

Dirty Little JCM

Building instructions

v1.0

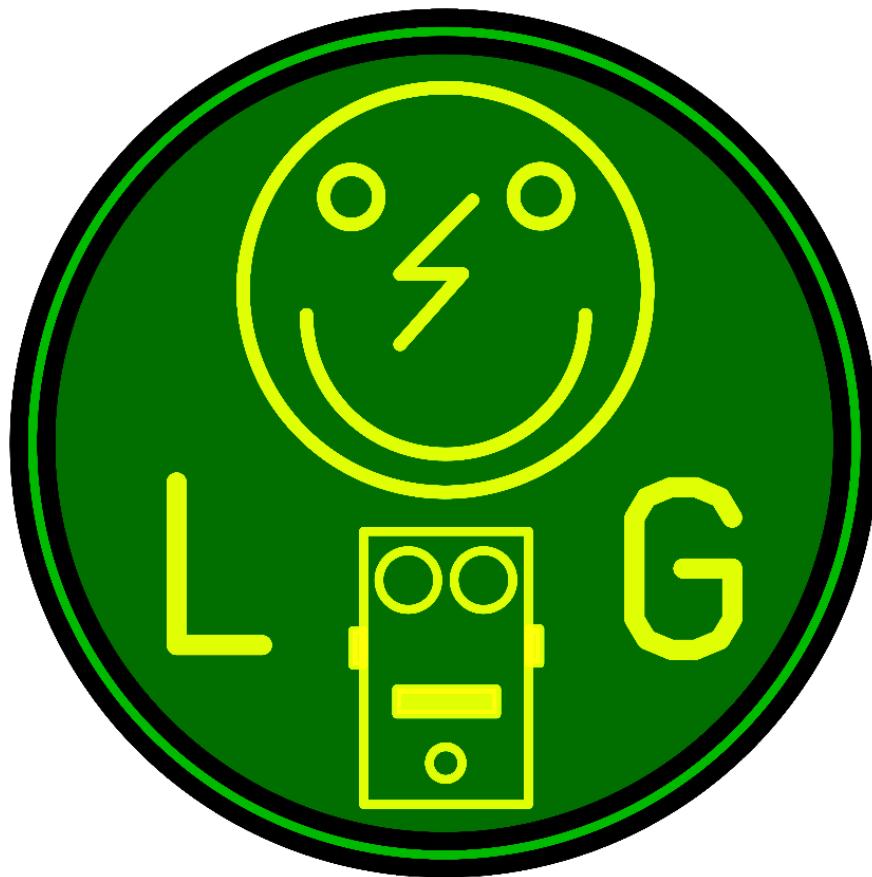


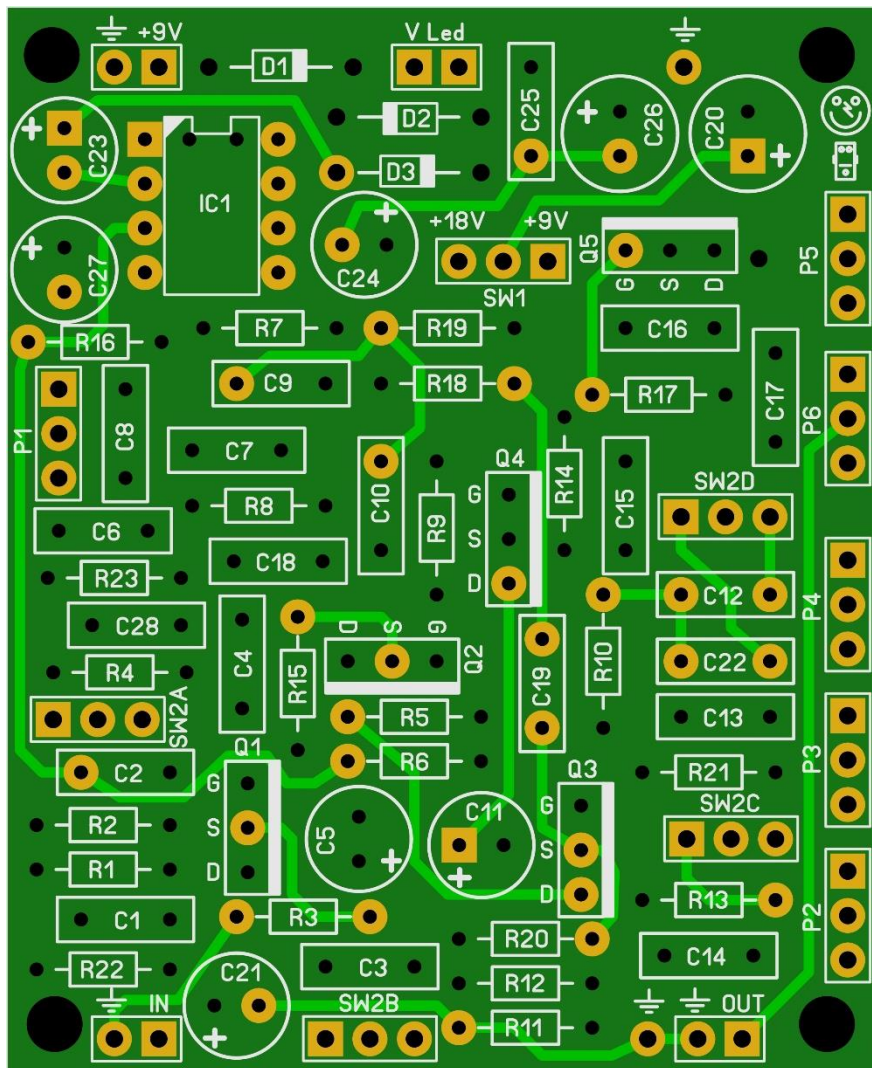
Table of contents

PCB layout	3
Components	4
Bill of Materials.....	5
Power section.....	6
Modifications.....	7
Build sequence	8
Off board wiring	8
Potentiometers.....	8
(foot)switch.....	9
Troubleshooting	10
Schematic	11

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: 24-04-2017

PCB layout



Dimensions: 50 mm x 61 mm
1.97 inch x 2.4 inch



Components

All parts must be rated 25V+

Name	Value	Comment	Name	Value	Comment	Name	Value	Comment
C1	220n		R1	470k		D1	1N5817	
C2	1n		R2	33k		D2	1N5817	
C3	680n		R3	820R		D3	1N5817	
C4	2n2		R4	1M		IC1	ICL7660S	Or LT1054
C5	2u2		R5	1M		P1	A1M	GAIN
C6	22n		R6	1M		P2	B25K	MID
C7	470p		R7	470k		P3	A1M	BASS
C8	47p		R8	470k		P4	B250K	TREBLE
C9	470p		R9	560R		P5	B10k	PRESENCE
C10	680n		R10	100k		P6	B250K	VOLUME
C11	2u2		R11	1M		Q1	MPF4393	
C12	470p		R12	1M		Q2	MPF4393	
C13	22n		R13	56k		Q3	MPF4393	
C14	22n		R14	1k		Q4	MPF4393	
C15	22n		R15	1k		Q5	MPF4393	
C16	220n		R16	47k				
C17	100n		R17	2M2				
C18	220p		R18	2M2				
C19	470p		R19	4k7				
C20	47u		R20	1k				
C21	22u		R21	33k				
C22	270p		R22	1M				
C23	10u		R23*	4M7				
C24	10u		SW1	SPDT	Voltage			
C25	100n		SW2A	SPDT				
C26	100u		SW2B	SPDT				
C27	Not connected		SW2C	DPDT				
C28*	10n		SW2D	DPDT				

* R23 and C28 are part of a mod. Do not place them if you do not want to use the mod!



Bill of Materials

Capacitors			Resistors		
Amount	Part	Type	Amount	Part	Type
1	47p	Ceramic	1	560R	1% metalfilm
1	220p	Ceramic	1	820R	1% metalfilm
1	270p	Ceramic	3	1k	1% metalfilm
4	470p	Ceramic	2	4k7	1% metalfilm
1	1n	MKT	2	33k	1% metalfilm
1	2n2	MKT	1	47k	1% metalfilm
1	10n	MKT	1	56k	1% metalfilm
4	22n	MKT	1	100k	1% metalfilm
2	100n	MKT	3	470k	1% metalfilm
2	220n	MKT	6	1M	1% metalfilm
2	680n	MKT	2	2M2	1% metalfilm
2	2u2	Electrolyte 25V+	1	4M7	1% metalfilm
2	10u	Electrolyte 25V+	1	B10k	16 mm Alpha
1	22u	Electrolyte 25V+	1	B25K	16 mm Alpha
1	47u	Electrolyte 25V+	2	B250K	16 mm Alpha
1	100u	Electrolyte 25V+	2	A1M	16 mm Alpha
Semi conductors			Other		
Amount	Part	Type	Amount	Part	Type
3	1N5817		3	SPDT	
5	MPF4393		1	DPDT	
1	ICL7660S	Or LT1054	1	3PDT	Footswitch
1	LED		1	Jack	Mono jack
			1	Jack	Stereo jack
			1	Box	Hammond 1590B/BB
			1	DC	2,1mm DC jack
			4	Spacer	Plastic PCB spacers
			1	DIL-8	Socket for IC

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.



Power section

The effect has been designed to be able to operate on 9V or 18V. SW1 enables you to switch dynamically between these settings. If you do not want to use the switch then do not forget to solder a jumper between the middle pad and the desired voltage pad. **NEVER** let the outer 2 lugs of SW1 touch each other and also never connect all 3 pins of SW1 together. If your IC gets hot then there is a short in the power section and you must disconnect the effect immediately from the adapter and find the fault.

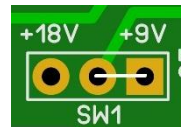
You can choose between 3 options for the power section:

1. **No charge pump:**

Do **NOT** install: C23, C24, D2, D3, IC1, J1, SW1

Extra jumpers: SW1 on 9V pad and middle pad

The effect can now be used with a standard 9V adapter or a 18V adapter

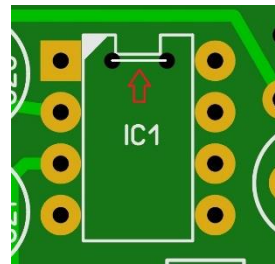


2. **Charge pump using a ICL7660S:**

Place all components as stated on the PCB and connect a jumper between leg 1 and 8 of IC1.

Do not forget to place the jumper before placing the DIP-8 socket of the IC.

NB. Only use the 7660S and not other versions without the S.



The effect can now only be used with a 9V adapter and you can switch it internally to 9V or 18V.

3. **Charge pump using a LT1054:**

You can place all components as stated on the PCB just do **NOT** connect pin 1 and 8 of IC1!!

The effect can now only be used with a 9V adapter and you can switch it internally to 9V or 18V.



Modifications

The effect can be perceived as dark and wooly. If you experience this, I recommend the following mod (thanks to John Kallas & Ivlar on tagboardeffects):

- C4: 220n
- C17: 22n
- C28: 10n
- Q1-Q5: 2N5457 (or J201)
- R9: 1k
- R15: 220R
- R20: 220R
- R23: 4M7

These parts are extra and not in the bill of materials (except for R23 and C28 which are on the BOM).

Switching between the “Super Lead” and “Super Bass” can be done by either soldering SW2A-D on a single 4PDT or as mentioned in the off board wiring by 2 SPDT (SW2A and SW2B) and one DPDT (SW2C and SW2D).

The MPF4393 are hard to find and expensive. You could replace them with PN4393, 2N5457 or J201. I advise the 2N5457 for the pure subjective reason that I like the sound.

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. Usually capacitors differ a lot in size depending on their rating.

Note: Do not blow on your solder in an attempt to cool it down. That will possibly result in a bad join that might corrode!

Start by soldering the resistors, then the diodes and then the ceramic capacitors.

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 small MKT (<680n) capacitors then the rest of the MKT capacitors and then the electrolytes.

I suggest you now drill the holes in your enclosure so you can use it during the off board wiring.

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.

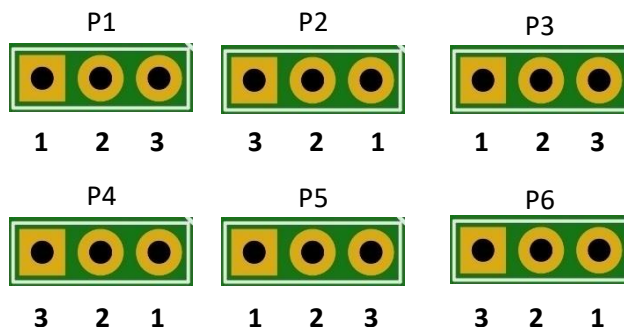
Off board wiring

Potentiometers

In the pictures below you see the correct pin numbering of the pots.

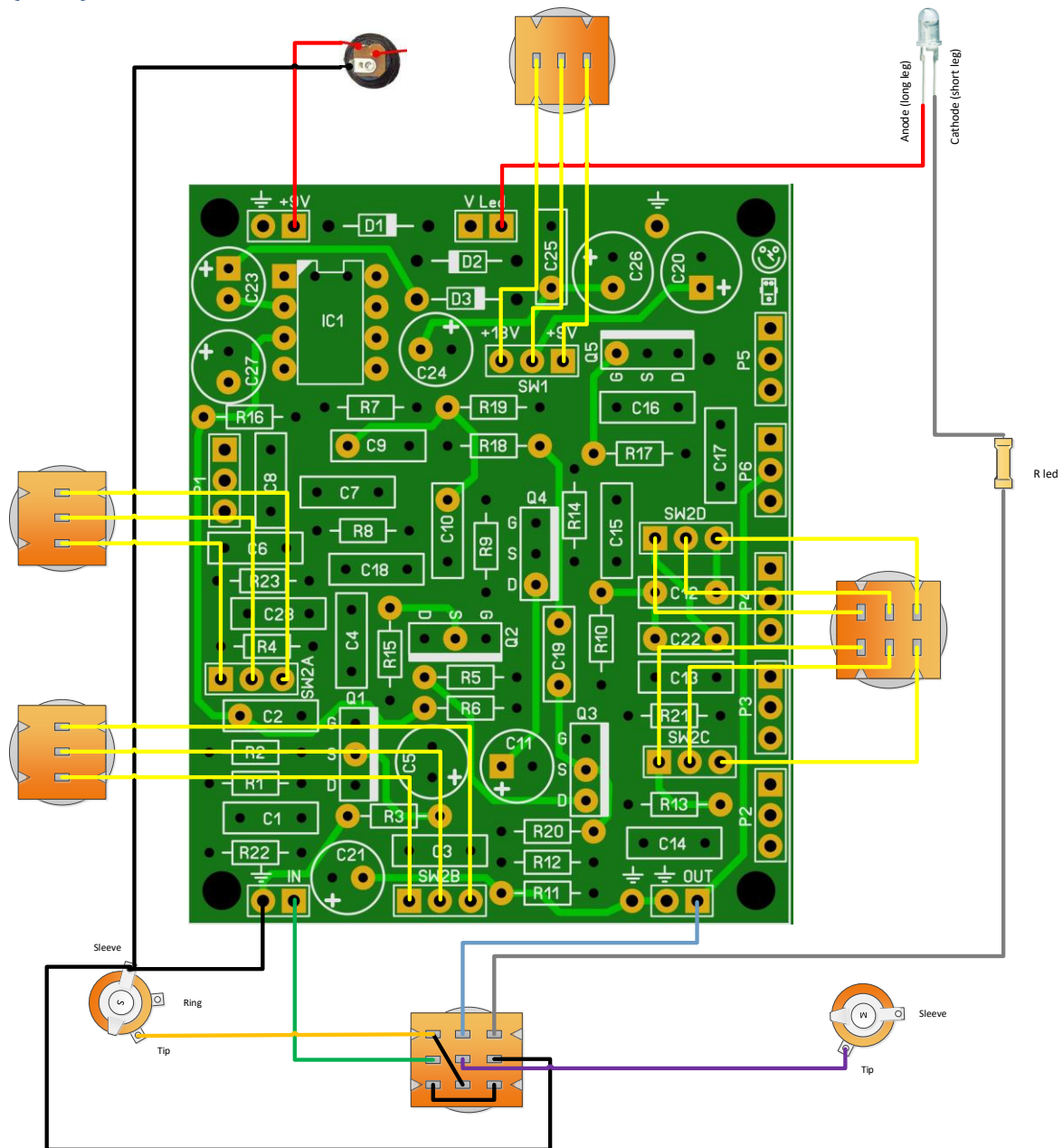
Solder the wires accordingly and it is always a good idea to twist the wires together to create a sort of extra shielding against external noise.

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





(Foot)switch



Note that this wiring is using a sort of star grounding and the output connector must make good electrical contact with the enclosure else the grounding will not work properly and the effect might start to buzz loudly.

Also Note that **R led** is a **4k7** resistor. You can change this value depending on the type of LED you use but 4k7 is safe enough for almost all LEDs for both 9V as well as 18V.

As mentioned in the beginning of this manual, you could combine the 2 SPDT with the DPDT switch of SW2A-SW2D to one 4PDT switch. Make sure you connect all the 4 rows in the same way, making the square pad always the left most pin on the switch.

It is now time to place your transistors and IC in the sockets if needed. Connect everything, build it in your enclosure and enjoy your effect!

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.



Troubleshooting

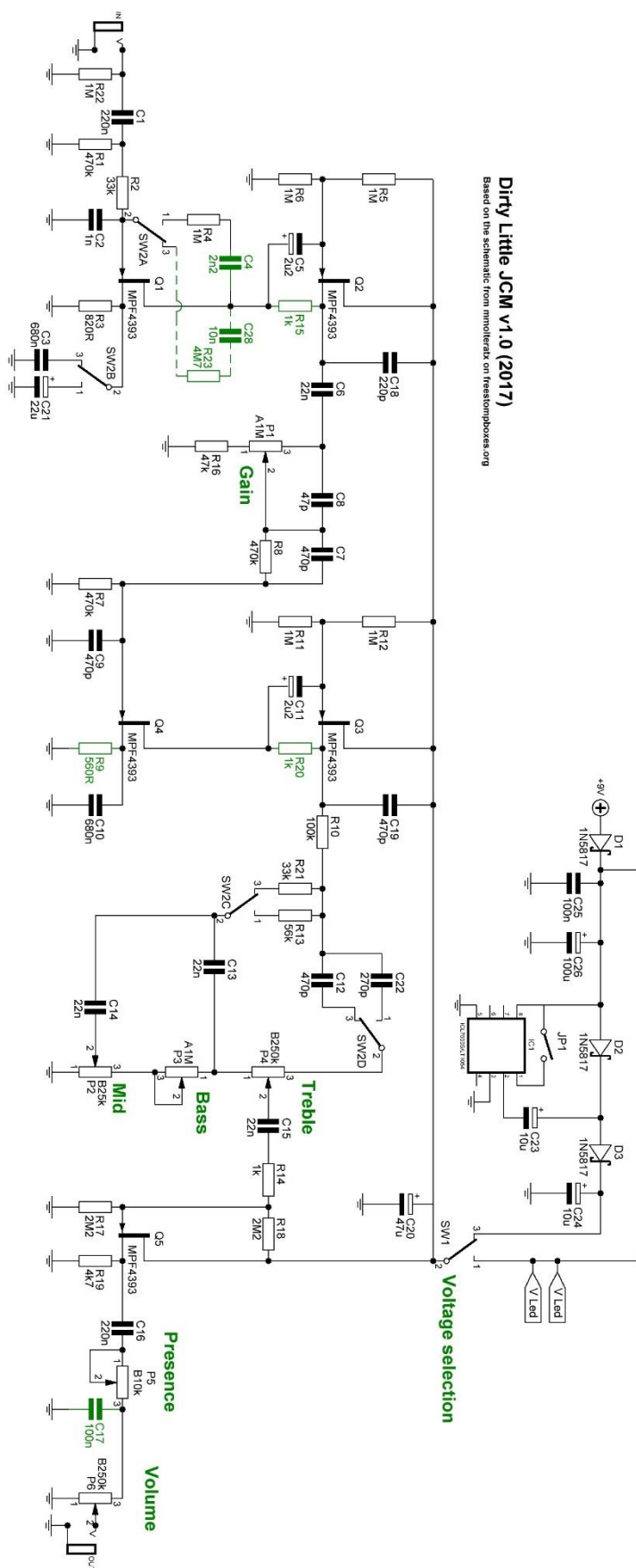
All PCB's have been 100% factory e-tested and out of every batch I receive I build an effect to double check, so there should not be a connection problem on the PCB itself.

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



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.