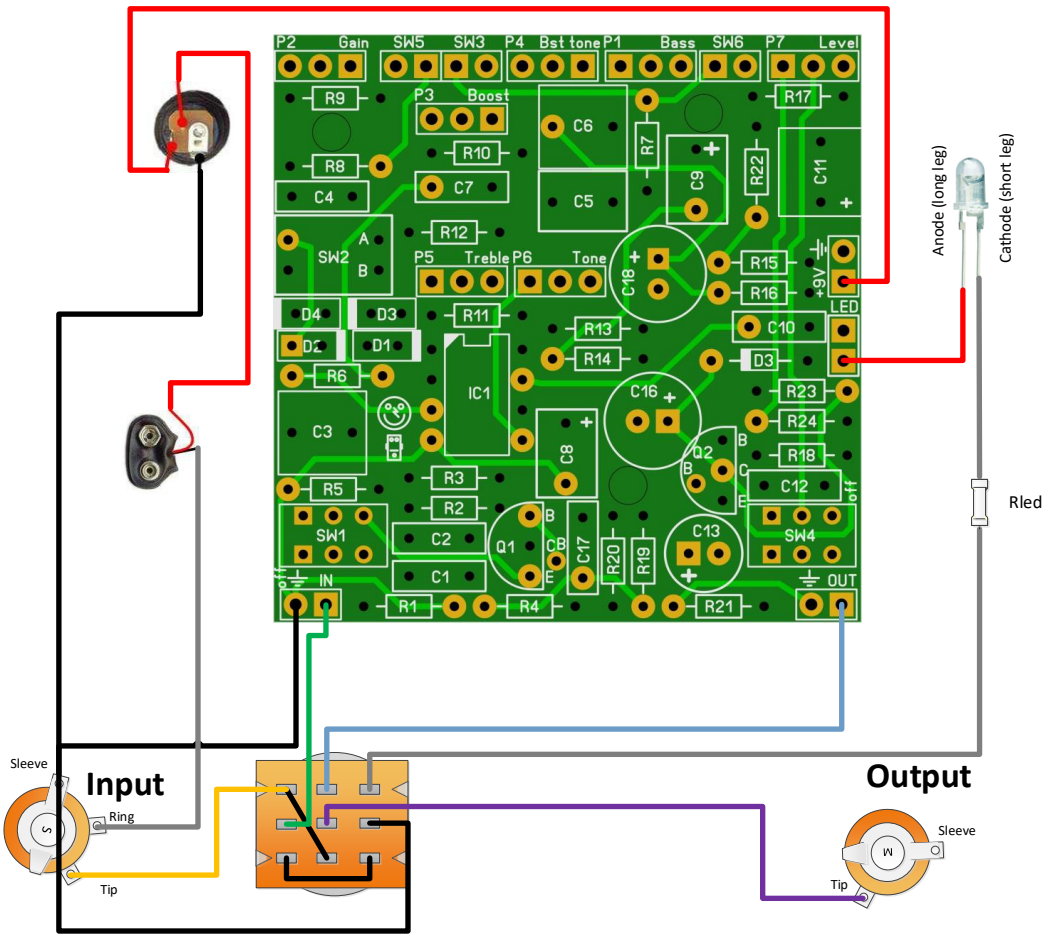
In the pictures below you see the correct pin numbering of the pots (Alpha 16mm style).
Solder the wires accordingly and it is always a good idea to twist the wires together to create some extra shielding against external noise. The **rectangle pad** marks **pin 1**.

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



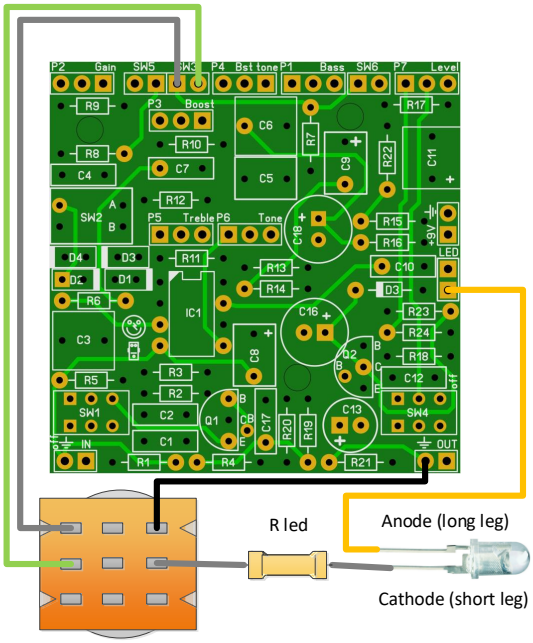
3 2 1

Main bypass wiring



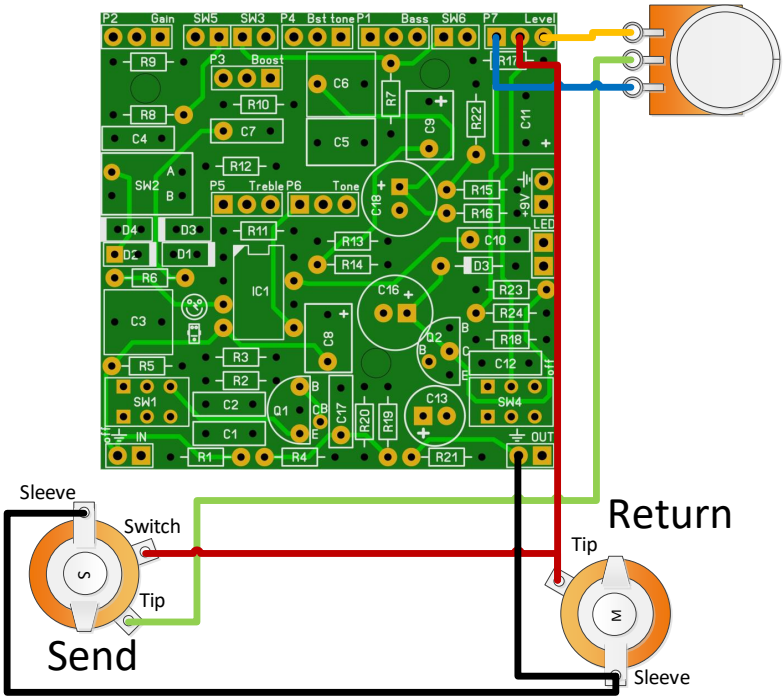
Rled can be somewhere between 1k5 and 10k (try 4k7). The lower the value the brighter the LED shines, but at the cost of power consumption and wear of the LED.
The diagram is also based on star wiring where all ground connections go to the sleeve of the input jack. It requires very good conductivity between the input jack, output jack and the enclosure to work correct. If you do not have good conductivity, please connect a wire between the input jack sleeve and the output jack sleeve.

Boost wiring Tim™ (optional)



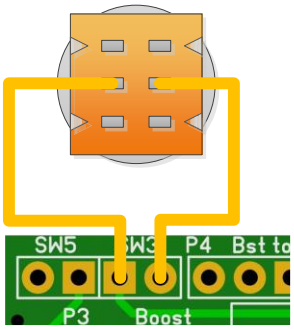
You could also use a DPDT footswitch

Effects loop wiring Tim™ (optional)



External Diodes configuration

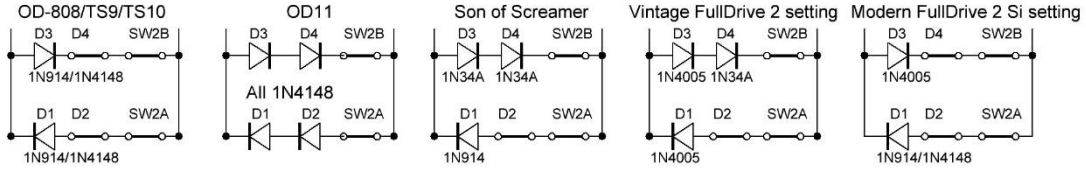
The wiring for the **SW5** switch is the same as for potentiometers, marking pin 1 with a **rectangle pad**. There is no wrong way to wire it as long as the middle lugs of the DP3T are connected to one of the pads like this:



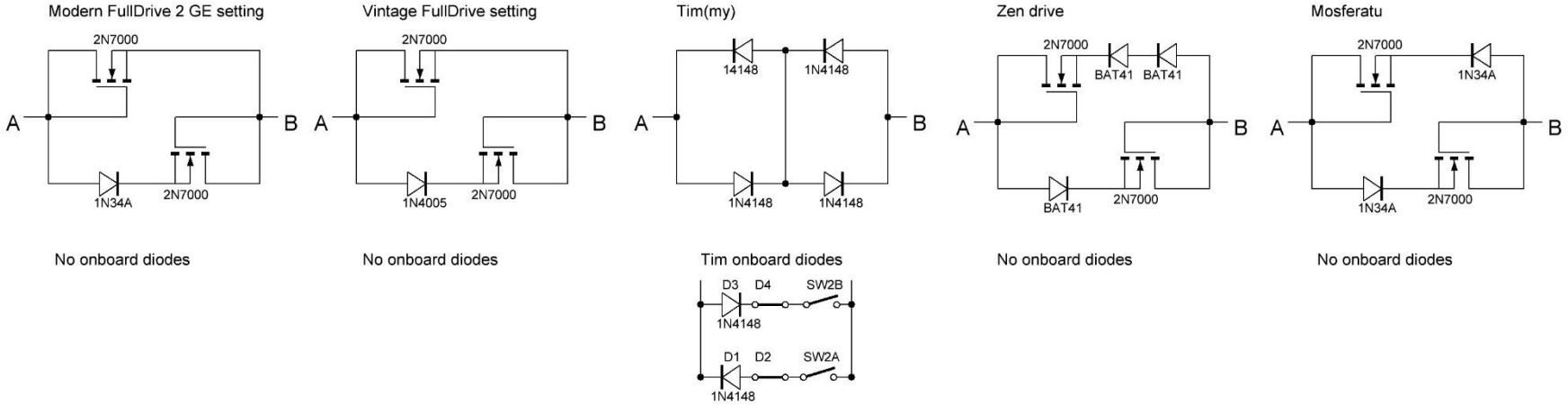
The optional diode switch (**SW5**) enables you to switch between 2 sets of diodes. There are so many combinations possible and that is why there is a whole page committed to all kinds of combinations. Feel free to experiment and combine all kinds of diodes.

If you want to use both the onboard as well as the off board diode combinations then make sure you add SW2 (DIP2) to the PCB so you can switch off the onboard diodes when you wish (like the Tim™).

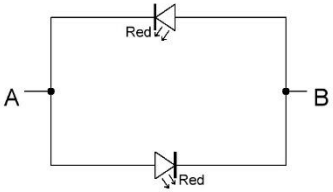
Onboard clipping suggestions



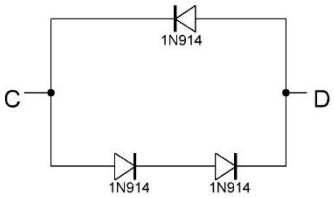
SW5 off board suggestions



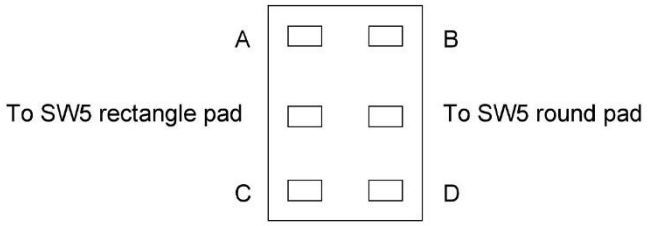
Dynamic OD setting 1



Dynamic OD setting 2



SW5 layout (DPDT On-Off-On)



Modifications

OpAmp

Almost all types of dual OpAmps can be used as long as the pinout is the same. Just a few that have been tested with success (meaning, they sound good). JRC4558(P), JRC4559, OPA2134, TL072, TL082, JRC 1458, AD712, OPA2604.

Transistors

The buffer transistors can als be 2N3904 or 2N5088. These transistors do have a different pinout than the 2SC1815. Extra pads added have been added to the PCB to accommodate these pinouts. Check the pinout of your transistor before you insert then in the corresponding B/C/ or E pad.

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.

Schematic