Sultan of Tone

Building instructions

V4.1

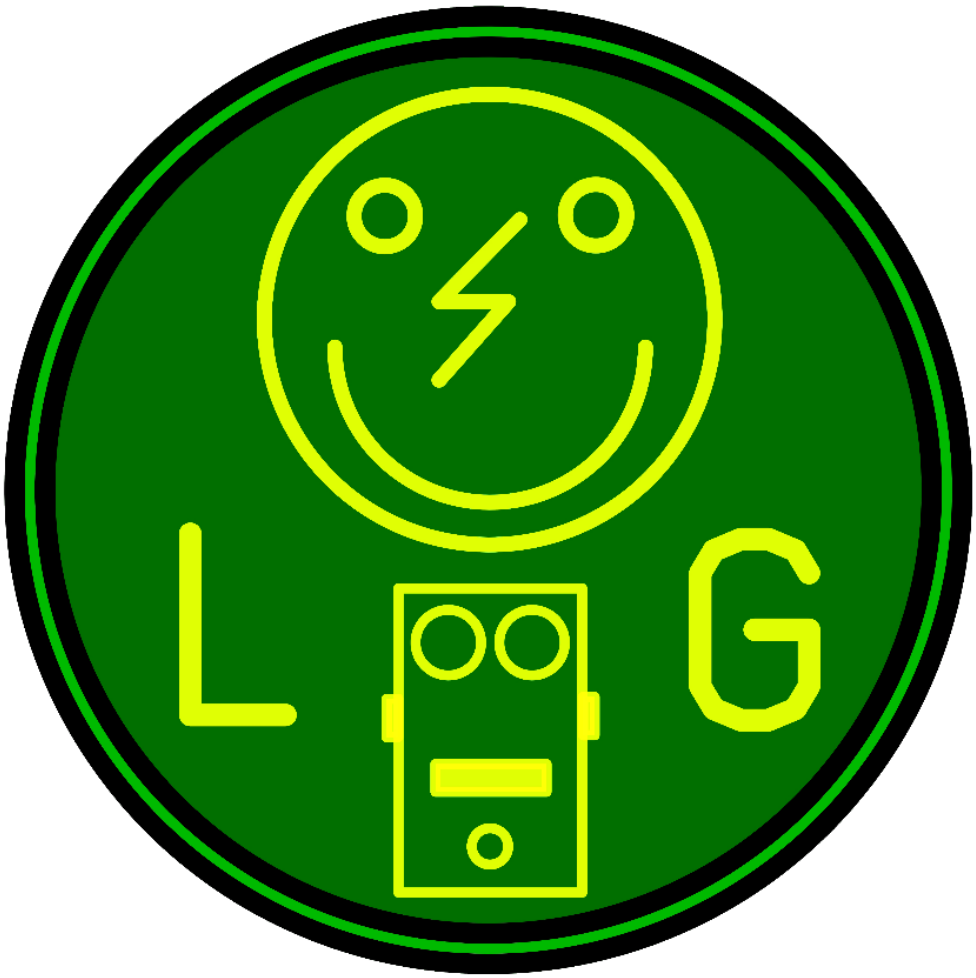


Table of contents

Components 3
Bill of Materials 4
PCB layout 5
Power section 6
Building sequence 7
Modifications 8
Off board wiring and drilling 9
Diode switch settings 10
Troubleshooting 11
Schematic 12

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

Last update: 28-09-2017

Components

All Components must be rated 25V+

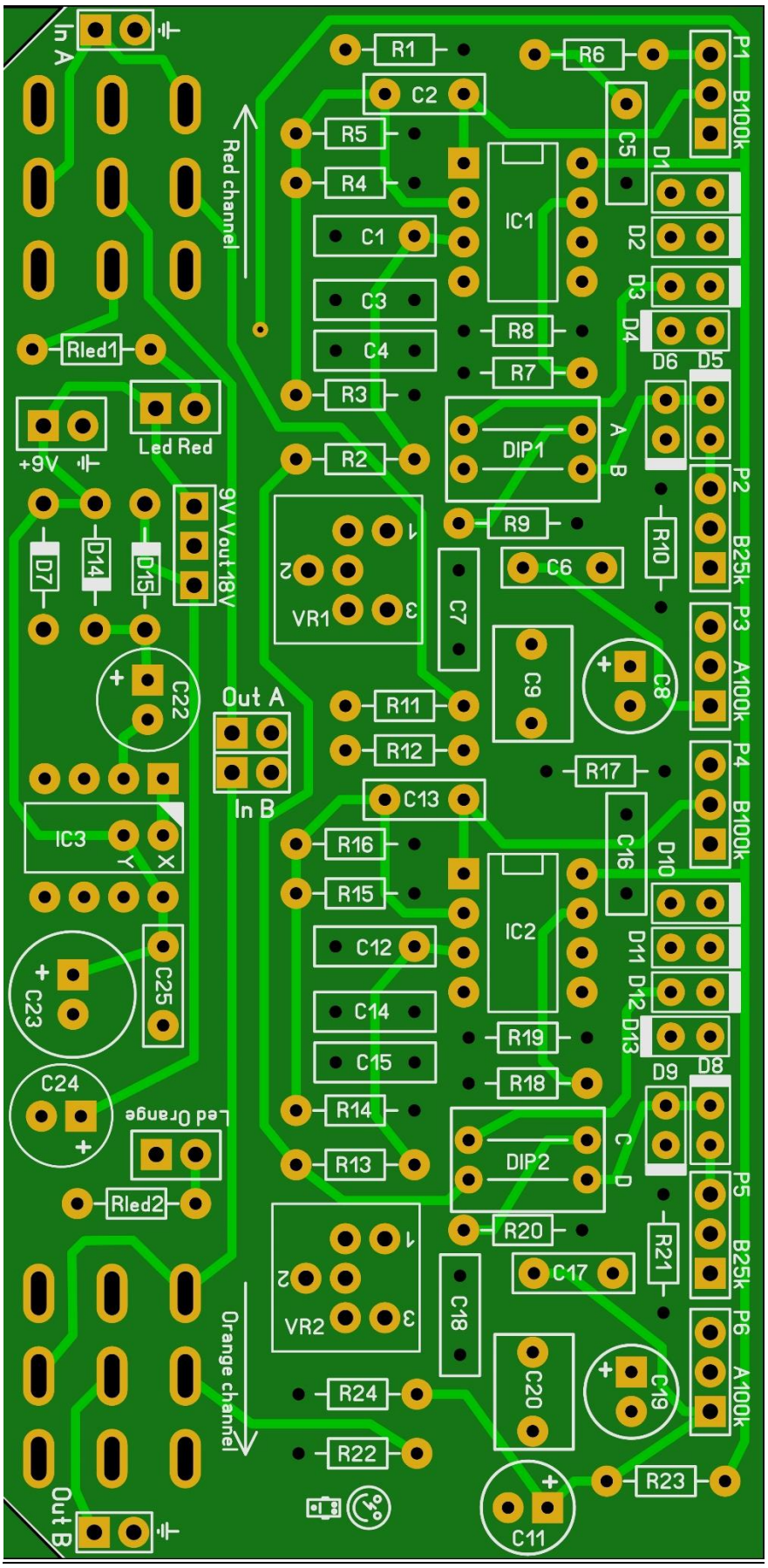
Name	Value	Comment	Name	Value	Comment
C1	10n	SMF	**R1**	1M	
C2	100p	Ceramic/SML	**R2**	1M	
C3	10n	SMF	**R3**	27k	
C4	10n	SMF	**R4**	33k	
C5	100n	SMF	**R5**	10k	
C6	10n	SMF	**R6**	10k	
C7	10n	SMF	**R7**	220k	
C8	1u	Electrolyte	**R8**	6k8	
C9	1u	SMF	**R9**	1k	
C11	100u	Electrolyte	**R10**	6k8	
C12	10n	SMF	**R11**	1M	
C13	100p	Ceramic/SML	**R12**	1M	
C14	10n	SMF	**R13**	1M	
C15	10n	SMF	**R14**	27k	
C16	100n	SMF	**R15**	33k	
C17	10n	SMF	**R16**	10k	
C18	10n	SMF	**R17**	10k	
C19	1u	Electrolyte	**R18**	220k	
C20	1u	SMF	**R19**	6k8	
C22	10u		**R20**	1k	
C23	100u	Electrolyte	**R21**	6k8	
C24	10u	Electrolyte	**R22**	1M	
C25	100n	MKT	**R23**	47k	
			R24	47k	
			R led1	3k3	For Red LED
Led Orange		3mm	**R led2**	3k3	For Orange LED
Led Red		3mm			
D1	MA856	Or MA858	**P1**	B100k	Gain Red Channel
D2	MA856	Or MA858	**P2**	B25k	Tone Red Channel
D3	MA856	Or MA858	**P3**	A100k	Volume Red Channel
D4	MA856	Or MA858	**P4**	B100k	Gain Orange Channel
D5	1S1588		**P5**	B25k	Tone Orange Channel
D6	1S1588		**P6**	A100k	Volume Orange Channel
D7	1N4001		**VR1**	B50k	Treble boost Orange Channel
D8	1S1588		**VR2**	B50k	Treble Boost Red Channel
D9	1S1588				
D10	MA856	Or MA858	**DIP1**		2 position dip switch
D11	MA856	Or MA858	**DIP2**		2 position dip switch
D12	MA856	Or MA858			
D13	MA856	Or MA858	**IC1**	JRC4580D	
D14	1N5817		**IC2**	JRC4580D	
D15	1N5817		**IC3**	LT1054/ICL7660s	

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

Bill of Materials

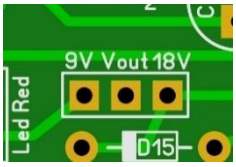
Capacitors			
Value	**Amount**	**Type**	**Rating**
10n	10	SMF	25V or higher
100n	2	SMF	25V or higher
100n	1	MKT	25V or higher
1u	2	SMF	25V or higher
100p	2	Ceramic Multi Layer	25V or higher
1u	2	Electrolyte	25V or higher
10u	2	Electrolyte	25V or higher
100u	2	Electrolyte	25V or higher
Diodes			
Value	**Amount**	**Type**	
1N5817	2		
1N4001	1		
1S1588	4		
MA856	8		
LED	1	Red 3mm	
LED	1	Orange 3mm	
IC			
Value	**Amount**	**Type**	
LT1054 or ICL7660s	1	See power section!	
JRC4580D	2		
Resistors			
Value	**Amount**	**Type**	
1k	2	metalfilm 1%	
3k3	2	metalfilm 1%	
6k8	4	metalfilm 1%	
10k	4	metalfilm 1%	
27k	2	metalfilm 1%	
33k	2	metalfilm 1%	
47k	2	metalfilm 1%	
220k	2	metalfilm 1%	
1M	6	metalfilm 1%	
A100K	2	16 mm alpha pot	
B100k	2	16 mm alpha pot	
B25K	2	16 mm alpha pot	
B50k	2	6 mm trimpot	
Switches			
Value	**Amount**	**Type**	
DIP2	2	2 position DIP switch	

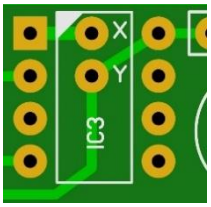
PCB layout


In A
Red channel
R1
R6
C2
R5
R4
C1
C3
C4
R3
R2
C5
D1
D2
D3
D4
D6
D5
IC1
R8
R7
DIP1
A
B
P1
B100k
P2
B25k
P3
A100k
P4
B100k
Rled1
Led Red
+9V
9V Vout 18V
D7
D14
D15
VR1
C7
C6
R9
R10
C9
C8
C22
Out A
In B
R11
R12
C13
R17
C16
D10
D11
D12
D13
D9
D8
IC3
R16
R15
C12
C14
C15
R14
R13
IC2
R19
R18
DIP2
C
D
P5
B25k
P6
A100k
C23
C25
C24
Led Orange
Rled2
VR2
C18
R20
C17
R21
C20
C19
R24
R22
Orange channel
R23
C11
Out B

Power section

This board has a built-in charge pump which can feed the board with 18V. This can be made into a switchable option! If you look at the 3 power pins (marked 9V/Vout/18V), you can use a 3 pin male header and a shunt to choose between the two voltage (9V/18V). If you short the left 2 pins you will feed the effect with 9V. If you short the right 2 pins, you will feed the board with 18V. Be careful not to connect all 3 pins at the same time! This will short the power section with all dramatic consequences. Of course you can use a SPDT switch instead of a 3 pin male header to make the power option externally switchable.





IC3 is the heart of the charge pump. You can use a LT1054 or a ICL7660s.

- **LT1054**: Do not connect pads X and Y.
- **ICL7660s**: Short pads X and Y with a spare piece of lead wire.

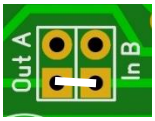
Note: Take special care building the power section. If the IC (LT1054/ICL7660s) heats up then check the polarity of the capacitors (C22,23 and 24) and check that the power selection pins do not short all 3 together.

If you do not want the 18V option and stick to the standard 9V option, then skip D14, D15, C22, C24, C25 and IC3. You will also need to short the **9V** and **Vout** pads on the power pins (as depicted in the left image above)

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. If you want to experiment with other diodes than 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.

This board gives you the option to separate both channels with their own input and output. If you do not want that and keep the stock option then solder the output of channel A to the input of channel B:



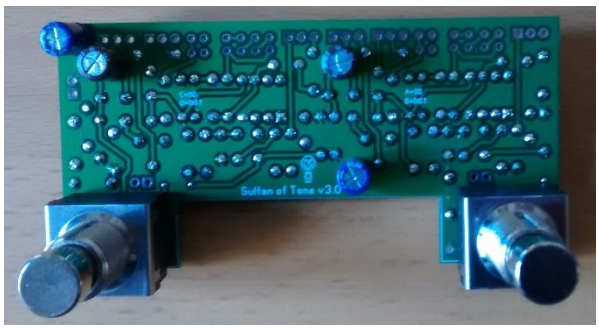
Start by soldering the small resistors and then the diodes. All diodes need to be soldered in an upright position except for **D7**, **D14** and **D15**. Again, If you want to socket the diodes then skip them till you start soldering the sockets.

Note: Diodes are fragile!! Do not expose them to heat for too long or they'll break. Do not blow on your solder in an attempt to cool it down. That will possibly result in a bad join that might corrode!

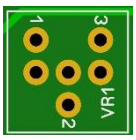
Solder the small ceramic capacitator **C2** and **C13** next.

Solder the 10n and 100n SMF/MKT capacitors. Next, solder the IC sockets (and diode sockets), then the DIP switches (only if you do not intend to use the external diode mod on page 9 else leave out the dipswitches!) and lastly solder the 1u SMF and electrolyte capacitors.

NB. It might be best that you solder the big electrolytes on the bottom side of the PCB instead of the top side. This will save you a lot of space. Measure first, then solder!!!



Finally solder the presence/treble boost trim pots **VR1** and **VR2**. If you want to control the presence externally then do not solder **VR1** and **VR2** but use B50k Alpha 16mm pots. For your convenience I marked the pinning on the PCB. The pots default setting is all the way to the right. To boost treble, turn the pots to the left.



Place the IC (and diodes) and you are almost ready to rock.

Besides the components mentioned in the bill of materials, 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.
- **2 x 3PDT footswitch** (9 pins).
- **2,1mm DC jack** (isolated).
- **9v battery clip** (optional). I advise not to use it as there is very little space left in the enclosure.
- **22 gage stranded hook-up wire.**
- **Hammond 1590BB** case (or similar) in your favorite color. A **Hammond 1590DD** or **Hammond 1590XX** will give you more room to build especially when using an external presence pot.

Modifications

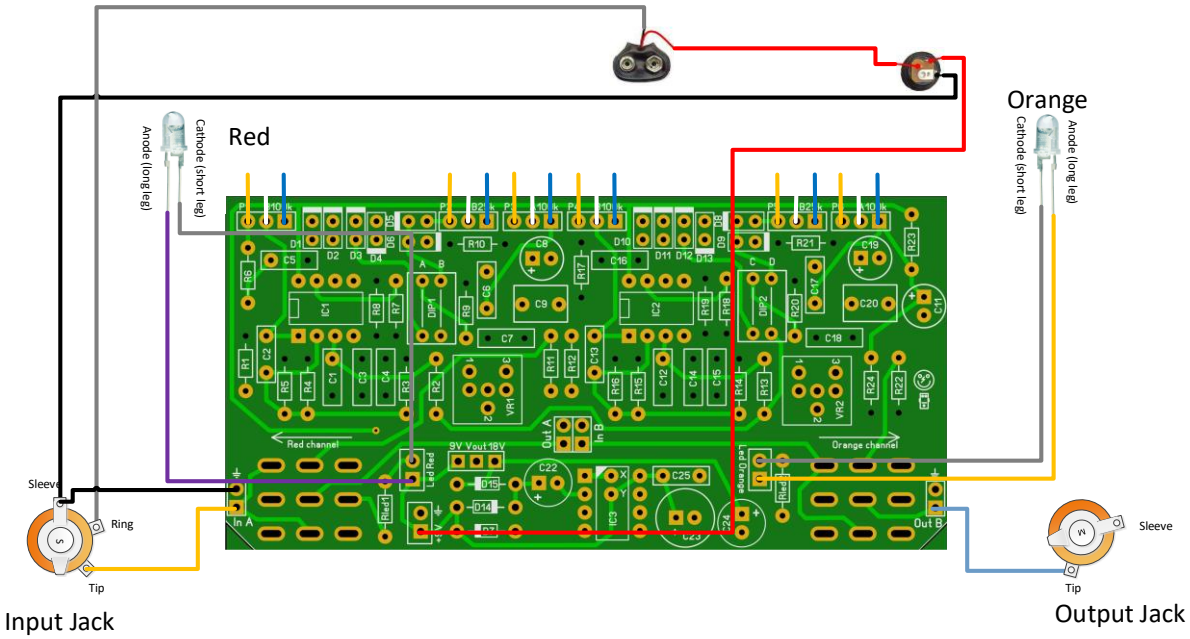
- You can experiment with different diodes for **D1**-**D6** and **D8**-**D13**. MA856/MA858 and 1S1588 are very hard to find and there are a lot of fakes sold from China. An alternative for MA856 is BA282. For 1S1588 there are 1n4148, 1n914, but also think outside the box with 1N4001, BAT41, BAT46, LED's etc. If you want to experiment then socket the diodes!
  Watch out with using Germanium diodes. The volume drop might be too big.
- Replacing C1, C6, C12 and C167 with 22nF instead of 10nF will give you some more low end grunt.
- You can also choose to do 1 side MA856 and the other 1S1588. For instance make **D1**-**D6** all 1S1588 and **D8**-**D13** all MA856.
- For the High Gain mod just switch out the B100k P1 (and/or P5) for a B250k.
- You can also try different OpAmps like TL072, CA3240E, OPA2134, AD712, NE5532, JRC4558 etc.

Off board wiring and drilling

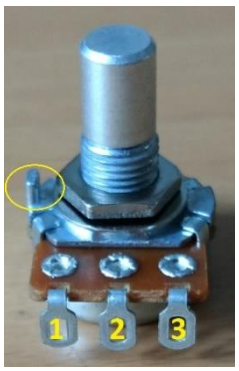
This version of the Sultan of Tone has integrated 3PDT print. This will save you time for off board wiring, but It will require you to drill more precise!

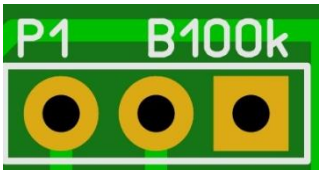
Before you start you must drill the 2 footswitch holes in your enclosure. Heart to heart distance of the footswitch holes is **76,65 mm**. When using a Hammond 1590BB I suggest you drill a **13mm** hole at **75,5mm from the top** and **17,7 mm from the left** of the enclosure. The other **13mm** hole should be at **75,5mm from the top** and **94,3 mm** from the left of the enclosure.

Now put the 3PDT footswitches in the enclosure and tighten them secure. Place the PCB on the footswitches and solder them. Remove the footswitches (and thus the PCB) and drill the holes for the pots and switches. The rest of the pots and LEDs can be put anywhere you want, just remember that you probably soldered the electrolyte caps on the bottom side so measure before drilling!



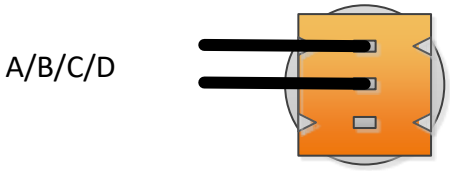
Soldering the potentiometers on P1-P6 is quite simple. The square pad marks pin 1 of the potentiometer. You can break the pin off the potentiometer circled in the bottom left picture.





Diode switch settings

Instead of using the dipswitches, you can make either 4 external SPDT switches for or 2 DP3T switches. If you choose 4 separate switches, it will look something like this:



Switches A, B, C, D can all be connected the same way as shown in the picture above. All ways of connecting both wires to the switch are valid as long as you do not connect it to both outer lugs.

The switch settings are independent of using DIP or separate switches and are as follows:

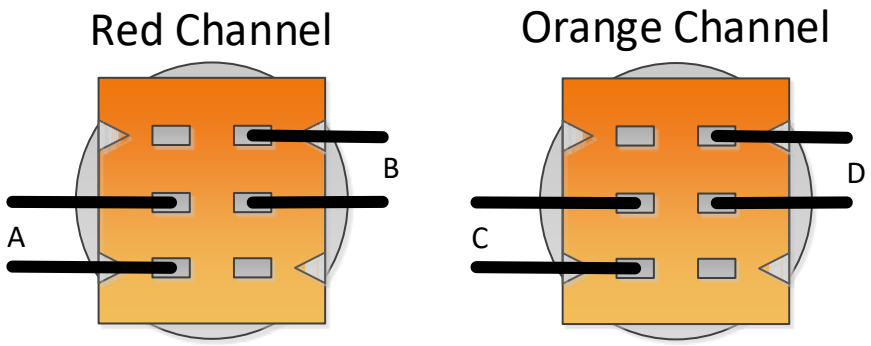
Channel	**Red**		**Orange**	
Switch	**A**	**B**	**C**	**D**
Clean	OFF	OFF	OFF	OFF
OD	ON	OFF	ON	OFF
Distortion	X	ON	X	ON

X = setting is irrelevant, either ON or OFF

So If you want an OD Red channel and a distortion Orange channel then it is:

A:ON **B**: OFF **C**: X **D**:ON

You can also use a DP3T switch per channel to select OD (Up)/Clean (Middle)/Distortion (Down). You will need an ON/OFF/ON switch for that and wire it like this:



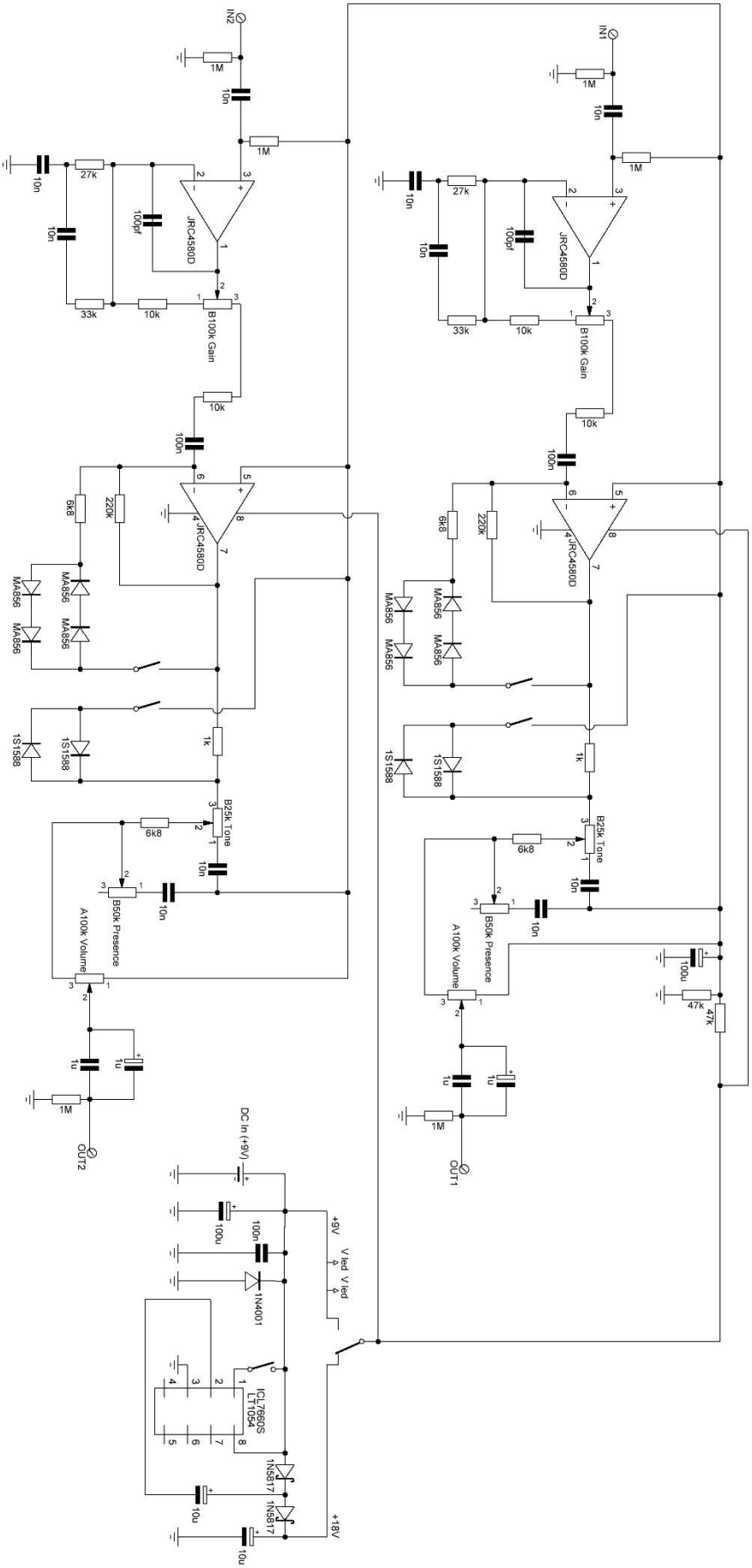
Troubleshooting

All PCB's have been 100% factory e-tested and out of every batch I receive I build a 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 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 loose or cold solder can be really bad for your board. Also check if switch enclosures do not short connections as most switches have a metal enclosure.
- Replace the IC's and transistors, one might be defective. Before doing that first unplug the 9V and wait 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


IN2
IN1
1M
10n
27k
100pf
JRC4580D
33k
10k
B100k Gain
100n
6k8
220k
MA856
1S1588
1k
B25k Tone
B50k Presence
A100k Volume
1u
100u
47k
OUT2
OUT1
DC in (+9V)
+9V
V led
1N4001
ICL7660S
LT1054
1N5817
10u
+18V