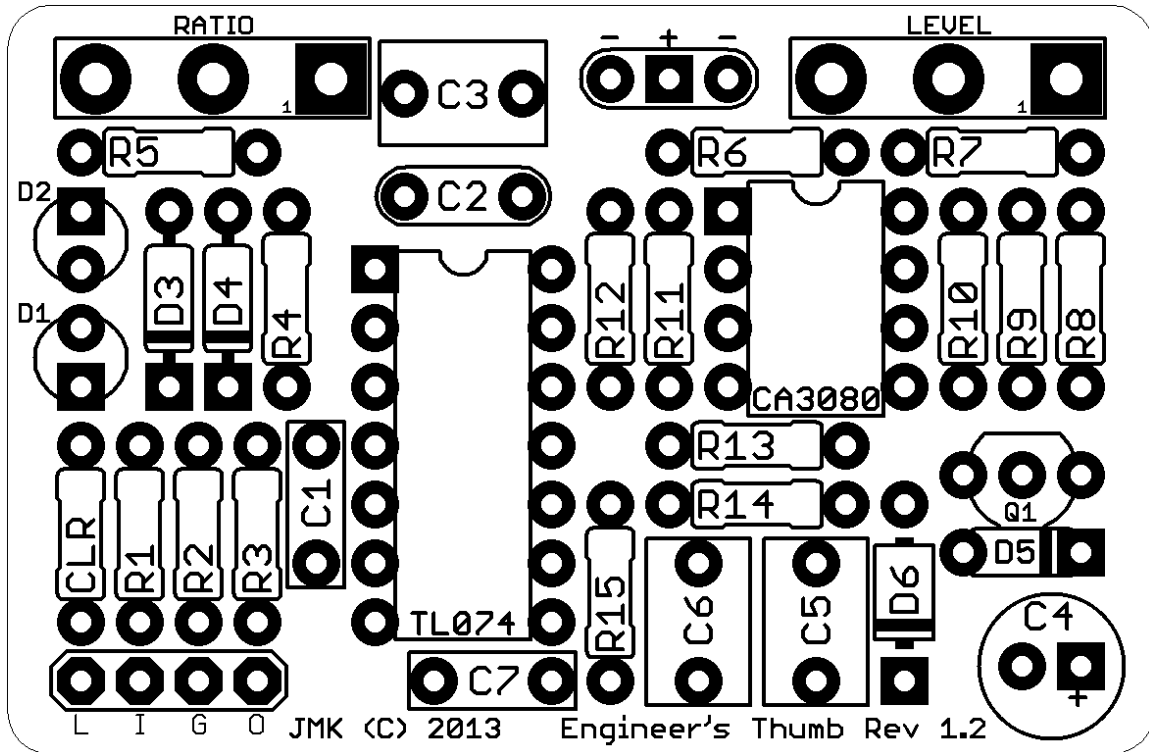


JMK PCBs PRESENTS...

ENGINEER'S THUMB

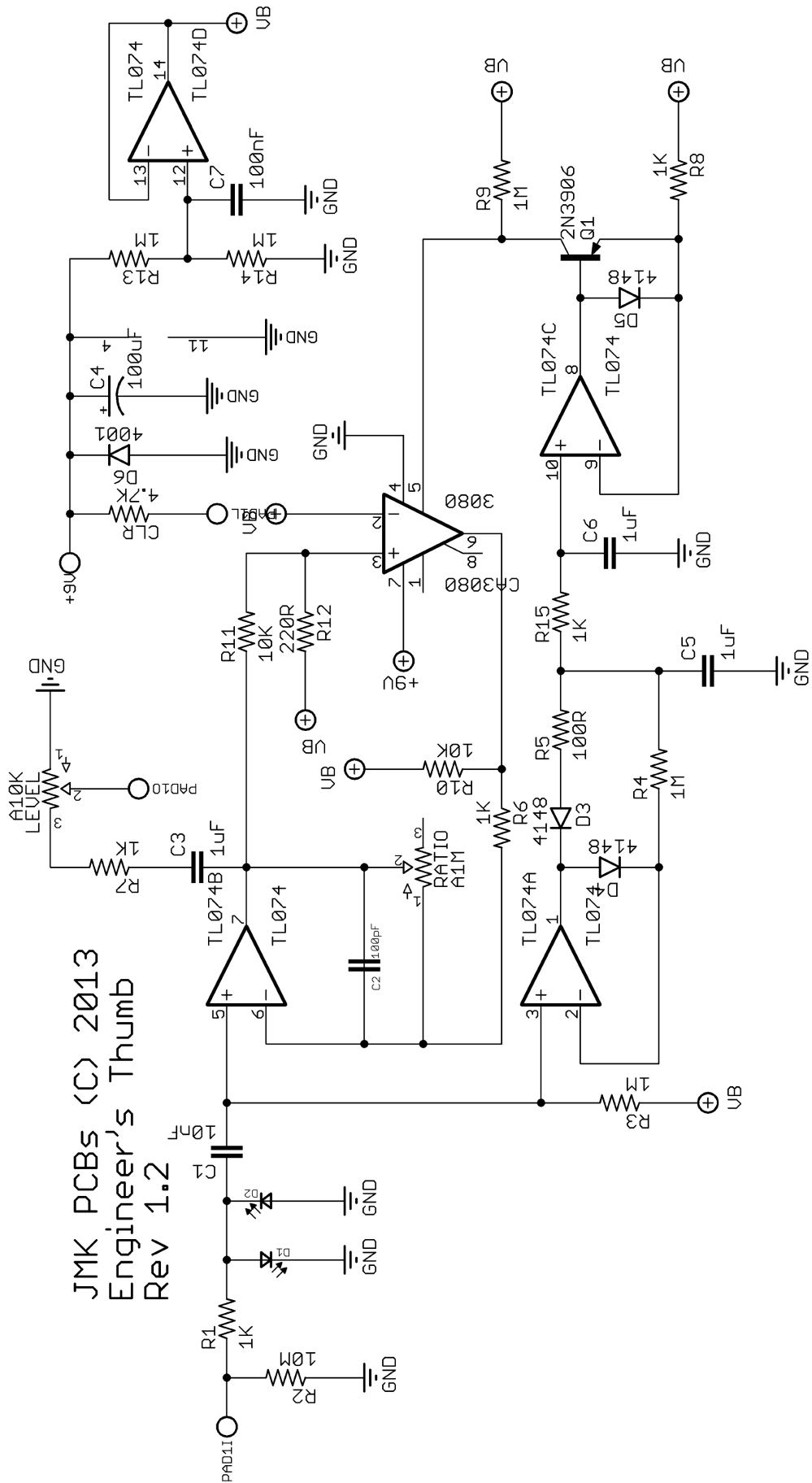
PCB AND SCHEMATIC ARTWORK (C) 2013 JMK PEDALS
VERSION 1.2.1: 01/7/2013



Resistors		Capacitors		Transistor					
R1	1K	R9	1M	C1	10nF	C5	1uF	Q1	2N3906
R2	10M	R10	10K	C2	100pF	C6	1uF	ICs	
R3	1M	R11	10K	C3	1uF	C7	100nF	IC1	TL074
R4	1M	R12	220R	C4	100uF*			IC2	CA3080
R5	100R	R13	1M	Potentiometers		Diodes			
R6	1K	R14	1M			D1, D2	Red LED		
R7	1K	R15	1K	LEVEL	A10K		D3-D5	1N4148	
R8	1K	CLR	4.7K	RATIO	A1M		D6	1N4001	

This Document is designed for personal use only! Do not use this to create a product for sale without permission of it's owner: jmkpcbs@gmail.com

JMK PCBs (C) 2013
 Engineer's Thumb
 Rev 1.2



This Document is designed for personal use only! Do not use this to create a product for sale without permission of it's owner: jmkpcbs@gmail.com

BUILD NOTES

- The Engineer's Thumb is a compressor circuit that improves on the classic OTA based Dyna/Ross tradition. Designed by MerlinB, this circuit originally featured a dual OTA with buffers built in, the LM13700 chip. This PCB version chooses to use the much smaller form factor CA3080 chip due to it's smaller size. Admittedly, this is a more expensive chip to use, and it is becoming harder to find, however, the smaller form factor of the PCB may compensate for the higher price of the chip due to the build being suited for a 1590B or 125B enclosure, making for a small, fairly inexpensive build otherwise.
- Not all CA3080's are created equal. We recommend that unless you have experience with getting CA3080's from a reliable source, look for your chip at Smallbear Electronics, which has a reliable chip readily available. Yes, less expensive chips may be found, but be aware that some out there are unreliable.
- Yes, D2 and D3 are supposed to be LEDs. On the PCB they look like silicon diodes, because we were hoping that silicon diodes would work there, but they don't. Go with LEDs, we used red ones in our prototypes.
- The PCB is 1.95" wide, and 1.275" tall. This means it will fit in a 1590B enclosure or larger - we recommend a 125B to give you extra room. The potentiometers are located precisely 1.3" centre to centre, and the PCB is designed for PCB mount pots, though lug pots will work fine as well.
- Hooking up the PCB is pretty simple, but to clarify: L = the connection for the + end of an LED; I = PCB Input; G = Ground for the Switch; O = PCB Output; + = 9V input; - = Ground for DC Jack; GND = Extra Ground for 1/4" Jack
- It should be noted that there are several modifications available for this circuit. Below are some of the common mods, the parts needed and a short description of how to apply them.
 - Replace R3 with a B1M potentiometer for a threshold control. Connect lug one to one side, and both lug 2 and 3 to the other side of the resistor.
 - Increase R5 for longer attack. Currently that resistor is slated to be 100R, but you can try 1K, 10K, or 100K as desired. Socket it to try and find your perfect value.
 - Decrease R4 for a shorter release. Currently that resistor is slated to be 1M, but you could try 100K, 10K, or even 1K. Socket it to try and find your perfect value.
 - Add a series 10K and 4.7nF capacitor in parallel with R10 for a treble boost mod. You can add it on a switch , or simply add the parts directly to the PCB.
- **We highly recommend socketing your transistors and ICs!** Socketing allows you to switch your parts easily if you have installed them backwards, and also allows you to swap out and try other parts to see which you like the best.

TRUE BYPASS WIRING DIAGRAM

