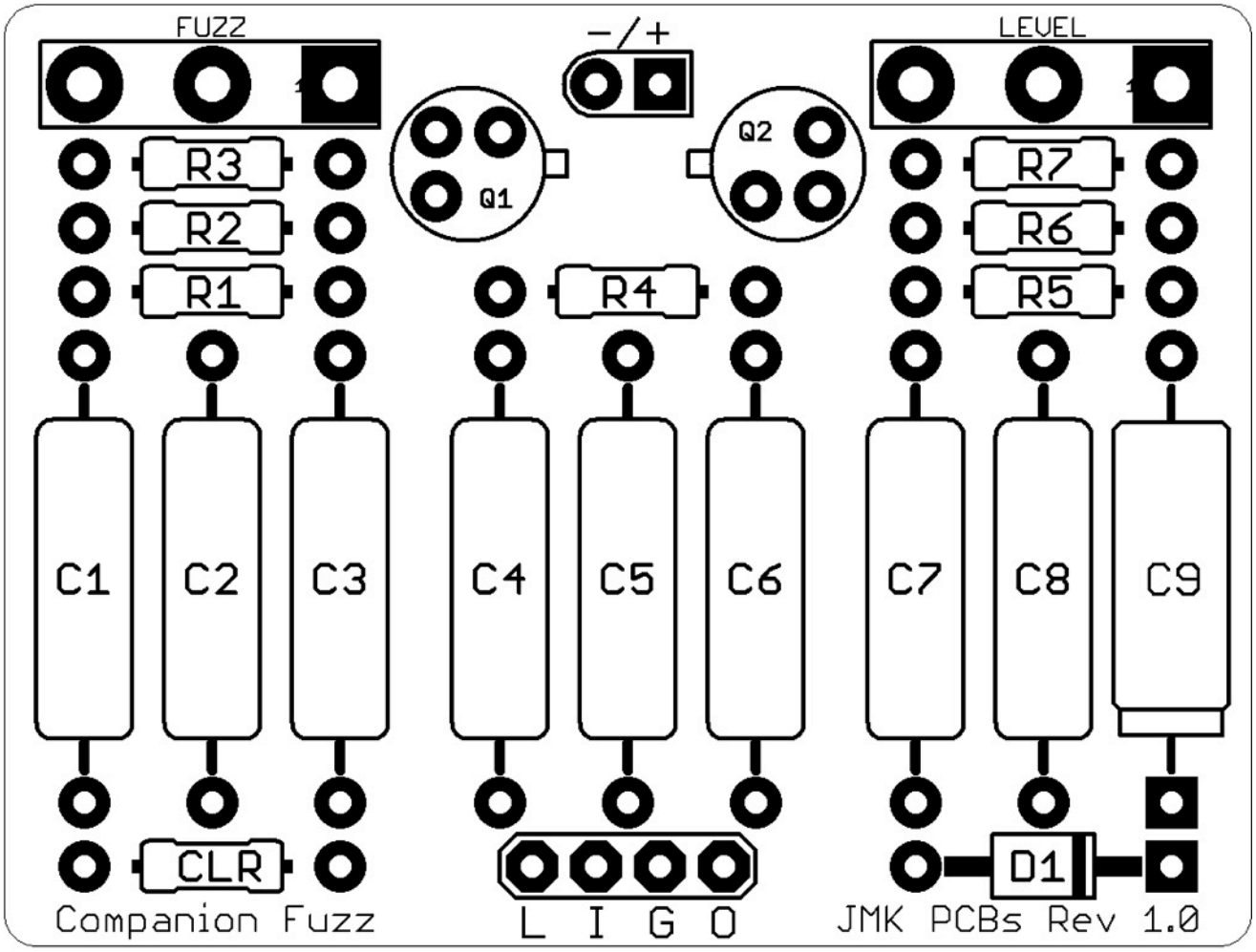
JMK PCBS PRESENTS...

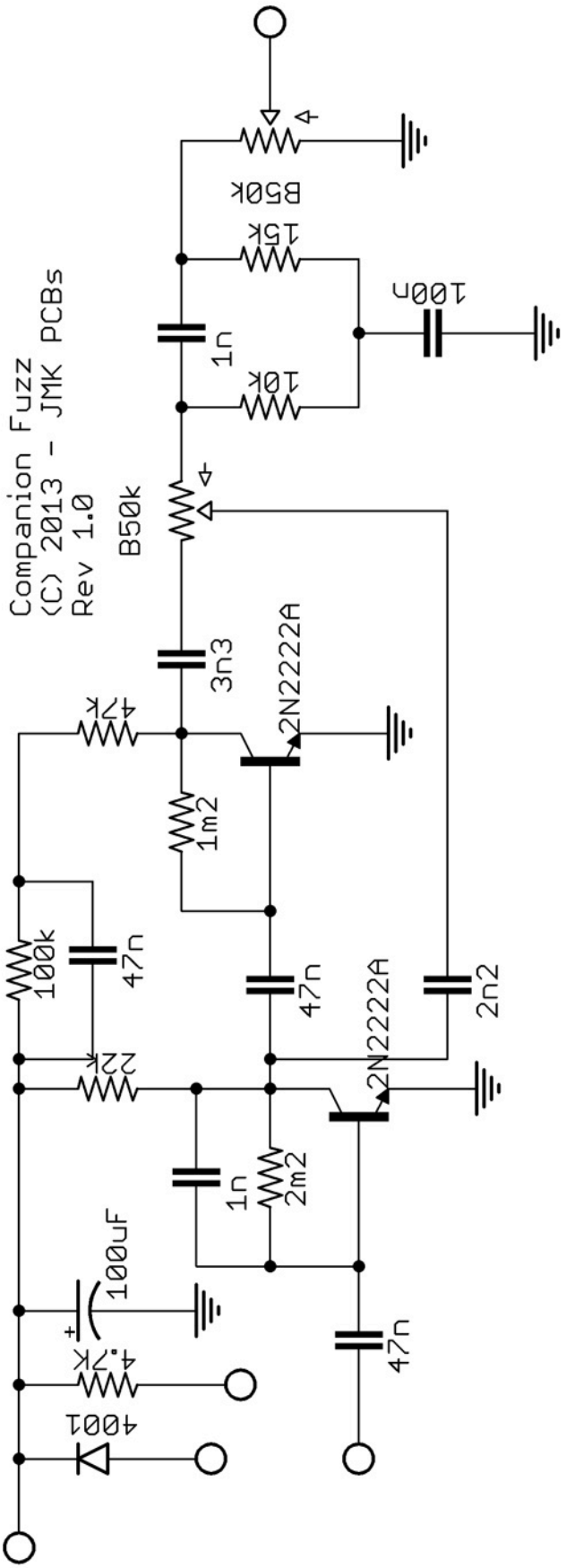
COMPANION FUZZ

PCB AND SCHEMATIC ARTWORK (C) 2013 JMK PEDALS

VERSION 1.0.1: 6/27/2013



Resistors				Capacitors				Transistors	
R1	22K	R5	47K	C1	47n (film)	C6	47n (film)	Q1, Q2	2N2222A
R2	2.2M	R6	100K	C2	1n (film)	C7	1n (film)	**Potentiometers**	
R3	10K	R7	15K	C3	2.2n (film)	C8	100n (film)	FUZZ	B50K
R4	1.2M	CLR	4.7K	C4	47n (film)	C9	100u (elec)	LEVEL	B50K
				C5	3.3n (film)			**Other**	
								D1	1N4001



Shopping Details

• Companion Fuzz is made to accommodate some very specific parts. The recommended caps are Axial Film and Electrolytic types, and the recommended resistors are 1/2 watt Carbon Comp Resistors. Why? Mojo of course. The parts are easily found at Mouser, and run less than $12 for everything on the PCB.

• *Please note!* Rev 1.0 PCBs will NOT accommodate 1/2 watt Carbon Comps, but will work for typical 1/4 watt film/carbon comp resistors. Check your PCB version! All PCBs sold after July 1, 2013 will be Rev 1.1 or higher PCBs!

• The transistor that is recommended is slated to be discontinued and obsoleted. Even though that's the case, at the time of this document's creation, Mouser had more than 80,000 of the 2N2222A transistor, and substitution options are available which are not slated for discontinuation.

• Here is the project (Rev 1.1 and later) for Mouser with all the parts needed.

• Here is a list of Mouser part numbers (Rev 1.1 and later) if you just want to get some of the parts, or look for similar substitutions elsewhere:

BOM Value	Mouser Part #
R1	66-RC20GF223J
R2	791-RC1/2-225JB
R3	66-RC20GF103J
R4	791-RC1/2-125JB
R5	66-RC20GF473J
R6	66-RC20GF104J
R7	66-RC20GF153J
CLR	66-RC20GF472J
C1, C4, C6	539-150473J250BB
C2, C7	539-150102J630BB
C3	539-150222J630BB
C5	539-150332J630BB
C8	539-150104J100BB
C9	647-TVX1E101MAD1LS
Q1, Q2	511-2N2222A
D1	625-1N4001E-E3/73

Build Notes

• The Companion Fuzz is a pretty awesome, wacky, almost out-of-control fuzz from the way back days. This is not your Dad's old Fuzz Face.

• There are two major things most noted to remember about this fuzz:

a. This fuzz may not 'boost' your signal. In fact, vintage versions were reported to actually have a slight volume drop. With the JMK Version, we could not detect a 'significant' volume drop. If there was a drop, it was hard to tell, but we had to have the volume all the way up to avoid a drop. That is not to say that you might not find a bit of a drop in yours. If you do experience a drop, one method to combat this is to simply add a boost to the output - something like a simple Mosfet or jFet boost will do - which will simply allow you to increase the signal level.

b. The 'Fuzz' control is almost useless. In fact, it's an reported phenomenon that some people didn't even know that the Fuzz control was actually hooked up to the circuit. With the JMK Version, we found this to be a very subtle control (read, we wondered why it's in the design). But, for cloning and experimentations sake, we left it in, because an insane Fuzz pedal should have a Fuzz control that you can dime which you can then point to and explain why it sounds so out of control.

• The 2N2222A are not the original transistors, however, they are pretty great sounding in this circuit. They were chosen as the recommended transistors because: a) they're metal can TO-18 packages, which is pretty mojorific, and b) they sounded better than the other BJT types we had in our lab. So, while you can obviously switch these out for another transistor type, we recommend getting a pair. **We highly recommend socketing important parts like Transistors.** Socketing allows you a quick swap if there's an issue, or if you want to try another part out.

• This PCB is the second design from JMK that's actually meant to go into a 1590B! We normally design for the 125B, but with this design we planned for those who like to work their PCBs into smaller enclosures. In a 1590B, with jacks on the side and battery below the switch, there's lots of room for this PCB. If you want to go with a 125B with top mounted jacks you may have to carefully plan you build so that the top mounted jacks don't interfere with the PCB when it's mounted to the pots.

• Hooking up the PCB is pretty simple, but to clarify: L = the connection for the + end of an LED (CLR is R24); I = PCB Input; G = Ground for the Switch; O = PCB Output; + = 9V input; - = Ground for DC Jack; G = Extra Ground for 1/4" Jack (not on V1.0)

True Bypass Wiring Diagram