StormBoost

Building instructions

V1.0

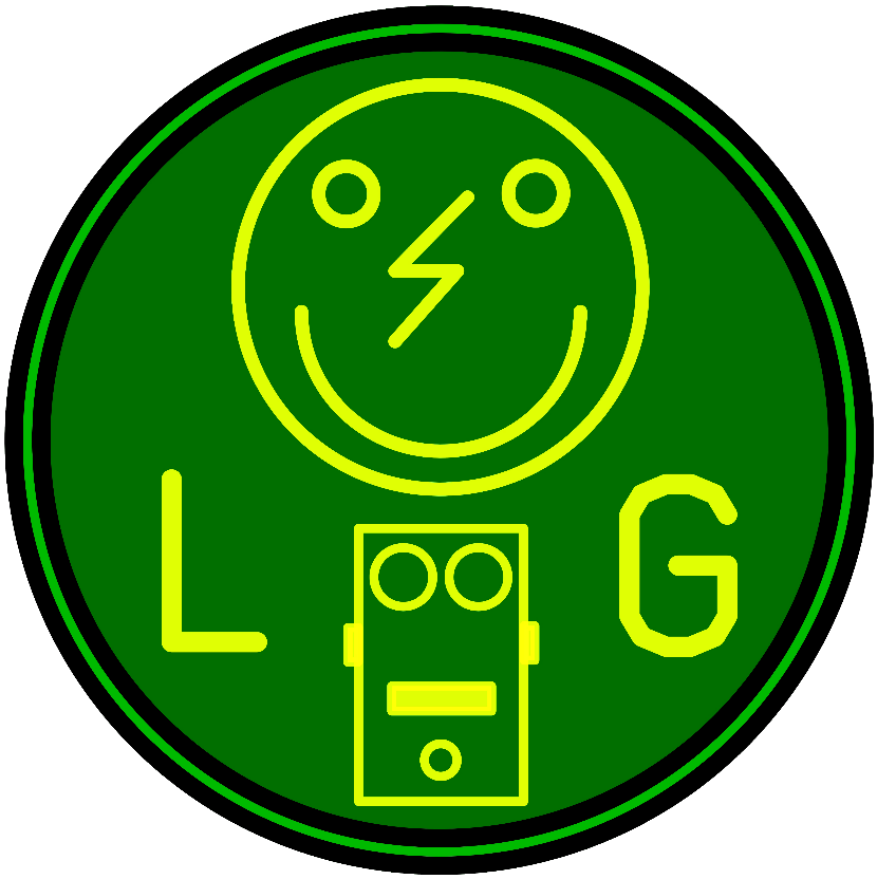


Table of contents

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.

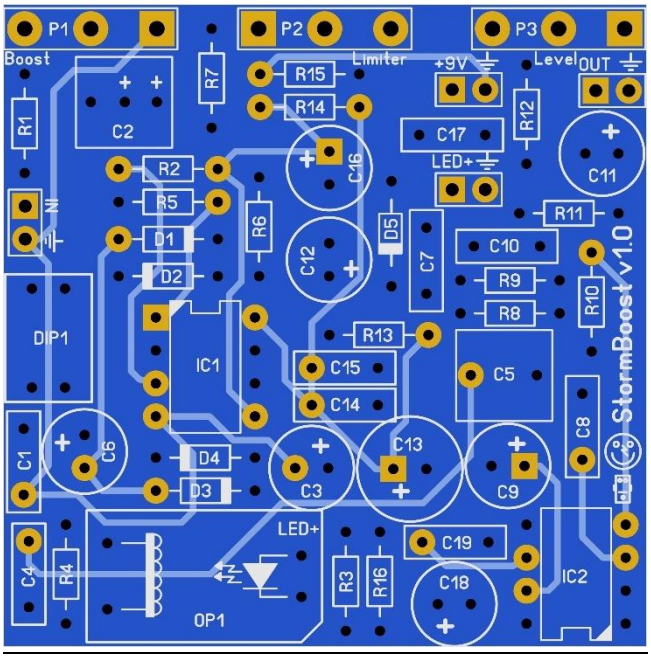
Last update: 02-02-2020

Components

Name	Value	Comment	Name	Value	Comment
C1	120p	SMF/MKT/Wima	**DIP1**	DIP2	
C2	1u	SMF/MKT/Wima/Electrolytic	**IC1**	RC4558P	
C3	10u	Electrolytic 25V+	**IC2**	LM386N-1	
C4	120p	SMF/MKT/Wima	**OP1**	VTL5C3	original Vactrol NSL-32
C5	1u	SMF/MKT/Wima	**P1**	B100k	Boost
C6	10u	Electrolytic 25V+	**P2**	A100k	Limitter
C7	1n	SMF/MKT/Wima	**P3**	B1k	Level
C8	100n	SMF/MKT/Wima	**R1**	2K	1% metalfilm
C9	10u	Electrolytic 25V+	**R2**	1M	1% metalfilm
C10	100n	SMF/MKT/Wima	**R3**	2k	1% metalfilm
C11	10u	Electrolytic 25V+	**R4**	15k	1% metalfilm
C12	10u	Electrolytic 25V+	**R5**	1k	1% metalfilm
C13	10u/100u	Electrolytic 25V+	**R6**	10k	1% metalfilm
C14	100n	SMF/MKT/Wima	**R7**	10k	1% metalfilm
C15	100n	SMF/MKT/Wima	**R8**	2k	1% metalfilm
C16	10u	Electrolytic 25V+	**R9**	8k2	1% metalfilm
C17	100n	SMF/MKT/Wima	**R10**	2k	1% metalfilm
C18	10u	Electrolytic 25V+	**R11**	10R	1% metalfilm
C19	100n	SMF/MKT/Wima	**R12**	10R	1% metalfilm
D1	BAT42		**R13**	10R	1% metalfilm
D2	BAT42		**R14**	10k	1% metalfilm
D3	BAT42		**R15**	10k	1% metalfilm
D4	BAT42		**R16**	10R	1% metalfilm
D5	1N5817				

PCB layout

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



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!

If you want to experiment with other diodes (**D1**-**D4**) 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. **Note:** Glass diodes are fragile. Do not expose them to heat for too long or they'll break!

Start by soldering the jumper (if needed), resistors and then the diodes (if not socketed). Solder the IC sockets next and then the small SMF/MKT capacitators, the DIP switch, and finish with the electrolytes.

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

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 125B** case (or similar) in your favorite color. If you need more space, use a Hammond 1590BB.

Modifications

Onboard DIP switches

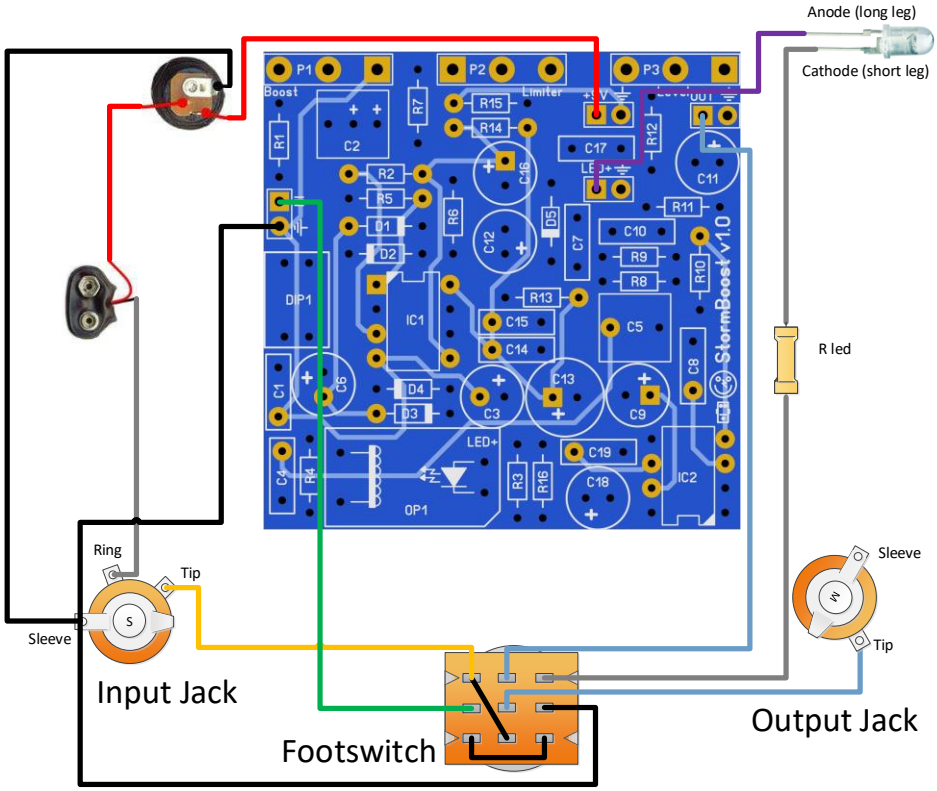
The original has 2 unconnected parts. First there is a RF filter cap (**C1**) and secondly there is an extra electrolyte cap (**C6**) that controls the attack time of the boost. Both can be optionally engaged using the Dip switches. If you do not want them, you can leave them out (**Dip1**, **C1**, **C6**).

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. The image 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.

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