Epic Polar Boost

Building instructions

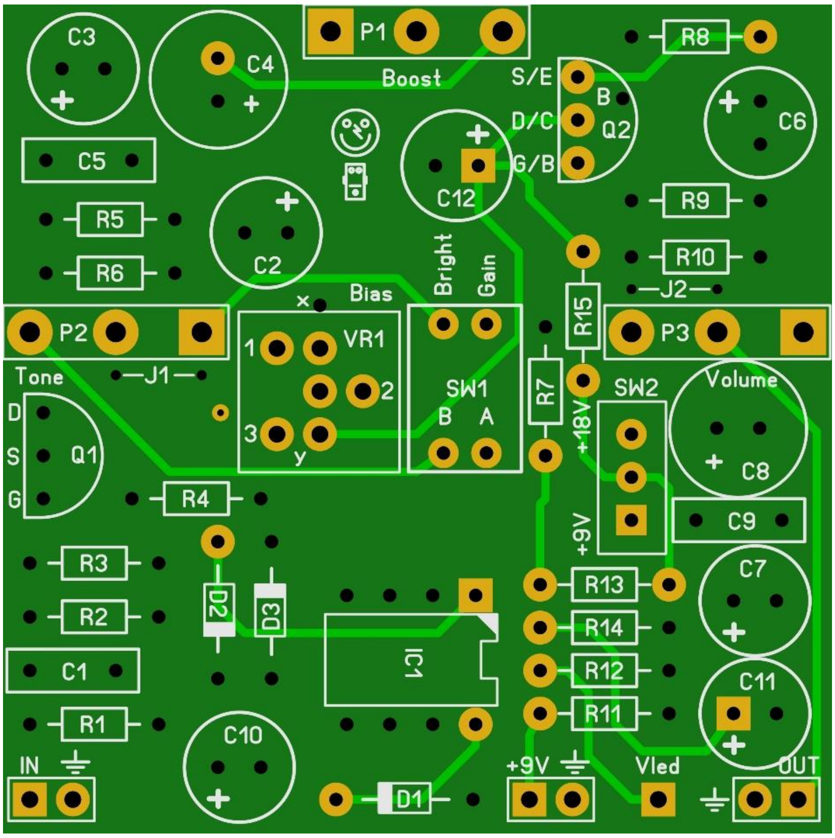
v1.0

Table of contents

Read this <u>entire</u> manual <u>thoroughly</u> before you start building the effect! There are a lot of available mods and you should choose which one you want to incorporate <u>before</u> starting your build.

Last update: 30-03-2019

PCB layout



Dimensions: 49 mm x 49 mm

1.93 inch x 1.93 inch

Components

Original configuration

ID	Value	Comment	ID	Value	Comment
C1	47n	SMF/MKT/Wima	**R1**	1M	1% metalfilm
C2	10u	Electrolytic 25V+	**R2**	33k	1% metalfilm
C3	10u	Electrolytic 25V+	**R3**	1M	1% metalfilm
C4	100u	Electrolytic 25V+	**R4**	4k7	1% metalfilm
C5	3n3	SMF/MKT/Wima	**R5**	1k	1% metalfilm
C6	10u	Electrolytic 25V+	**R6**	15k	1% metalfilm
C7	4u7	Electrolytic 25V+	**R7**	1M	1% metalfilm
C8	100u	Electrolytic 25V+	**R8**	10k	1% metalfilm
C9	100n	SMF/MKT/Wima	**R9**	47k	1% metalfilm
C10	10u	Electrolytic 25V+	**R10**	100R	1% metalfilm
C11	47u	Electrolytic 25V+	**R11**	33k	1% metalfilm
C12	47u	Electrolytic 25V+	**R12**	Jump	
D1	1N5817		**R13**	10k	1% metalfilm
D2	1N5817		**R14**	10k	1% metalfilm
D3	1N5817		**R15**	Jump	
IC1	ICL7660S		**SW1**	DIP2	Gain/Bright
P1	C10k	Boost	**SW2**	SPDT	Voltage
P2	NC		**VR1**	8k2	1% metalfilm
P3	NC		**J1**	Jump	
Q1	2N5457		**J2**	Jump	
Q2	2SC1815				

* Optional parts (see modifications section)

All parts need to be 25V+ rated

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

Jump means place a jumper in this spot instead of a component
NC means Not Connected, so leave it empty

The main effect is configured as the original. The only deviation is the added switchable 18V charge pump. If you do not want that then read the modifications section on how to leave it out.

Optimized configuration

ID	Value	Comment	ID	Value	Comment
C1	47n	SMF/MKT/Wima	**R1**	1M	1% metalfilm
C2	10u	Electrolytic 25V+	**R2**	33k	1% metalfilm
C3	10u	Electrolytic 25V+	**R3**	1M	1% metalfilm
C4	100u	Electrolytic 25V+	**R4**	4k7	1% metalfilm
C5	3n3	SMF/MKT/Wima	**R5**	470R	1% metalfilm
C6	10u	Electrolytic 25V+	**R6**	22k	1% metalfilm
C7	10u	Electrolytic 25V+	**R7**	1M	1% metalfilm
C8	100u	Electrolytic 25V+	**R8**	3k3	1% metalfilm
C9	100n	SMF/MKT/Wima	**R9**	NC	
C10	10u	Electrolytic 25V+	**R10**	Jump	
C11	47u	Electrolytic 25V+	**R11**	4k7	1% metalfilm
C12	100u	Electrolytic 25V+	**R12**	390R	1% metalfilm
D1	1N5817		**R13**	10k	1% metalfilm
D2	1N5817		**R14**	10k	1% metalfilm
D3	1N5817		**R15**	100R	
IC1	ICL7660S		**SW1**	DIP2	Gain/Bright
P1	C10k	Boost	**SW2**	SPDT	Voltage
P2	B50k	Tone	**VR1**	50k	trimpot
P3	B100k	Volume	**J1**	NC	
Q1	2N5457	or J201	**J2**	NC	
Q2	2N5457	or J201			

* Optional parts (see modifications section)

All parts need to be 25V+ rated

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

Jump means place a jumper in this spot instead of a component

NC means Not Connected, so leave it empty

Added **Tone** (high cut) control, **Volume** control and transistor bias (Q1) as well as some minor part optimizations. Read the modifications section for further tone control options.

Build 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.

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 (if needed) and resistors. You can create a jumper using a spare piece of lead from a resistor or diode. The original version puts a 8k2 resistor in **VR1**. For this there are 2 special pads **X** and **Y** marked on the PCB which will fit this resistor. Next comes the diodes.

If you want to experiment with another transistors for **Q1** and **Q2** then you could socket them instead of soldering them to the board. You'll need a 20 SIL sockets, break off the sockets and solder them to the board. Now is the time to solder the sockets for **IC1**. Place the transistors and IC not before you are finished with all soldering and off board wiring!

Now continue by soldering the SMF and MKT capacitors then solder the optional internal trimpot (**VR1**) and Dip2 for **SW1**. Finish with soldering the transistor (if not socketed) and the Electrolytics. Note that the orientation of the transistor may not match your transistors! For that the pads are marked with the correct pinout. **Q2** has an extra base (marked **B**) pad for transistors that are not ECB but EBC.

I suggest you now drill the holes in your enclosure so you can use it during the off board wiring and to correctly fit the pots to the backside of the PCB.

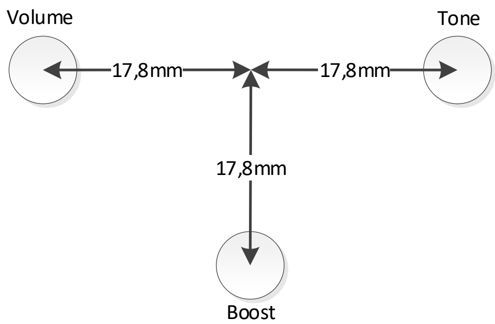
Note: Really take some time to determine where to place the pots, switches, jacks and PCB in the enclosure before you start drilling. **Measure twice, drill once**.

You are almost ready to rock, well… not really. The difficult part starts now.

Besides the components mentioned in the components table, you will need:

- **1 mono input jack socket, 1 stereo output jack socket**.
- **3PDT footswitch** (9 pins)
- **2,1mm DC socket** (isolated).
- **22 gage stranded hook-up wire.**
- **LED holder.** This enables you to mount the LEDs in the enclosure.
- **LED (3mm or 5mm depending on your taste).** This will be the status LEDs
- **Hammond 1590B** case (or similar) in your favorite color. This case will fit tightly and leaves no room for error! If you need more room you could consider using a **Hammond 1590N1 (=125B)**.

Drill template



Pots drill holes 7 mm

Switch drill holes 6 mm

Footswitch drill hole 13 mm

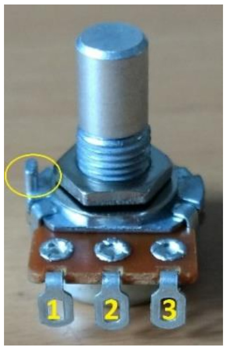
Jacks drill holes 10 mm

DC jack drill holes 12 mm

After drilling the holes, insert the potentiometers in the enclosure and then insert them in the back of the PCB.

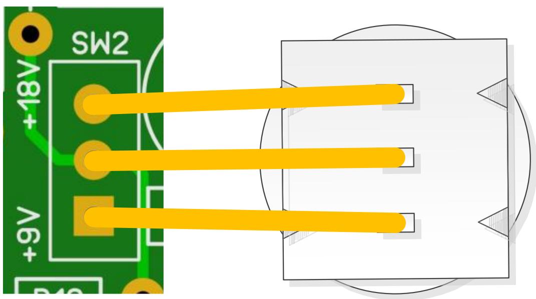
You can break off the pin I marked with the yellow circle with a small pair of pliers. The rectangle pad on the PCB marks the pad for **pin 1**.

If all fits well you can solder the potentiometers to the PCB.



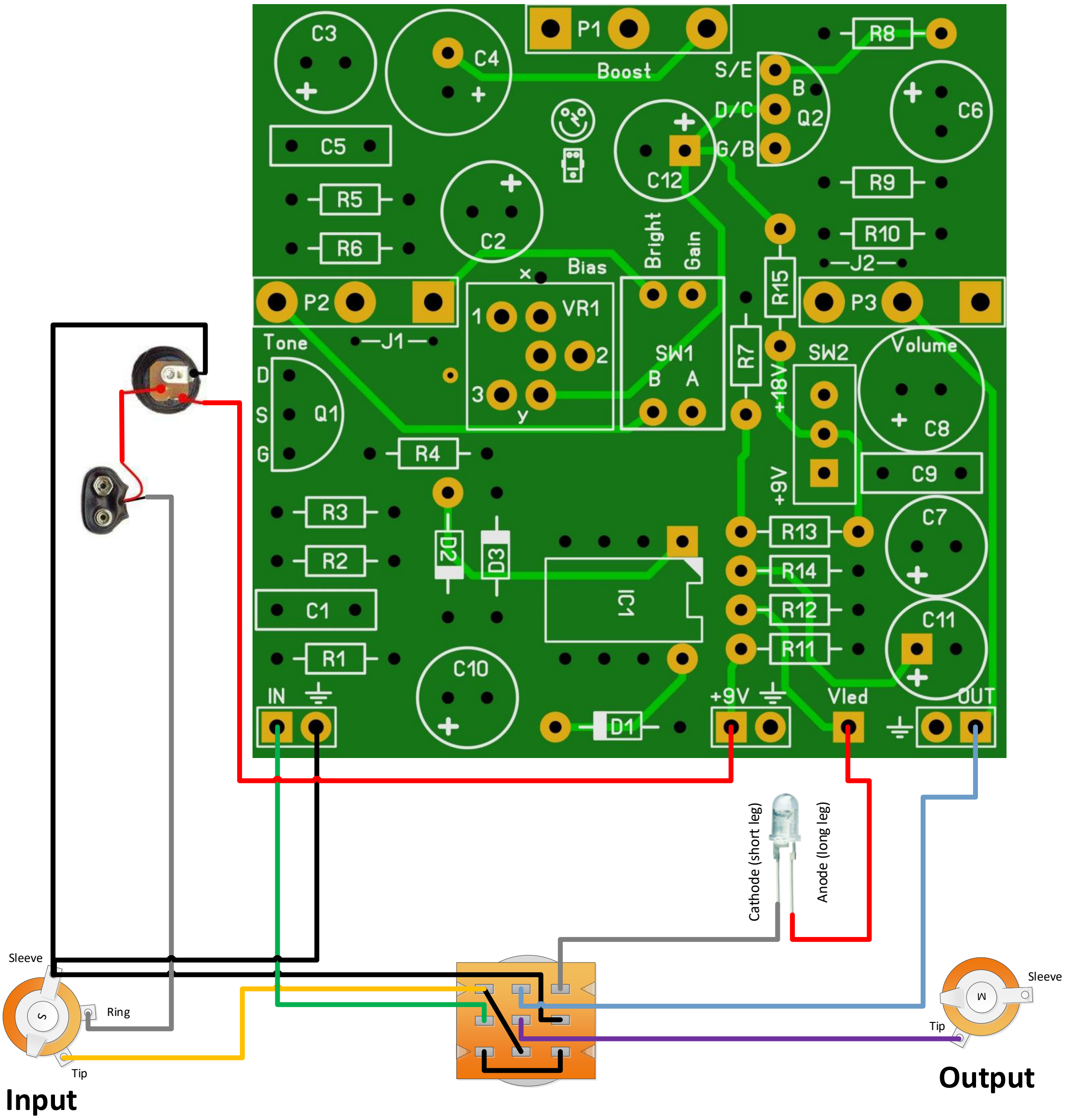
Switch

You can switch SW2 via a standard internal or external SPDT switch. You should connect it like this:



Manufacturers and product names are mentioned solely for circuit identification, and where applicable their trademarks are the property of their respective owners who are in no way associated or affiliated with the author. No cooperation or endorsement is implied.

Off board wiring



R11 is a **4k7** resistors. You can change the value depending on the type of LED you use but 4k7 is safe enough for almost all LEDs @9V.

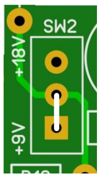
Note that the diagram is using star wiring for the ground connection. The sleeve of the output socket is not connected on purpose. It does however require a good contact between the output socket and enclosure to work. If this is not possible then connect the sleeve of the output socket to the sleeve of the input socket.

You should use Switchcraft/Neutrik type metal chassis sockets. When using plastic, you will need to connect the input socket sleeve to the output socket sleeve!

Modifications

Charge pump

If you want to leave out the charge pump then do not solder **C8**, **C9**, **C10**, **D2**, **D3** and **IC1**. You must solder a jumper between pad 1 and 2 on **SW1** like this:



You could also change **D1** back to a 1N4148 like the original. **D1** has no tonal function so it would be for optics/purism only.

Tone section

This is the interesting part and yes we are going to get a bit techy. Let's start by a simple explanation of the original tone part in the boost.

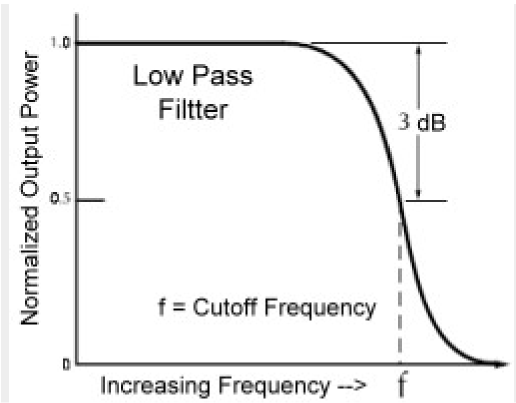
First of some EQ definitions:

Bass	60 Hz - 250 Hz
Low Mids	250 Hz – 500 Hz
Mids	500 Hz – 2kHz
High Mids	2 kHz – 4 kHz
Treble	4 kHz – 6 kHz

R5 (1k), **R6** (15k) and **C5** (3n3) form a fixed *high cut/low pass RC filter* with a cut off frequency of:

$$f_c = \frac{1}{2\pi RC} = \frac{1}{2\pi \cdot 16000 \cdot 3.3\, e^{-9}} \approx 3kHz$$

Note: **R5** and **R6** are in series so **R = R5+R6** = 1k + 15k= 16k



So the original effect cuts off the top high mids and treble part of the signal.

When you switch the bright switch on (**SW1B**) the resistance is cut to 1k since the switch bypasses **R6**. So the cutoff frequency changes to:

$$f_c = \frac{1}{2\pi RC} = \frac{1}{2\pi \cdot 1000 \cdot 3.3\,e^{-9}} \approx 48kHz$$

So now the signal cuts off in the *very high treble* section which is almost not audible, but something else also changed.

By lowering the resistance the gain of Q2 also changed! The effect is now brighter and also a bit louder (approx. 3db).

By adding a tone control to the effect you can change the bandwidth of the cut off frequency.

Mark Hammer introduced the "*Stupidly Wonderful Tone Control*" (SWTC) in 2005 which combines a simple adjustable RC filter with the ability to keep the gain constant. This is done in the optimized version of the boost by introducing **P2** (B50k). If you want to remain close to the original boost but still want to be able to change the cut off frequency, I changed **R6** to 22k.

This way the 50 pot and 22k resistor in parallel give $R_t = \frac{1}{\frac{1}{50000}+\frac{1}{22000}} \approx 15k$ which is about the same as **R6** in the original. Now the tone control gives you a sweep between 3kHz and 48kHz while not changing the level of boost while turning the knob! The bright switch (SW1B) will act a bit different than the original as is will take out R6,P2 and C5 all at once, just leaving **R5** in the circuit. This way there is no RC filter left, just a small extra boost. Changes are that high frequency interference is more audible when you switch **SW1B** on.

If you use a jumper in **J1** the filter will act more like the original and will cut some volume when you turn it up (=cut highs). However the bright switch will function exactly as in the original. To end this lecture, some suggestions to make the tone control even more usable:

Author	R5	R6	P2	C5	*fc* min	*fc* max
LG	1k	22k	B50k	3n3	2,9kHz	48kHz
LG	1k	NC	B25k	3n3	1,85kHz	48kHz
LG	1k	NC	B50k	3n3	945 Hz	48 kHz
LG	1k	NC	B100k	3n3	477 Hz	48kHz
F Briggs	470R	NC	B10k	10n	1,5kHz	33,8kHz

NC= not connected

NB. When using J1 "jumped", the higher P2 the bigger the volume drop when turned down.

Transistors

The original uses a **2N5457** as **Q1** and **2SC1815** as **Q2**. A lot of people prefer a **J201** as **Q1** since it has a bit more gain and a **2N5088** as **Q2**. Check the pinout of your transistors before you solder them! The correct pinout is printed on the PCB and insert the transistors accordingly. You can always experiment with other types of transistors and if you find better combinations, please let me know!

Calibration

Calibration is the part where you determine your personal bias sweet spot on **Q1** and so determining the maximum amount of boost. This section is only applicable if you chose to use **VR1** instead of a fixed 8k2 resistor.

Set all the knobs to max (boost and when applicable volume and tone) and turn **VR1** until you get to the spot that you like best as *maximum* boost. This is up to personal taste and I always prefer to set it by ear, but you could start by setting **VR1** to a resistance of 8k2 between legs 1 and 2 *before* you solder it to the PCB. This way you start of at the original value.

Note that it is normal that you will turn the volume all the way up as well as tone. Since both controls cut the signal or frequency, the best way to start is by turning them all the way up and if you want some highs cut, turn down the tone knob. Ditto if you want a bit less volume then turn down the volume knob. Also note that logic dictates that the volume and boost knobs interact…

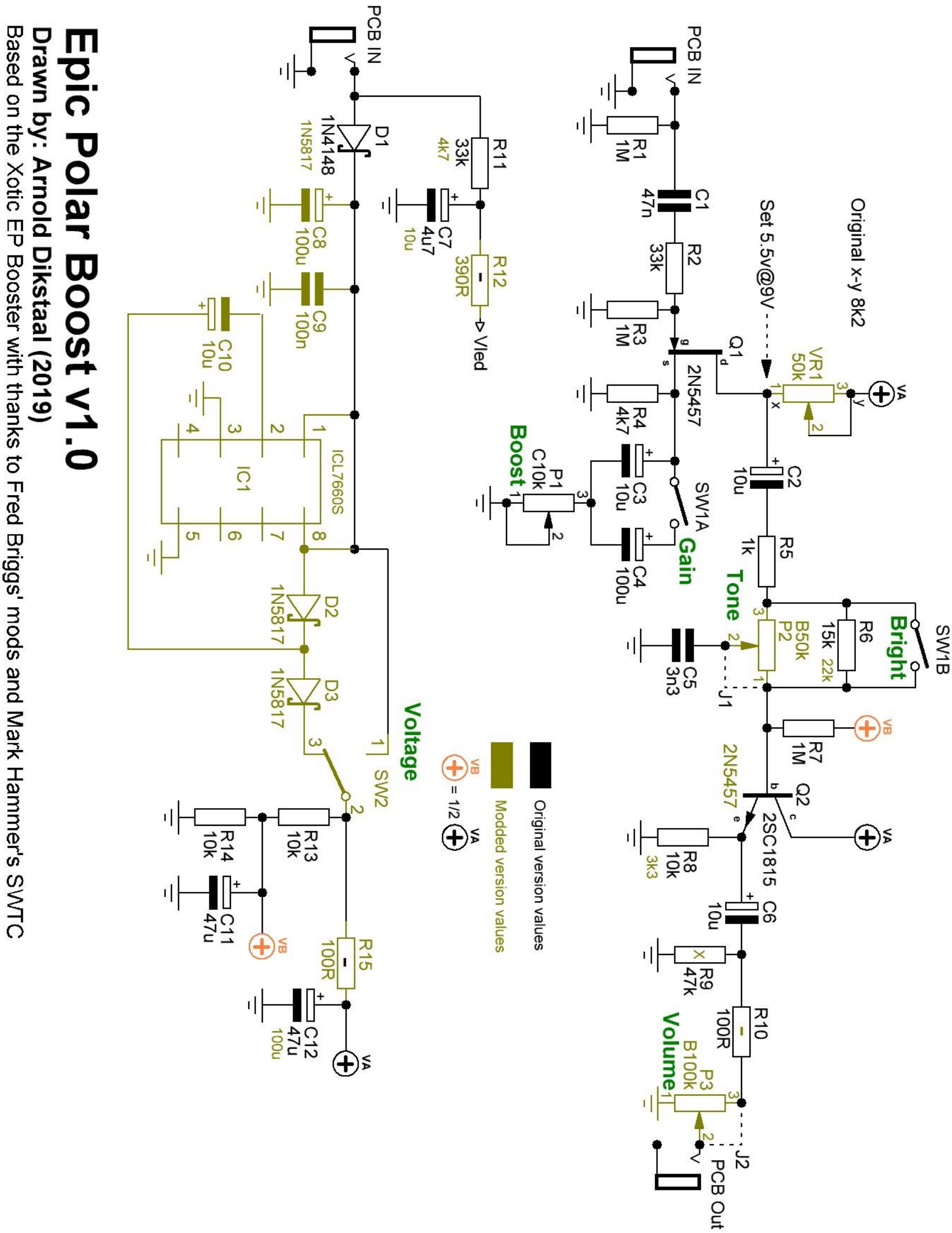
Troubleshooting

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.

Schematics



Epic Polar Boost v1.0
Drawn by: Arnold Dikstaal (2019)
Based on the Xotic EP Booster with thanks to Fred Briggs' mods and Mark Hammer's SWTC
Original version values
Modded version values
VB = 1/2 VA
Original x-y 8k2
Set 5.5v@9V
PCB IN
PCB Out
R1 1M
C1 47n
R2 33k
R3 1M
Q1 2N5457
R4 4k7
VR1 50k
C2 10u
C3 10u
C4 100u
SW1A
Gain
P1 C10k
Boost
R5 1k
R6 15k 22k
SW1B
Bright
Tone
P2 B50k
C5 3n3
J1
R7 1M
Q2 2N5457 2SC1815
R8 10k 3k3
C6 10u
R9 47k
R10 100R
Volume
P3 B100k
J2
R11 33k 4k7
D1 1N4148 1N5817
C7 4u7 10u
R12 390R
Vled
C8 100u
C9 100n
C10 10u
IC1 ICL7660S
D2 1N5817
D3 1N5817
Voltage
SW2
R13 10k
R14 10k
C11 47u
R15 100R
C12 47u 100u