Clean Blade Boost
Building instructions
V1.0

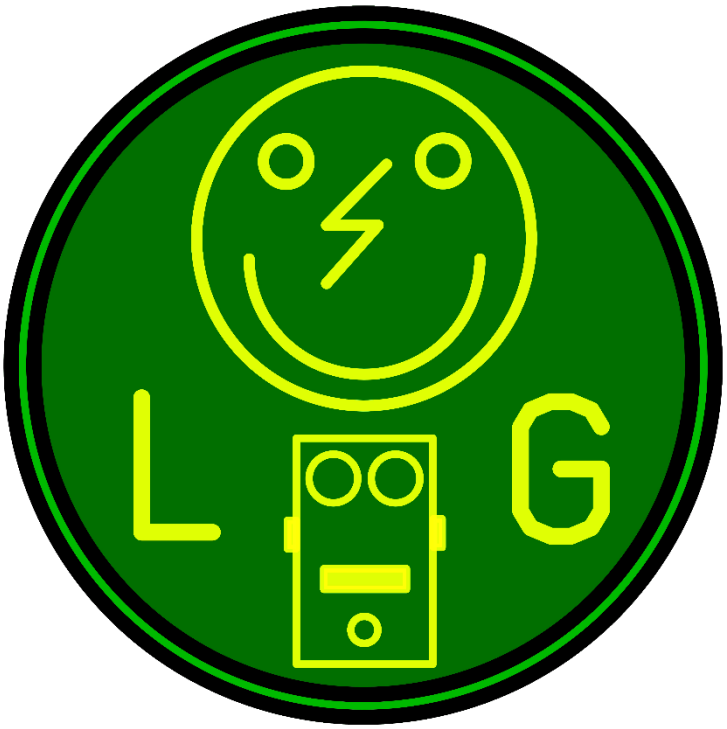


Table of contents

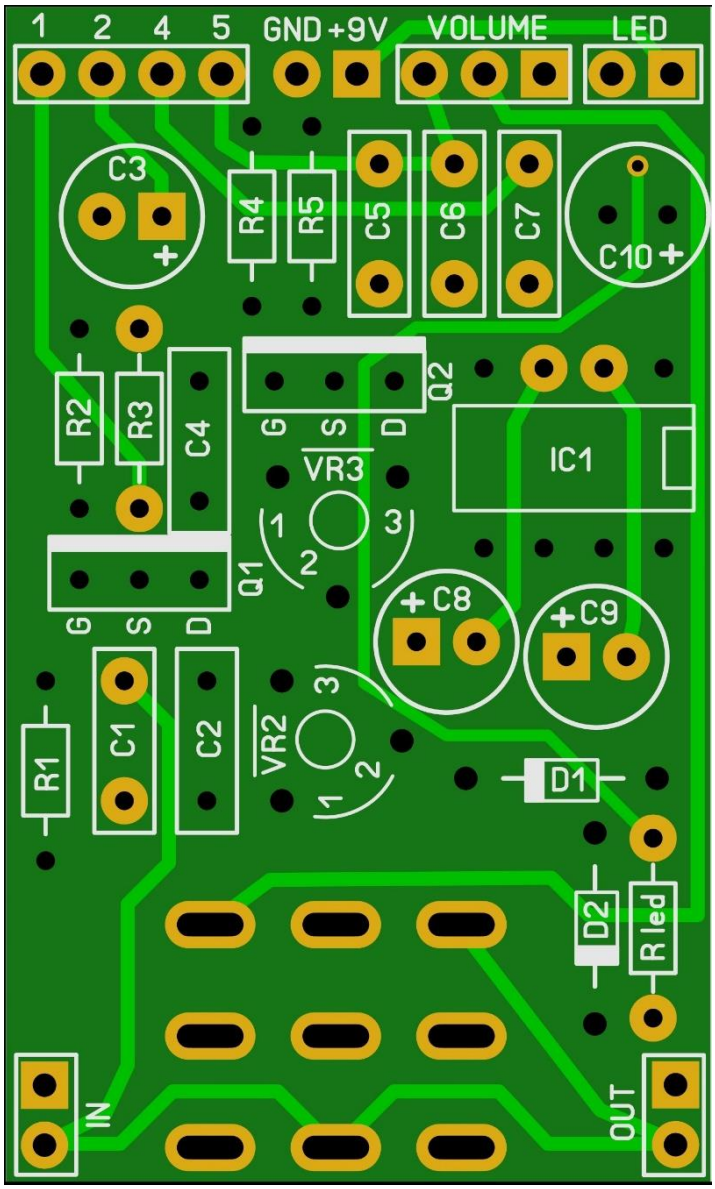
Before starting your build, read the entire document!

Last update: 17-10-2021

Components

Part	Value	Comment	Part	Value	Comment
C1	100p	Multilayer Ceramic	**IC1**	ICL7660S	
C2	220n	SMF	**Q1**	2N5484	
C3	10u	35V+ Electrolyte	**Q2**	2N5484	
C4	220n	SMF	**R1**	1M	
C5	220n	SMF	**R2**	1M	
C6	1n	MKT	**R3**	680R	
C7	15n	SMF	**R4**	1M	
C8	10u	35V+ Electrolyte	**R5**	680R	
C9	10u	35V+ Electrolyte	**R led**	3k3	
C10	10u	35V+ Electrolyte	**VOLUME**	B250K	(Alpha DPDT combi pot)
D1	1n5817		**VR2**	B25k	6mm trimpot
D2	1n5817		**VR3**	B25k	6mm trimpot

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

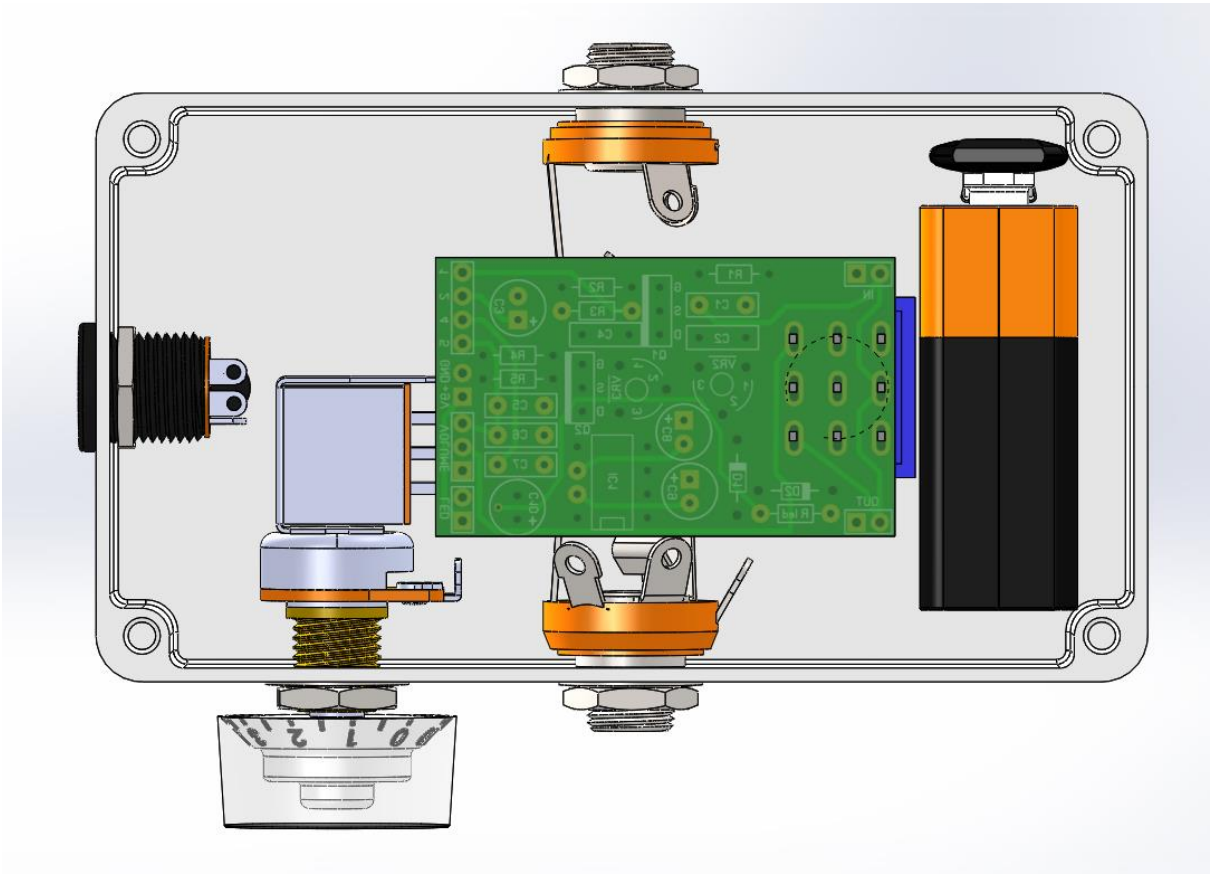


General guideline for components

- Capacitors: All values under 1nF should be ceramic disks. From 1nF up to 1uF should be SMF or MKT (foil/metal film capacitors) and over 1uF use electrolyte caps 35V+ rated and watch out for polarity!
- Resistors: use 1% metal film for the best results.
- Socket all transistors and the voltage regulator. This way you can easily mod them or replace them if they brake.
- Orientation of the transistors: the white stripe on the PCB indicates where the flat side of the transistor should be.

General building tips

Before starting, do note that height might become an issue during this build. Try to solder everything as close to the PCB as possible. It might be a good idea to solder the transistors to the board after you selected the best ones to your taste. If you want to keep them socketed then you'll need to bend them. Same goes for the electrolyte capacitors. So examine the maximum height of the components before starting your build. Also note that the 3PDT should can be placed on both the bottom as the top side of the PCB. The components should be placed on the top side. By default it is advised to mount the 3PDT on the bottom side, however If your enclosure is very small it might be a good idea to mount the 3PDT on the topside. Decide this before starting your build.



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 (not yet VR2 and VR3) and the diodes. Now solder the capacitors starting with the smallest. Next, solder the sockets for the transistors and voltage regulator. For the transistors you can buy a 20 pin SIL socket and cut of the pins you need.

If you want to make it a 100% copy of the original then you'll need to replace VR2 and VR3 with 4k7 resistors. Connect these resistors between pin 2 and 3 of the VR's. The Pins are clearly marked on the PCB. Else it is now time to solder VR2 and VR3. Try to pre-set them at 4k7 by measuring the resistance between pin 2 and 3. (PS the schematic on page 7 shows 10k pots but in practice 25k is much better for biasing different transistor types)

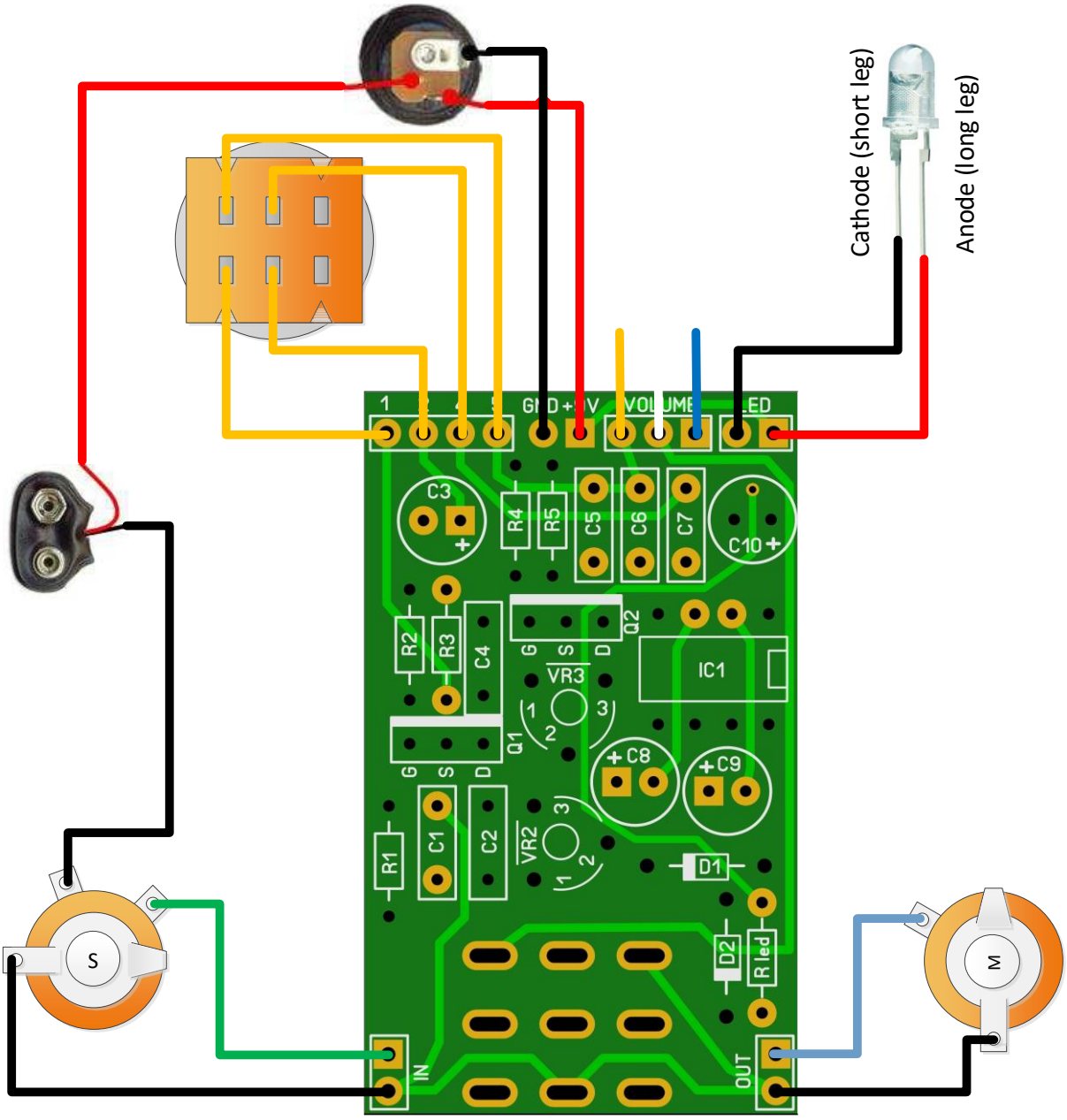
Finish by soldering the electrolyte capacitors. Place the transistors and you are almost ready to rock.

If you are going to use a 250k alpha pot with integrated DPDT than you will need to drill the hole **in the side** of the enclosure and not the top of the enclosure as the DPDT pots are too tall for a 1590B or BB enclosure!

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

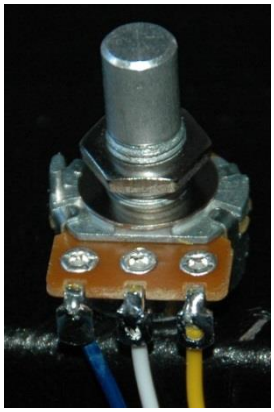
- **2 input jacks**. 2 mono jacks if you are not going to use a battery and only the 9V adapter. 1 mono (for output) and 1 stereo jack (for input) if you will be using both a 9V battery and the 9V adapter.
- **3PDT footswitch** (9 pins).
- **2,1mm DC jack** (isolated).
- **9v battery clip** (optional).
- **22 gage stranded hook-up wire.**
- **Hammond 1590B** case (or similar) in your favourite colour.

Offboard wiring



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 1k5 resistor, a normal Green requires 680R and an ultra-bright Green might need a 4k7.

The switch is drawn as in the downward position! So the middle lugs and the lowest pins (the pins without wires in the picture) are connected! Feel free to reverse the wiring if you prefer that.



Biasing transistors

To bias the boost to your liking you can turn trim pots VR2 and VR3 to get it set to your liking. The best way to start is to set them both to 4k7 between pin 2 and 3. This is the same value as the original and should work if you are using the 2N5484. Try to get the drain (D) voltage around 9V. But remember, if 9V does not sound good to you… try another value!

As alternatives you can try J201, 2N5485,2N5457, 2N5458, MPF102. It is reported however that the 2N5484 is still the best option.

NB. Even transistors of the same type may vary a lot in specs. I advise you to have at least 10 transistors per type to allow you to find the right ones for your circuit. So, if you cannot bias a 2N5484 to 9V, try another one!

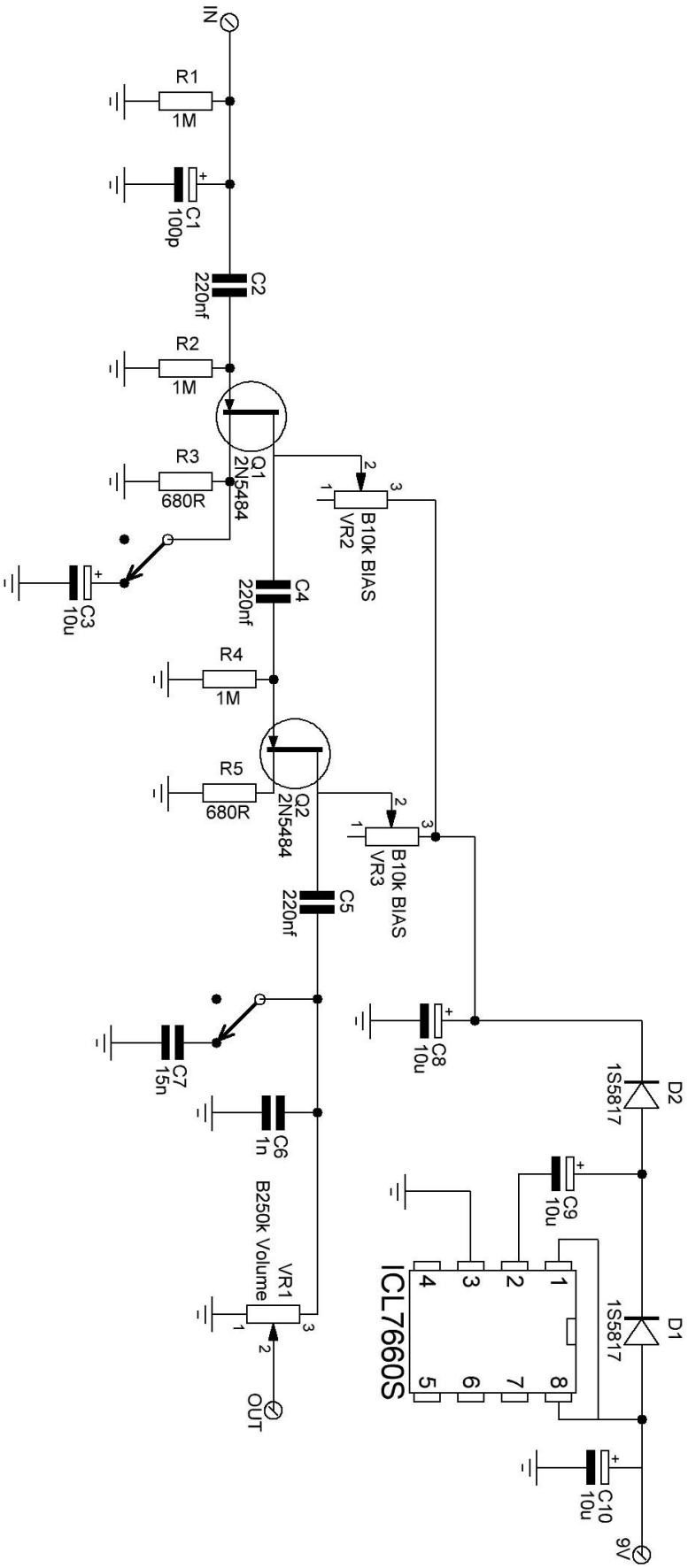
Troubleshooting

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.

Schematic


IN
R1
1M
C1
100p
C2
220nf
R2
1M
R3
680R
Q1
2N5484
B10k BIAS
VR2
C3
10u
C4
220nf
R4
1M
R5
680R
Q2
2N5484
B10k BIAS
VR3
C5
220nf
C7
15n
C6
1n
B250k Volume
VR1
OUT
C8
10u
D2
1S5817
C9
10u
ICL7660S
D1
1S5817
C10
10u
9V