Supreme Leader

Building instructions

v1.0

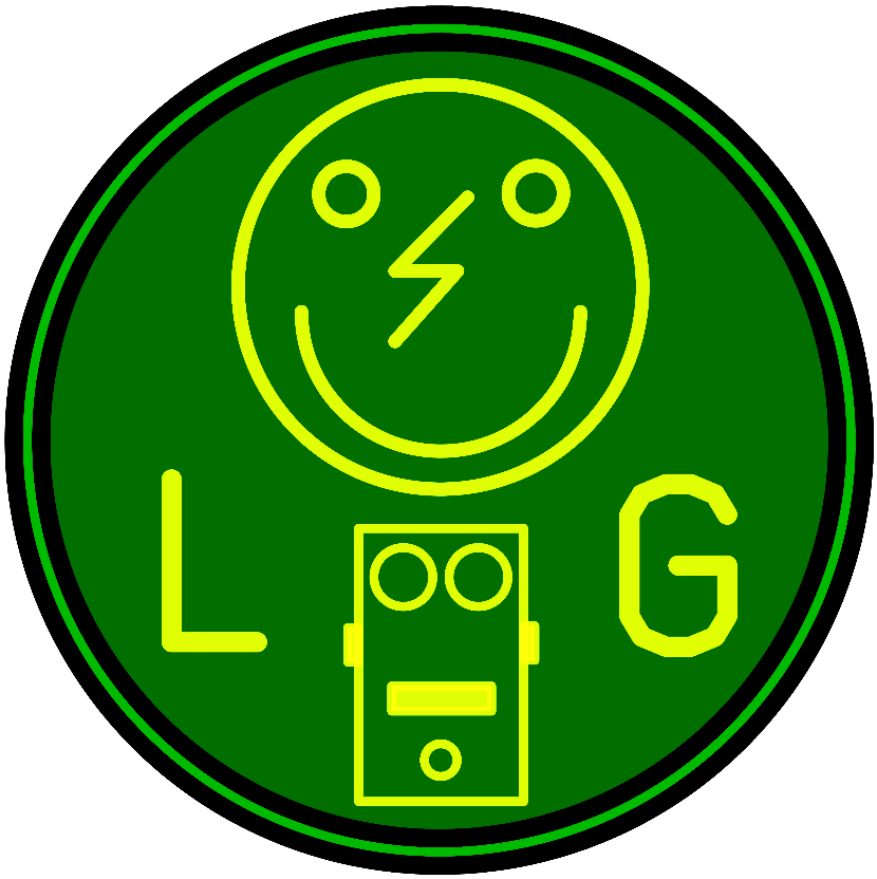
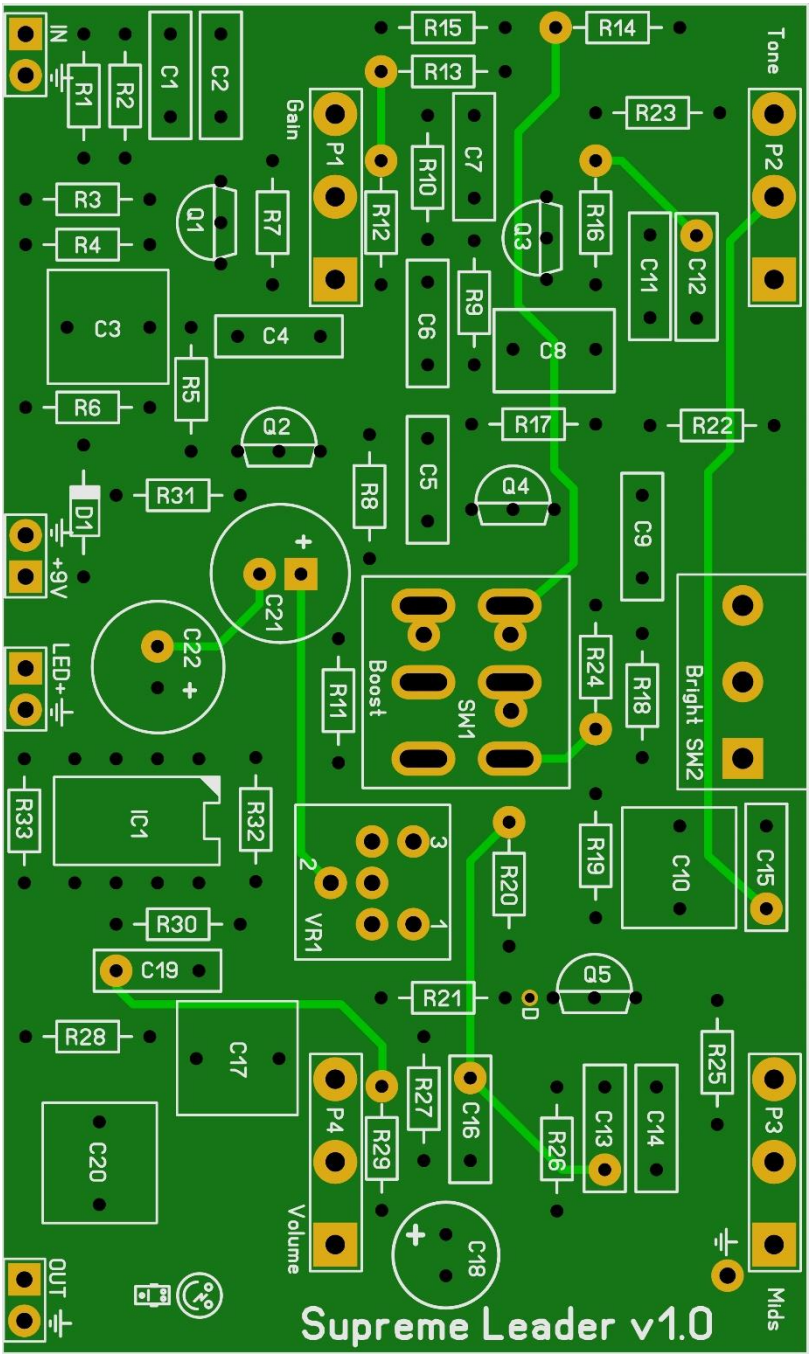


Table of contents

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: 24-11-2019

PCB layout



Dimensions: **83 mm x 49,55 mm**
3.27 inch x 1.95 inch

Components

Name	Value	Comment	Name	Value	Comment
C1	680p	MLCC	**R1**	1k	1% metalfilm
C2	47n	SMF/MKT/Wima	**R2**	1M	1% metalfilm
C3	1u	SMF/MKT/Wima	**R3**	1M	1% metalfilm
C4	56n	SMF/MKT/Wima	**R4**	10k	1% metalfilm
C5	22n	SMF/MKT/Wima	**R5**	1k	1% metalfilm
C6	51p	MLCC/Silver Mica	**R6**	1M	1% metalfilm
C7	220p	MLCC	**R7**	1M	1% metalfilm
C8	220n	SMF/MKT/Wima	**R8**	1M	1% metalfilm
C9	22n	SMF/MKT/Wima	**R9**	1M	1% metalfilm
C10	1u	SMF/MKT/Wima	**R10**	220k	1% metalfilm
C11	10n	SMF/MKT/Wima	**R11**	6k8	1% metalfilm
C12	10n	SMF/MKT/Wima	**R12**	4k7	1% metalfilm
C13	4n7	SMF/MKT/Wima	**R13**	15k	1% metalfilm
C14	33n	SMF/MKT/Wima	**R14**	1k	1% metalfilm
C15	6n8	SMF/MKT/Wima	**R15**	1M	1% metalfilm
C16	3n3	SMF/MKT/Wima	**R16**	1M	1% metalfilm
C17	1u	SMF/MKT/Wima	**R17**	1M	1% metalfilm
C18	10u	Electrolytic 10V+	**R18**	47k	1% metalfilm
C19	51p	MLCC/Silver Mica	**R19**	100k	1% metalfilm
C20	1u	SMF/MKT/Wima	**R20**	2k2	1% metalfilm
C21	220u	Electrolytic 10V+	**R21****	27k	1% metalfilm
C22	100u	Electrolytic 10V+	**R22**	33k	1% metalfilm
D1	1N5817		**R23**	68k	1% metalfilm
IC1	JRC4580D		**R24**	6k8	1% metalfilm
P1*	A500k	Gain	**R25**	10k	1% metalfilm
P2*	B100k	Tone	**R26**	10k	1% metalfilm
P3*	A50k	Mid	**R27**	47k	1% metalfilm
P4*	A100k	Volume	**R28**	1M	1% metalfilm
Q1	J201		**R29**	33k	1% metalfilm
Q2	J201		**R30**	100k	1% metalfilm
Q3	J201		**R31**	10R	1% metalfilm
Q4	J201		**R32**	10k	1% metalfilm
Q5	J201		**R33**	10k	1% metalfilm
SW1	DP3T	Boost (On-Off-On)	**VR1**	50k	Bias Q5
SW2*	SPDT	Bright (On-On)			

*** PCB mountable components. SW1 is a regular DP3T, with normal solder lugs.**

**** Leave out R21 when using VR1 to bias Q5, or leave out VR1 when using R21 and not bias Q5**

Build sequence

Read the modifications section before you start building!

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 resistors and diode.

If you want to experiment with other transistors then you could socket them instead of soldering them to the board. You'll need a some 20 SIL sockets, break off the sockets and solder them to the board. Now is the time to solder these sockets on the PCB as well as 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.

VR1 is added to enable you to bias **Q5**. The original pedal does not have that option, but only a fixed resistor **R21** (27k). If you want to be able to bias **Q5**, set the resistance between pin 1 and 2 of **VR1** to 27k before soldering it on the PCB and leave out **R21**. You can later experiment later by changing the setting of **VR1**. If you do not want to bias **Q5** then leave out **VR1** and insert **R21**.

Now finish by soldering the transistors (if not socketed), the bigger MKT/WIMA, Silver Mica and the electrolytic capacitors.

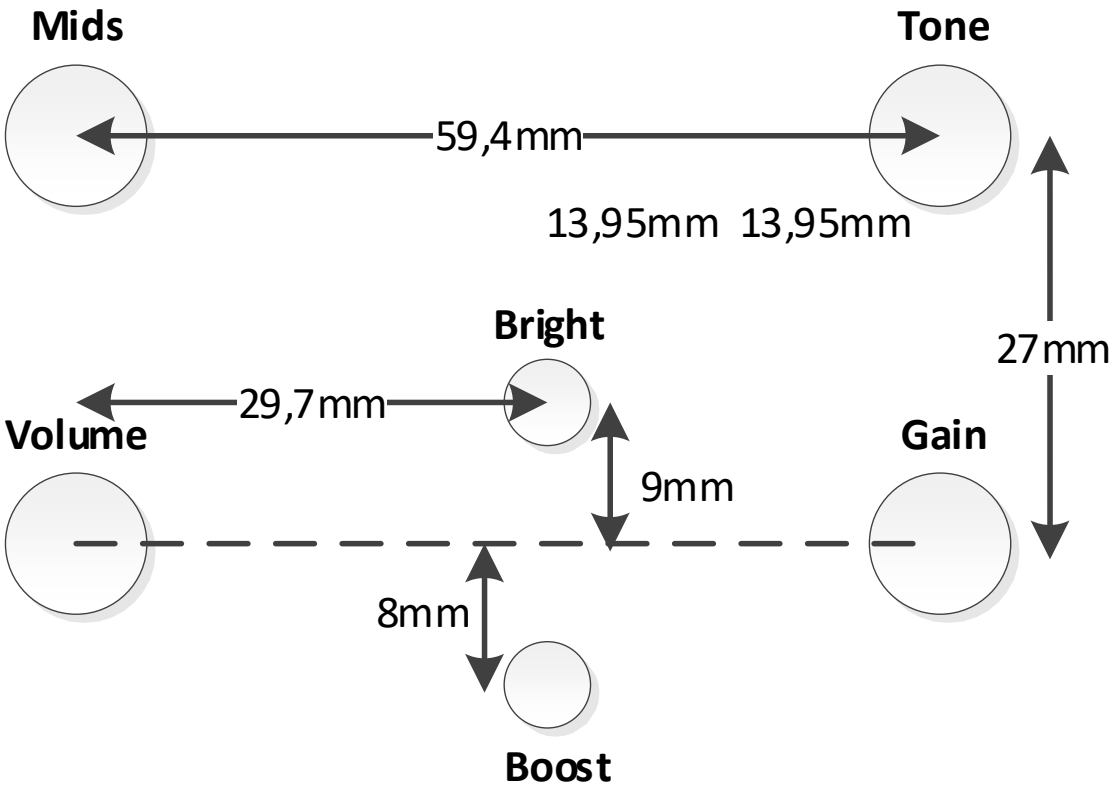
I suggest you now drill the holes in your enclosure so you can use it during the off board wiring. This PCB is very sensitive to noise! Keep the wires as short as possible and/or use shielding on the input and output.

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

- **1 mono and 1 stereo input jack**.
- **2,1mm DC jack** (isolated).
- **22 gage stranded hook-up wire.**
- **1 x LED** as status indicator , in your favorite color (or 2 if you want to use a footswitch for the boost)
- **1 x LED holders** or 2 if you want to use a footswitch for the boost.
- **1 x Footswitch 3PDT** (9 pins) or 2 if you want to use a footswitch for the boost.
- **Hammond 1590BB** case (or similar) in your favorite color. The PCB fits the enclosure in an upright position. You can of course experiment with other enclosures, but measure twice before you start drilling!

Drilltemplate

Pots need 7mm holes, switches need 6mm holes.

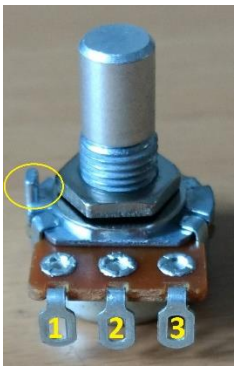


Off board wiring

Potentiometers and switches

In the pictures below you see the correct pin numbering of the pots (Alpha 16mm style).
The **rectangle pad** marks **pin 1**. Before soldering the potentiometers and switches, mount them in the predrilled enclosure and then slide the PCB over them and solder them to the board. Make sure you know which switch you are going to use for **SW1** before soldering (see modifications section).

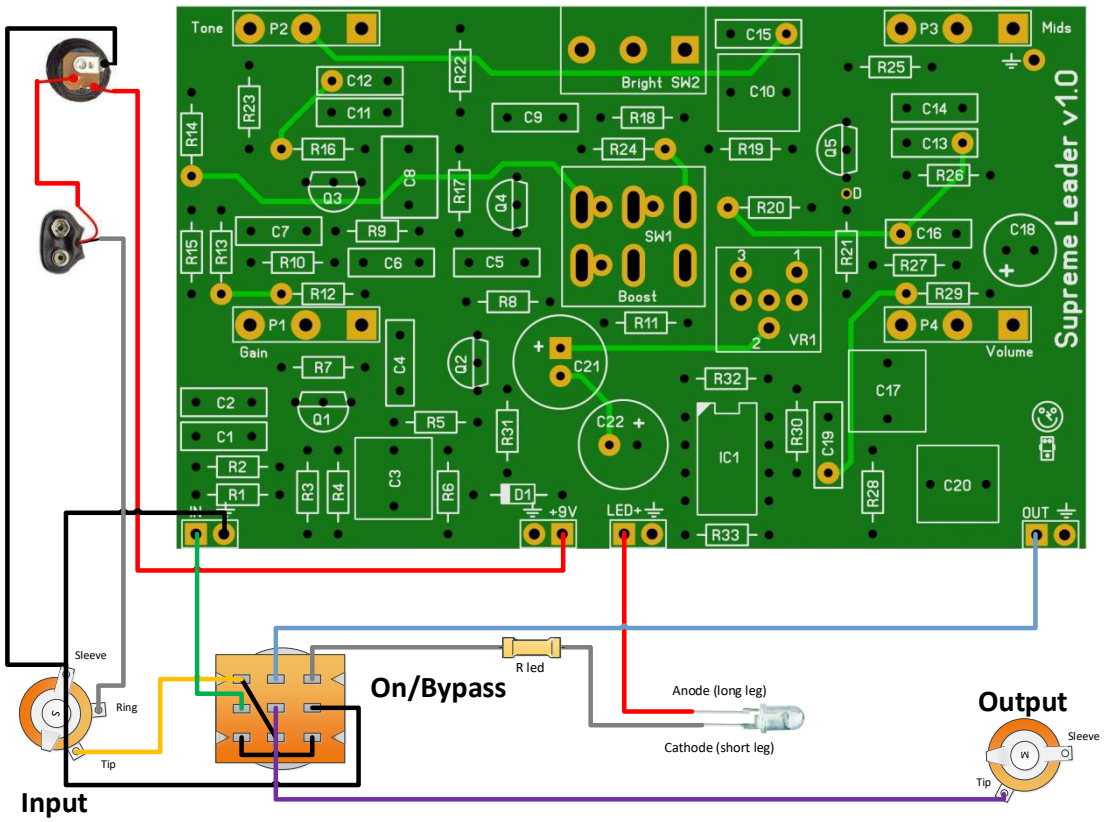
You can break off the pin I marked with the yellow circle with a small pair of pliers.



3 2 1

Note: **SW1** is a DP3T with standard solder lugs:

SW2 is a PCB mounted version SPDT:



R led is 4k7. This schema uses starwiring and requires good contact between the output jack and the enclosure. If that is not possible, connect a cable between the output sleeve and the input sleeve.
Keep cables as short as possible to prevent interference!

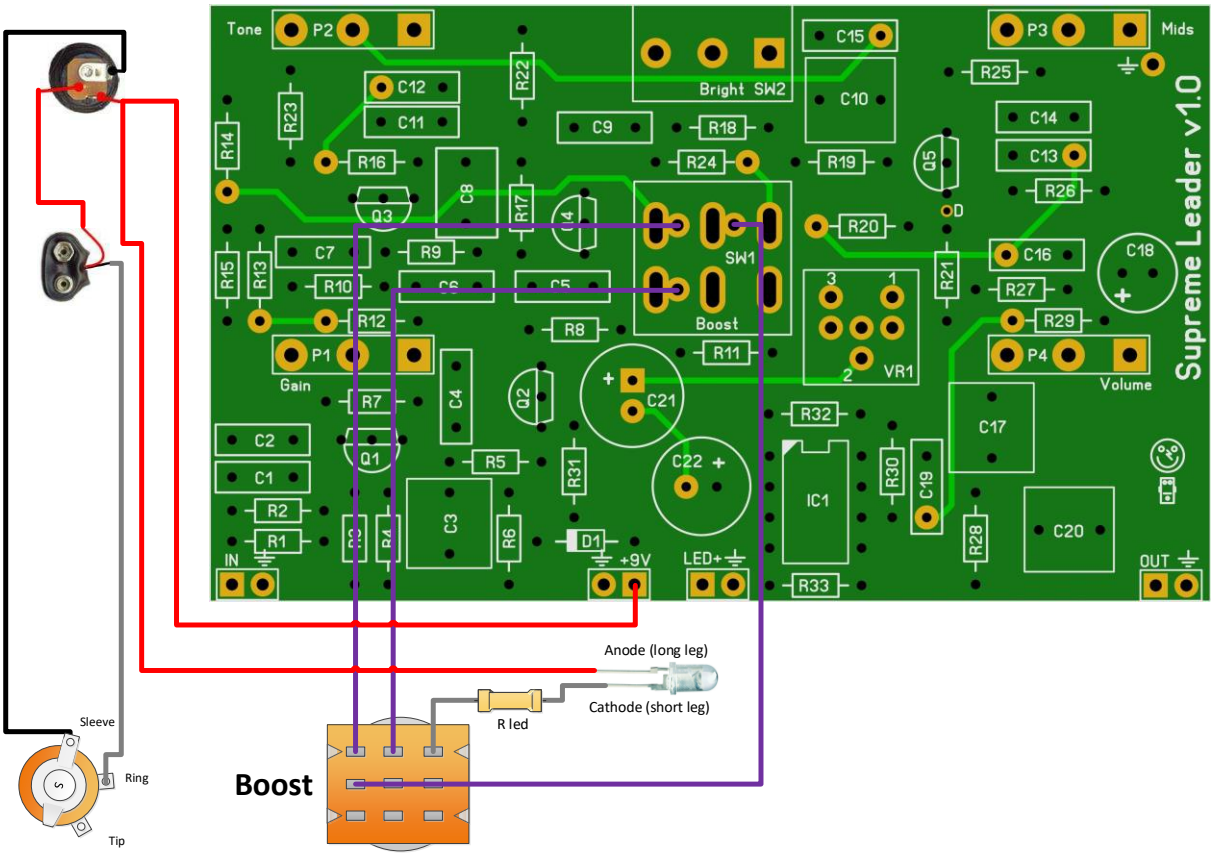
Modifications

Boost (foot)switch

On the original pedal **SW1** only has 2 boost positions (on and off). On this PCB **SW1** is by default a 3PDT (On-Off-On) switch soldered on the PCB and so giving you 3 options: High boost, no boost and a custom boost. As you can see left and middle position are as in the original (on and off). The right position enables you to create a custom boost level determined by **R11** and **R24**. By default these are 6k8 resistors, creating a 2/3 level boost. You can change these values in the range between 6k8 and 15k. 15k will create ½ level boost.

If you only want 2 boost options as in the original, then leave out **R11**, **R24** and use a standard DPDT (On-On) for **SW1**.

SW1 can also be foot switchable. For this leave out **R11** and **R24** and use a standard 3PDT footswitch. To make it a bit easier, there are 3 special pads attached to **SW1** on the PCB:



Remember that all 3 middle lugs on the switch must be soldered together by that single wire! As there is only one LED+ pad, you can solder the Anode of the LED directly to the + side of the DC jack as depicted above.

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