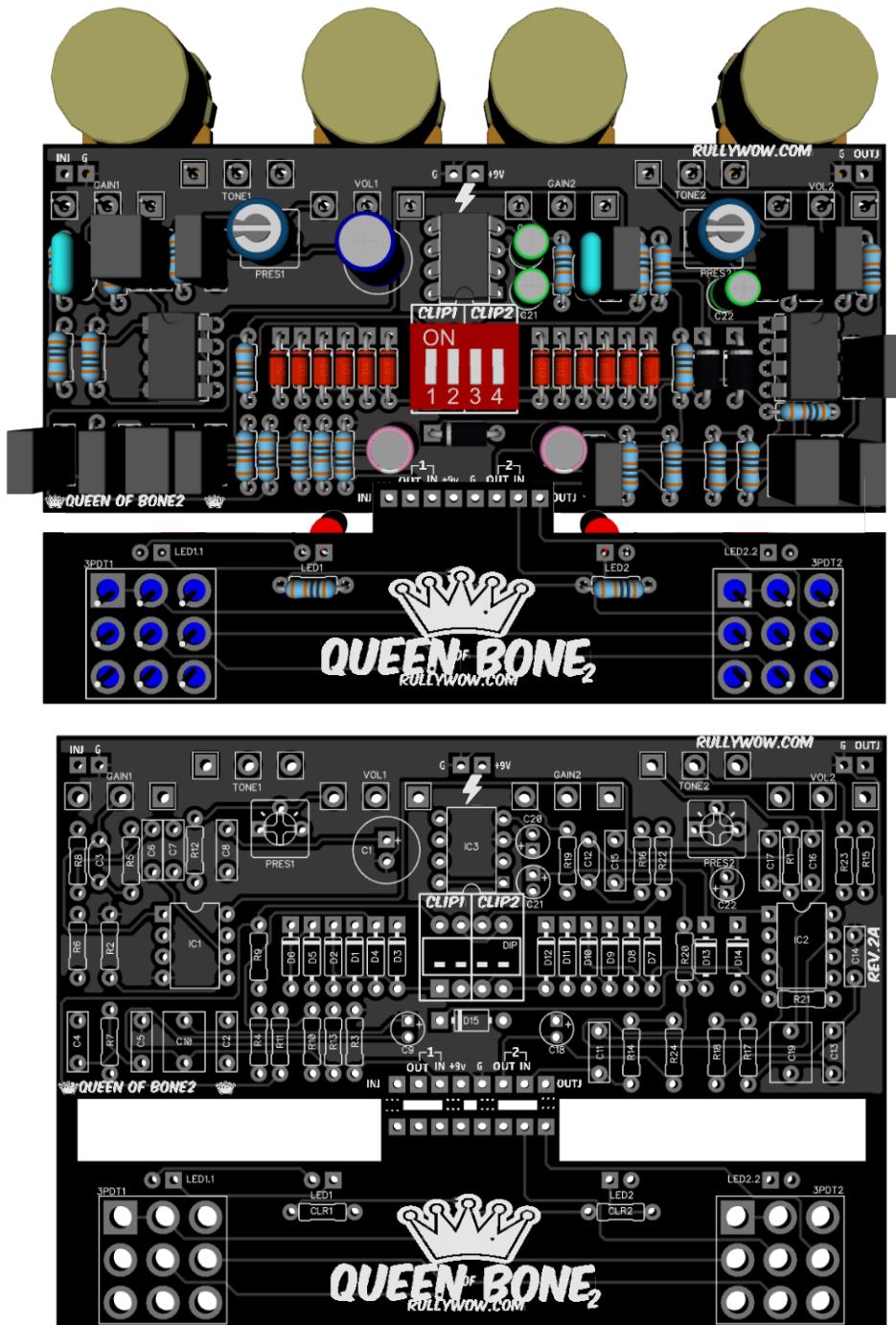


# QUEEN OF BONE<sup>2</sup>

Dual Marshall "BluesBreaker" Overdrive PCB Build Document

Revised: Feb 2018



## **WELCOME!**

---

The Queen of Bone dual overdrive PCB is back and better than ever. This classic boutique dual overdrive is one of the most popular pedals on the scene and for good reason. The QoB<sup>2</sup> is inspired by the popular King of Tone™ pedal, yet adds some special tweaks including a charge pump to boost standard pedal 9VDC to around 17-18VDC for increased headroom and clarity. This is a great DIY pedal project and rewarding to build.

The circuit utilizes two slightly modified Marshall Bluesbreaker™ circuits and cascades them together. You can engage each pedal, one at a time, or engage both for sonic high-gain glory. Because you can run both sides at the same time you can get some great combinations. Some like to set one side for a slight crunch and the other as a boost. The onboard DIP switch selects different diode combinations. The combinations are wonderful; from a mild boost to a wild overdrive, the QoB<sup>2</sup> is sure to become a staple in your overdrive arsenal.

## **CHANGES FROM THE ORIGINAL QUEEN OF BONE**

---

With your suggestions and feedback, we have made the QoB<sup>2</sup> even easier to build and use. Location of components were changed in order to be easier to populate. The trace layout ensures quieter operation. The newly improved layout now accommodates all jacks (1/4" and DC) being located on the top, which saves precious pedalboard space. The newly included breakout board is designed to attach with standard 2.54mm (.100") header pins and includes two LED mounting positions right on the board. The jumper option on the PCB to run at 9V has been removed as most everyone enjoys uses the 18V charge pump option. Let's get started!

## **CONTROLS**

---

VOL1 & VOL2 = Volume

GAIN1 & GAIN2 = Gain

TONE1 & TONE2 = Tone

PRES1 & PRES2 = Presence (high treble frequency) internal trimmer

CLIP1 and CLIP2 = DIP switches which change the clipping of the respective side. More switches engaged (moved up) will result in more clipping & lower output volume. Experiment for best results! Each side has two DIP switches assigned to it. Apparently DIP1 and DIP3 engage the "OD" side of each respective side. DIP2 and DIP4 engage the "Distortion" clipping. Experiment for best results.

## BUILD TIPS AND TRICKS

---

- **IMPORTANT!** Take your time to follow and, read **ALL** the instructions. These instructions are detailed in order to ensure a smooth building experience whether you are a novice or pro pedal builder. The new included 3PDT breakout board is very handy but only if you drill accurately and solder the Pots, header pins, and 3PDT stoms in a specific order to avoid stress on the components.
- There is a **video** available on YouTube which shows the actual assembly of the QoB2. Check it out at <https://youtu.be/SYShhTDP0lw>
- The **MS Excel** version of the BOM is available on the QoB<sup>2</sup> webpage. This is useful to sort the BOM by value or make notes for yourself for easier component shopping etc.
- **Enclosure Size...** This pedal is designed to fit into a 1590BB style enclosure in landscape format
- All pots are designed for 16mm Alpha Right Angle PCB mount. Tayda Electronics, Smalbear and Mammoth are all great places to get these.
- When shopping for header pins, try to get the LONG variety (20mm long). Pins are standard 2.54mm (0.100") spacing
- Revised: The original diodes are MA856 and 1s1588. The 1s1588 are somewhat hard to find and the MA856 are near unobtainium. You can use any standard clipping diodes. 1n4148 are a great choice as are many other silicon diodes. Try to keep each "style" of diodes the same type. Other diodes you can try are 1n914, RED LED, 1n4001, BA282 germanium etc. Even better you can choose to use some SIP sockets to swap these out for experimentation if you are unsure.

## ASSEMBLY

---

1. Carefully snap off the 3PDT PCB from the main PCB. Placing the snap points (perforated holes) over a sharp edge of a tabletop works fairly well. After separating the breakout PCB, remove the tabs shaded in RED shown in the picture below using pliers or similar. Lastly, use sandpaper or a file to smooth out the edges.



Do not breathe in the dust from this sanding process as it is really bad for your health. Use an approved respirator.



2. Populate the QoB2 PCB in this order for easiest building:
  - a. ALL 1/4w resistors (including the CLR1 and CLR2 on 3PDT board for LED brightness)
  - b. diodes
  - c. sockets for ICs
  - d. 50k trim pots (PRES1/PRES2)
  - e. 4 position DIP switch
  - f. film caps
  - g. electro caps

**⚠ STOP! DO NOT SOLDER 16mm POTS, LEDs, header pins, or 3PDT stomp switches in yet!**
3. Print out the included drilling template included in this document. Accuracy is important here, so take your time and ensure the printed drilling template is printed at 100% scale and you drill as accurately as possible in order to make assembly as easy as possible. Measure to ensure that your printer output the reference square 1 inch to verify this.

Fold and tape the template to your enclosure evenly. Carefully mark your holes using the template as a guide. It's recommended to use a spring loaded center punch to do this. The punch helps you drill accurately by making little marks which helps your drill bit not



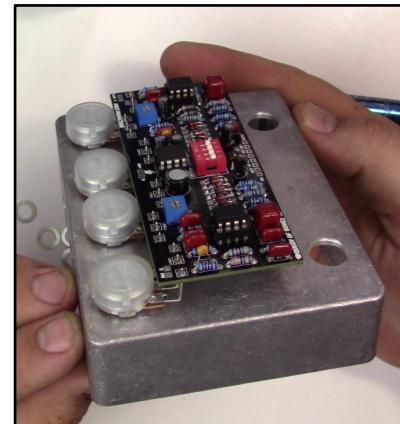
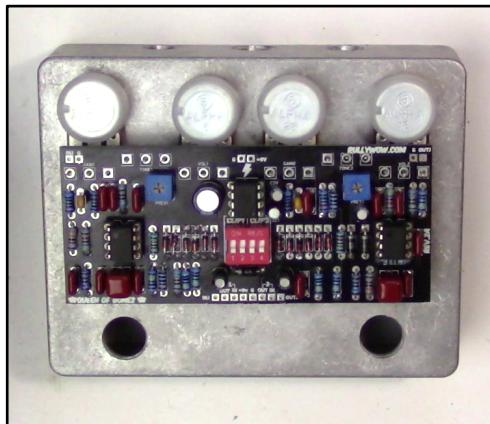
SPRING CENTER PUNCH

skate on the surface. If you don't have one it's well worth picking one up.

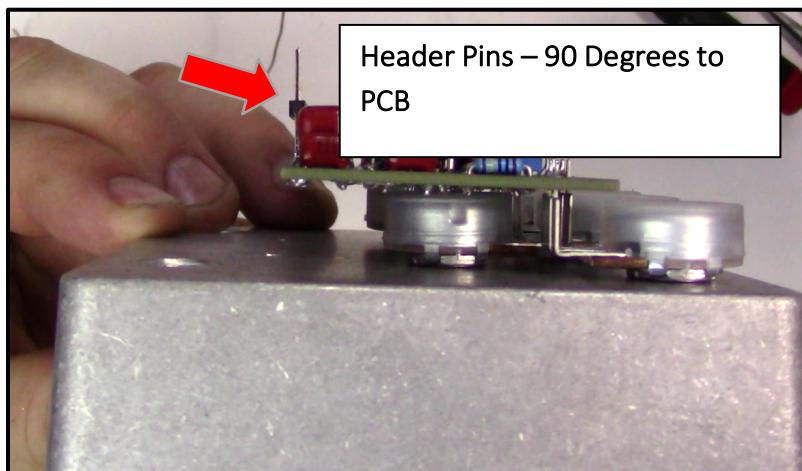
**⚠️** There are two (2) LED mounting locations by each 3PDT switch. Be sure to drill only ONE LED location pair. It's your choice if you want the LEDs above the footswitches or more towards the center...the PCB has both options.

Using a small drill (I prefer a 1/8" (3mm) drill bit), and a drill press if available, drill all your pilot holes. Then using a step drill, enlarge each hole to fit the components.

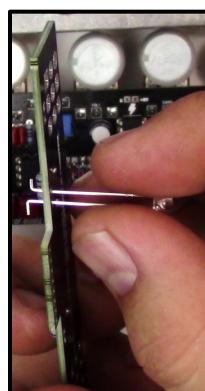
4. Insert the 6 16mm pots into the main PCB but don't solder them in just yet. Ensure you use pot covers or some insulating material to avoid the TONE pots shorting out on the back of the PCB. Also, make sure you snap off the tab on the pots if you aren't drilling a hole for them. Use the FACE (outside) of the drilled enclosure to line up the 6 pots and secure them with the nuts from the inside. Once you are happy the pots are lined up, and the board is square to the enclosure, solder one lug of a pot and try to make the main PCB as parallel to the enclosure as possible. Solder the rest of the lugs of the pots to the main PCB. Remove nuts and main PCB assembly.



5. Header pins. It is highly recommended to use header pins to attach the footswitch board to the main PCB. These are standard 20mm long pins with 2.54mm (0.100") spacing. You can use wire but header pins make for much easier installation provided you drilled accurately. Snap off a strip of 8 pins. As the 3PDT board will sit higher above the main PCB, we will solder the pins onto the main PCB sticking up as far as possible. Again, try to solder these as vertical (perpendicular) as you can. It is wise to tack one pin in first to check for straightness, then solder the remaining 7 pins.

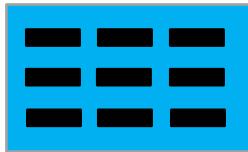


6. 3PDT board. Prep the 3PDT board by inserting two standard 3PDT switches in the holes (but do not solder them yet). The holes in the 3PDT lugs should be oriented vertically. Insert the two status LEDs of your choosing in the holes you drilled in step 3.  
**Pro Tip:** You can bend the end wires of the LED at 90 degree angles to ensure they won't fall out while you are inserting them.



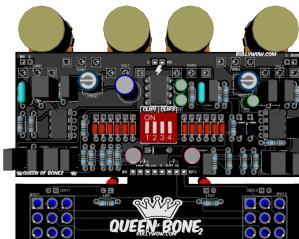
LED leads bent 90 degrees – won't fall out

**IMPORTANT!** Be absolutely sure the 3PDT switches are oriented so the lugs are horizontal. The holes in the lugs should run north to south:



Test the 3PDT board for fitment in the enclosure. Remove the outer nuts, and any washers which should attach from the outside of the enclosure from the switches. It is a good idea to ensure that both switches are mounted at the same height.

**IMPORTANT!** The 3PDT daughterboard should be have the Queen of Bone logo facing the lid. You should not see the “text quotes” when installed in the enclosure. Boards should look like this when together



Once you are satisfied with the 3PDT board fitment, reinstall the main PCB **inside** the enclosure using the 6 pot nuts and washers. Carefully lower the 3PDT board into its holes, lining up the main PCB header pins with the holes in the 3PDT board. Secure the 3PDT switches with the nuts from the outside. Once you are satisfied everything is lined up, the header pins are inserted, and the boards are parallel, solder the 3PDT lugs to the breakout PCB. Alternating the soldering from one switch to the other side will help reduce heat on the switches and reduce chance of failure by overheating. Try to use a high heat (750F or higher) and not spend more than 3 seconds on each lug. Solder the header pins to the 3PDT breakout PCB. Lastly, position your two LEDs through the holes in the enclosure (a small screwdriver is helpful here) and solder them in place. Trim the excess length off the header pin leads as well as the LED leads.

7. In/Out jacks and DC power. Using wires, solder the connections to the in/out jacks and DC power jack. Pay attention that the ground (G) is soldered to the in/out jack sleeve. Be sure the jack is not shorting out to the main PCB.

**Pro Tip:** If you use a DC jack which has a nut on the **outside**, it will make taking the entire PCB assembly in and out of the enclosure very easy as you won't need to desolder anything if you need to take the PCB assembly out of the enclosure.



External nut DC jack

If you need more assistance, please check out the helpful video on YouTube. Link here:  
<https://youtu.be/SYShhTDP0lw>

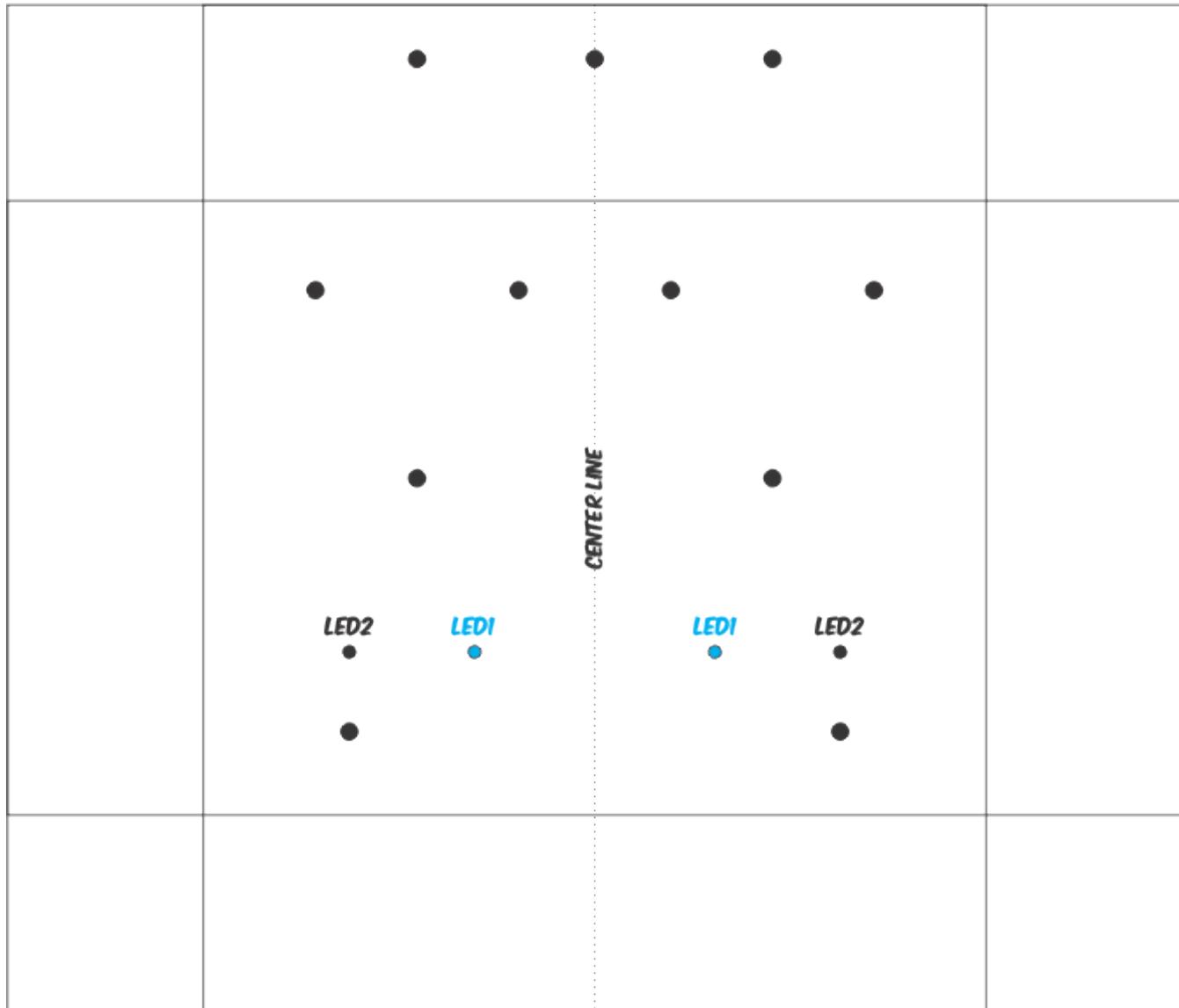
# BILL OF MATERIALS

---

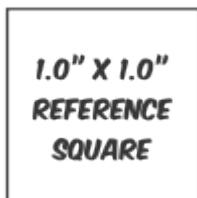
## QOB2 Component List - Sorted by Value

Capacitor			Resistor			Diode		
C6	100n	film	R5	10k		D1-D4; D7-D10	MA856	see notes
C15	100n	film	R8	10k		D5,D6,D11,D12	1s1588	see notes
C3	100pF	ceramic	R16	10k		D13-15	1n5817	
C12	100pF	ceramic	R19	10k		LED1, LED2	3mm or 5mm	only 2 LED required
C1	100uF	electro	R11	1k				
C2	10n	film	R22	1k				
C4	10n	film	R3	1M		PRES1, PRES2	50k	3362 type or similar
C5	10n	film	R4	1M		TONE1, TONE2	25kB	16mm Alpha PCB pin
C7	10n	film	R13	1M		VOL1, VOL2	100kA	16mm Alpha PCB pin
C8	10n	film	R14	1M		GAIN1, GAIN2	100kB	16mm Alpha PCB pin
C11	10n	film	R15	1M				
C13	10n	film	R24	1M		IC1, IC2	4558	DUALTH
C14	10n	film	R9	220k		IC3	TC1044 or MAX1044 or ICL7660	
C16	10n	film	R20	220k				
C17	10n	film	R7	27k		DIP	4pos	2.54mm spacing
C20	10uF	electro	R18	27k		3PDT1, 3PDT2	9 lug stomp	standard stomp
C21	10uF	electro	R6	33k				
C22	10uF	electro	R17	33k		Header Pins	Long Type	2.54mm (0.100") spacing
C9	1uF	electro	R1	47k				
C10	1uF	film	R2	47k				
C18	1uF	electro	CLR1	4k7				
C19	1uF	film	CLR2	4k7				
			R10	6k8				
			R12	6k8				
			R21	6k8				
			R23	6k8				

# QUEEN OF BONE2 DRILLING TEMPLATE

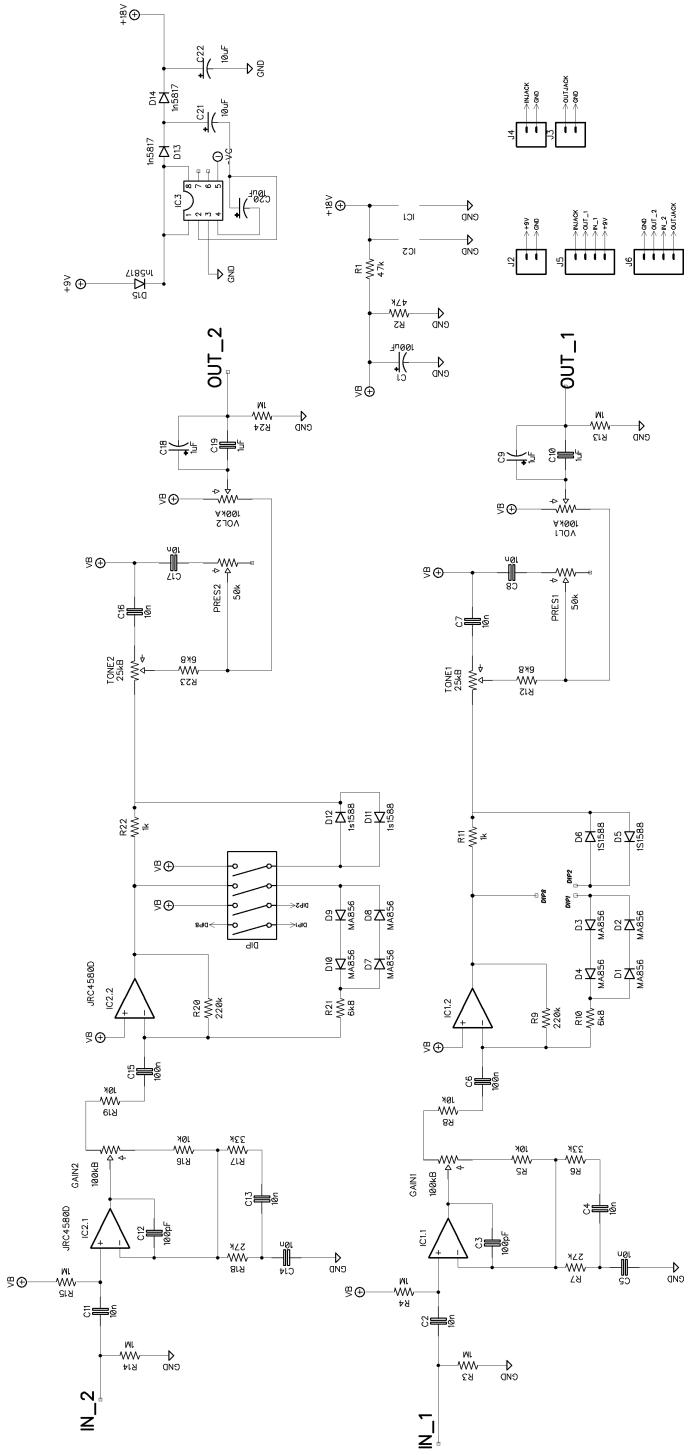


**BE SURE TO DRILL ONLY ONE PAIR OF LED HOLES...NOT BOTH!**



# SCHEMATIC

---



**QUEEN OF BONE2**