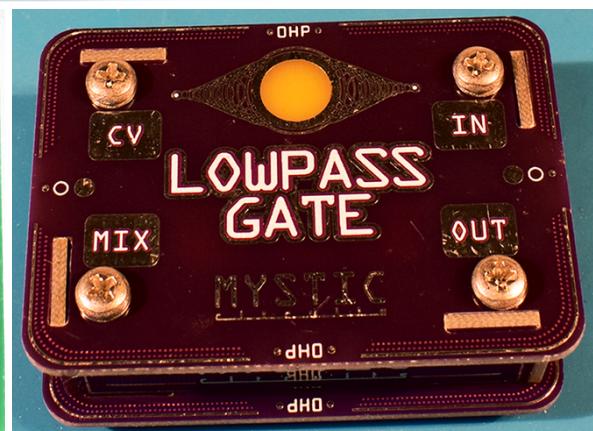
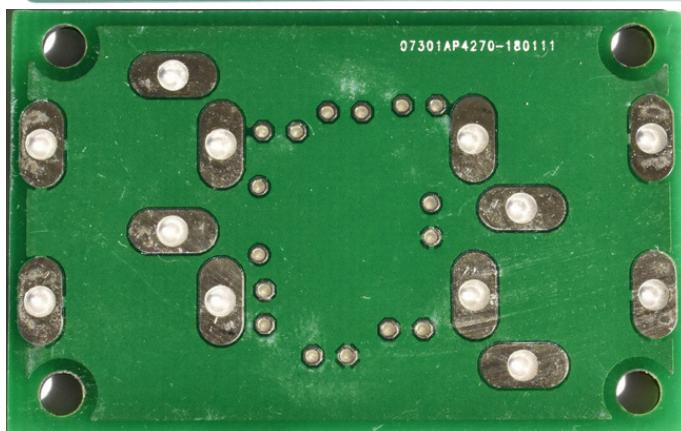
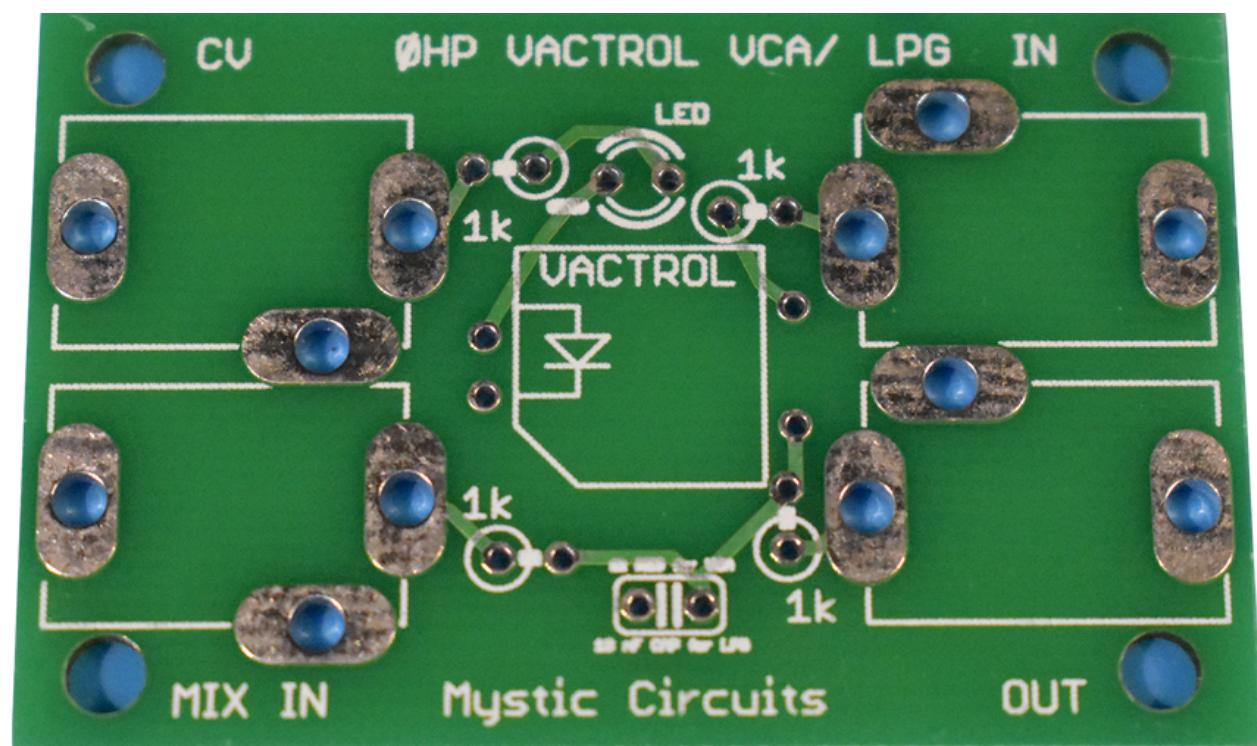


Build Guide

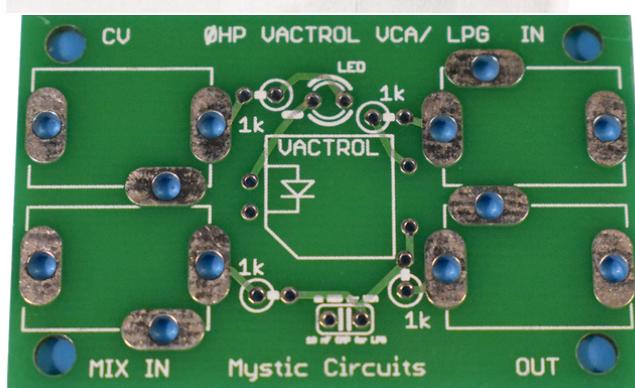
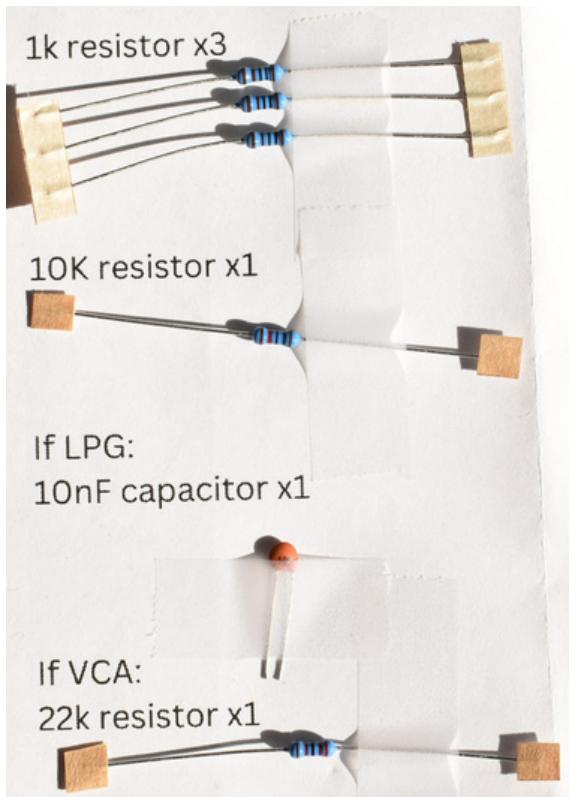
Vactrol VCA

Thank you for purchasing this OHP DIY kit from Mystic Circuits. This build guide is intended to help take you through the steps in order to make a full functional OHP module. You should have a soldering iron, solder, pliers, snippers, and a small screw driver in order to complete the build.



Start by verifying that you have all necessary parts for the kit. Packing kits is the kind of monotonous task that is very easy to make mistakes with, we have multiple methods to verify that the correct parts make it into kits but mistakes still happen. If you are missing any of the parts below please contact us through our contact page and we will remedy the situation. You will have to open all of the bags in your kit.

3x 1K Resistor
1x 10K Resistor
1x 10nF capacitor
1x 22k Resistor



ACTIVES:
1x LED
1x Vactrol



1x Main PCB

4x Rubber Feet
4x M3 Hex Nuts
4x M3 23mm Bolts
4x 3.5mm Mono Jacks
2x Short PCB Case Sides
2x Long PCB Case Sides
1x Generic PCB Case Bottom
1x Vactrol PCB Case Lid

(Case lids are dual side so you might have to flip the lid over to see the correct design)



Resistors:

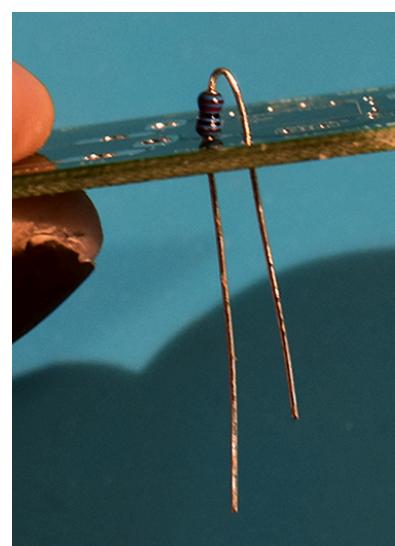
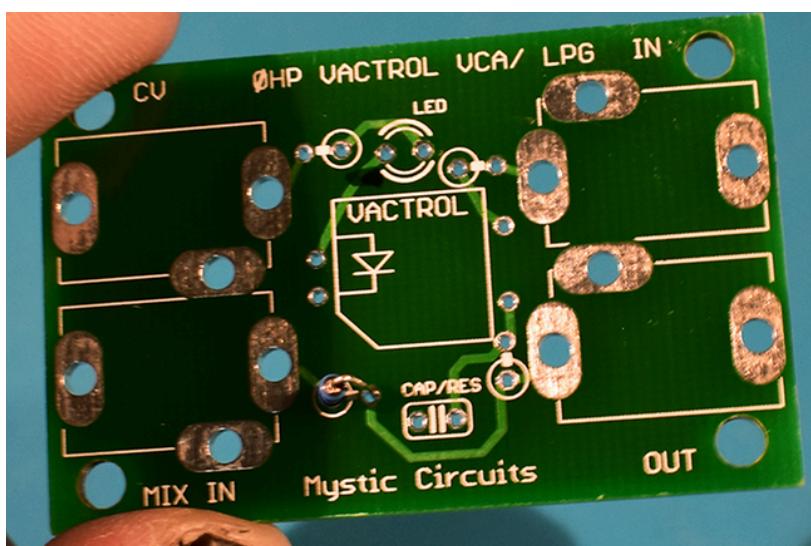
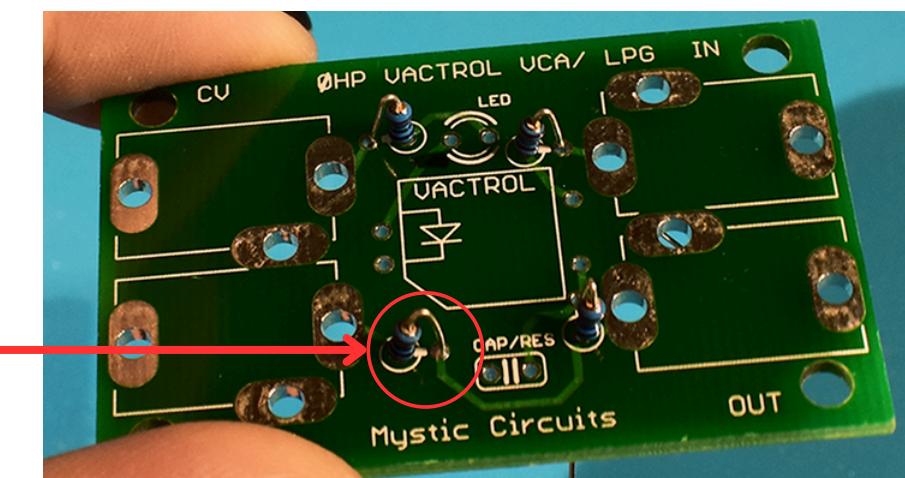
Please note: While the silkscreen says that all the resistors are 1K, the resistor next to the Mix In is actually a 10K resistor. It's smart to start with this resistor to make sure that it is done correctly.

Bend each resistor in order to install it in the PCB.

One leg should go through the board vertically, while the other will be bent over to go through the adjacent hole.

Next bend the legs away from each other in order to secure the resistor to the board. Once the resistors are populated flip the board over and solder, being careful not to get extra solder into any empty holes in the board. After the resistors are soldered, use your snippers to remove the part leg.

10k RESISTOR

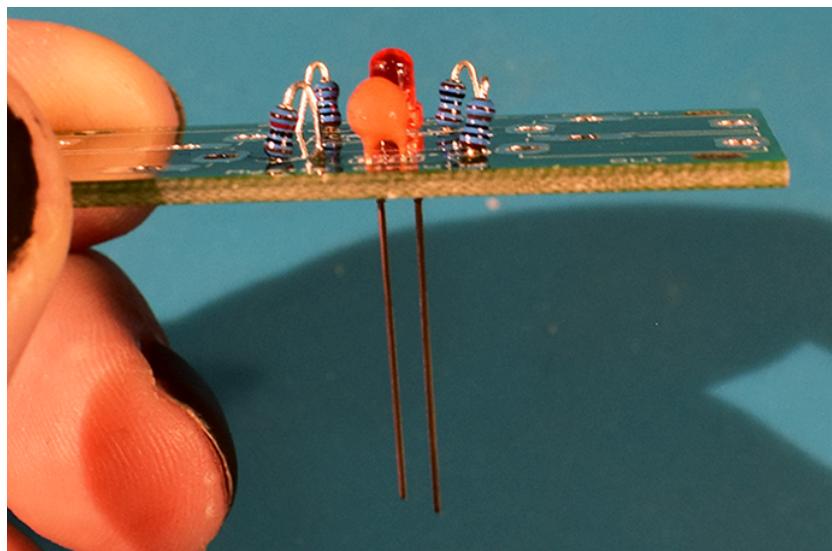
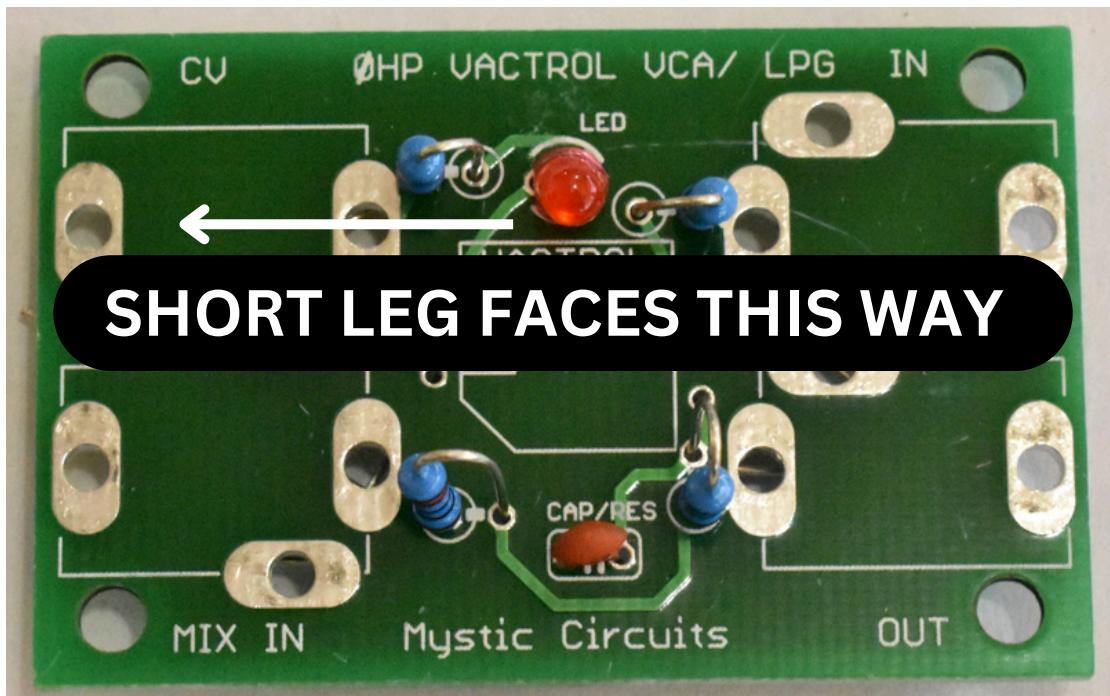


This build gives you a little bit of freedom, do you want a low pass gate with a slight amount of filtering, or a more straightforward VCA? The process is nearly the same for both, just with a different selection of parts. In the part marked “cap/res” you can either install the 10 nF capacitor (low pass gate) or the 22k resistor (VCA).



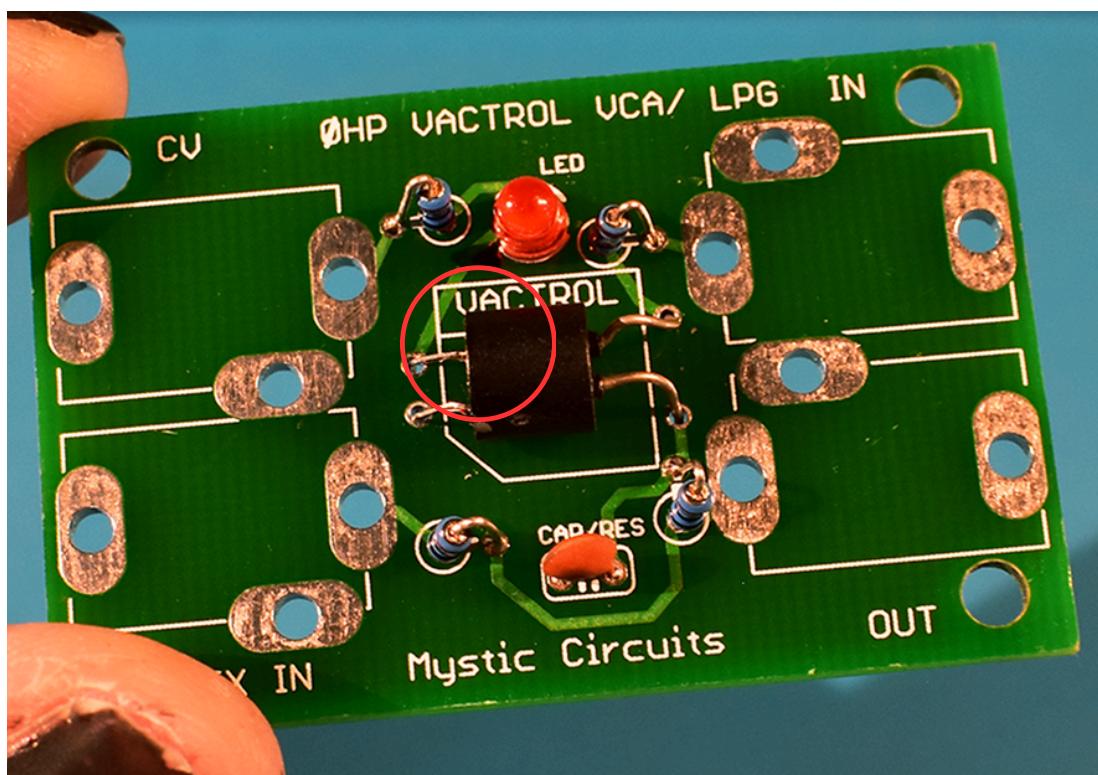
Solder in your part and then cut off the legs.

LED: LEDs (light emitting diodes) have two different sized legs, the longer leg is the positive anode and the shorter leg is the negative cathode. This means that the LED needs to be put in the board in a specific direction, with the shorter leg going into the hole on the left and the longer leg going to the hole on the right. Some of our circuit boards do not have the relevant mark to indicate the LED direction while others do (I blame Eagle). Either way your short leg should be pointing to the “CV” jack on the left side of the board. Make sure that this is done correctly or else your board will not work. Once you have verified the orientation of the LED, solder, and snip legs.

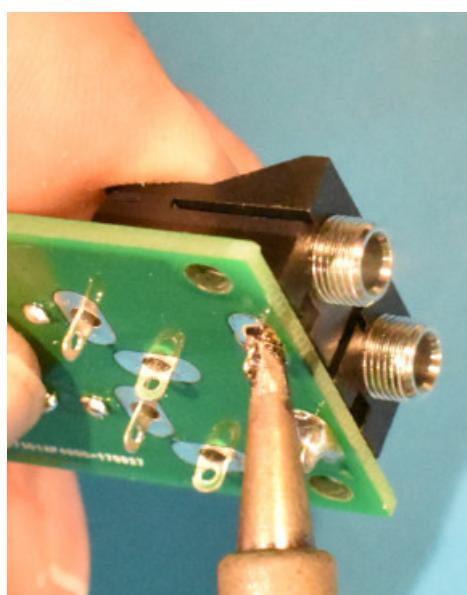
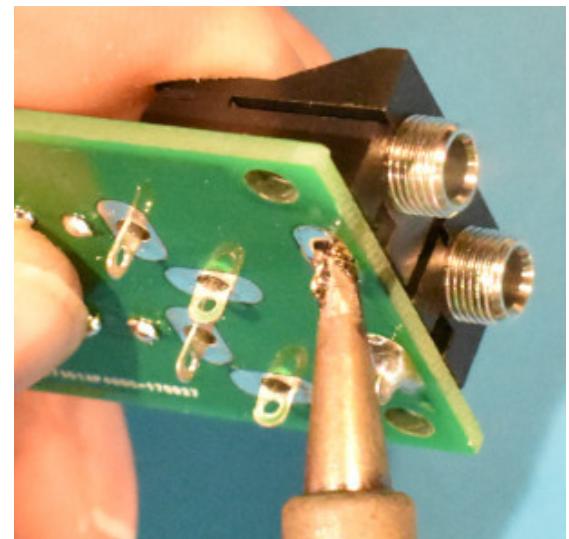
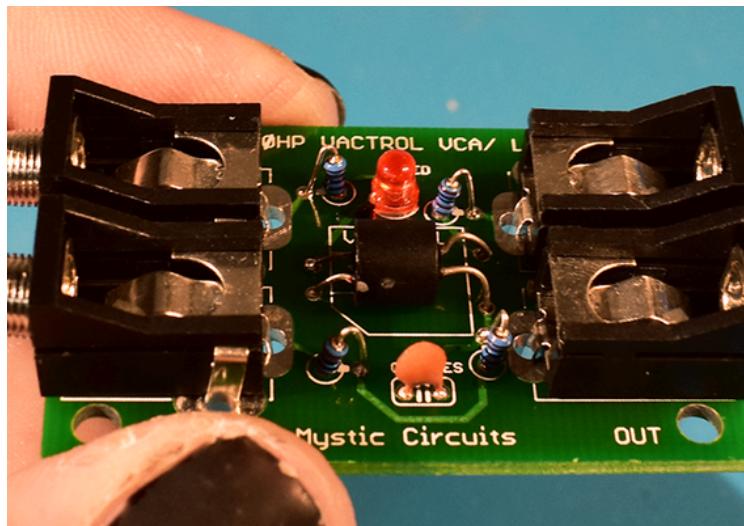


Vactrol: A vactrol is essentially a light emitting diode (LED) and a photosensitive resistor placed next to each other in a light-proof case. Applying a signal to the LED turns it on and allows the signal present on the other side to pass, creating a simple voltage controlled amplifier. The LED has a slight amount of decay as it goes from off to on, creating the characteristic vactrol tail present in these designs.

Vactrol cases have slight differences but there will always be a dot on the outside of the case marking the negative cathode of the LED. This dot should be pointing toward the bottom left corner of the PCB, matching the corner of the white vactrol square with a diagonal corner. Place the vactrol into the PCB being careful of the orientation and then solder and snip the legs. Make absolutely sure that the vactrol is oriented correctly before soldering - if this is wrong your board will not work.



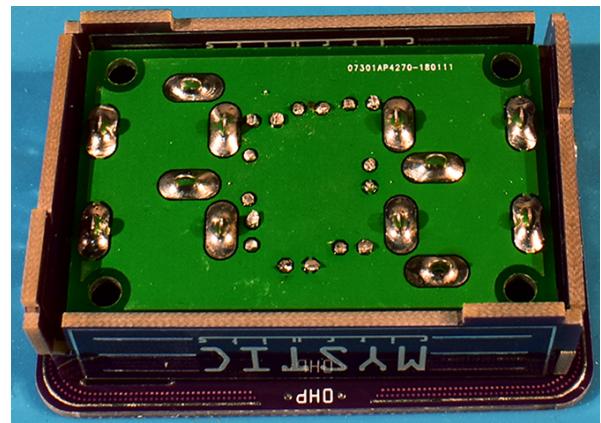
Jacks: Place jacks through the holes, push them flush with the board and then flip the board over. You may have to adjust the jacks for them to sit flush with the PCB. Apply solder to the front legs on each of the four jacks. Flip over to check the jacks are still flat against the board, you may have to press down on the jack and reheat the solder in order for it to sit flat against the board. Be careful as it can get pretty hot. Then solder the other jack legs in order to secure them to the board.



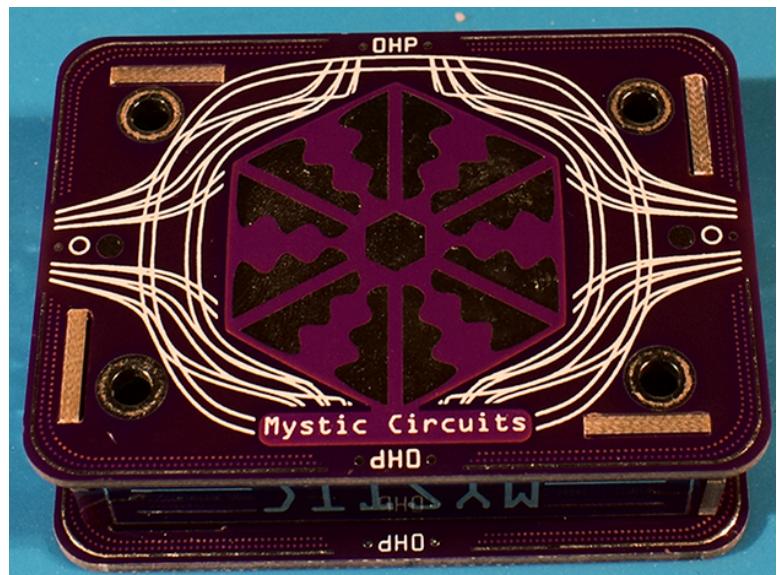
CASE ASSEMBLY: First place the jack sides onto the jacks, making sure they line up correctly. Then place the top of the case onto the sides, lining up the slots.



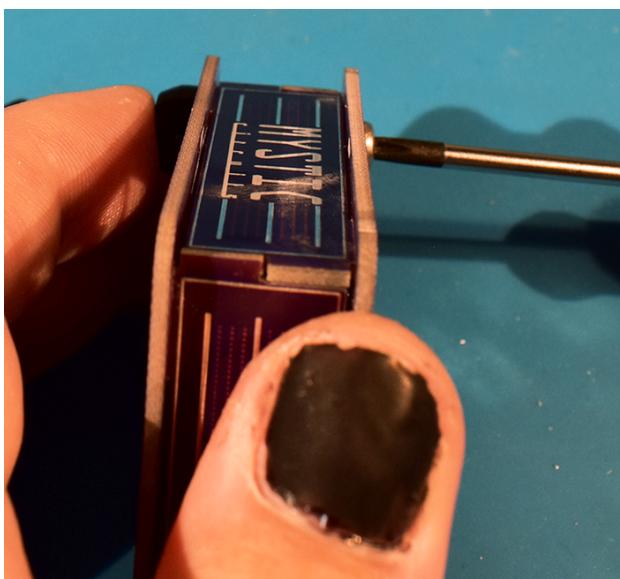
Next, slot the longer sides so they line up with the top and jack sides. Once all the sides are secured to the top, flip the case over.



Line the bottom of the case up with the side slots and push to install. Make sure the slots on the top and bottom of the case line up correctly and there are no gaps.



SCREWS: Place one screw in one of the corner holes, but don't push all the way through. Place the nut in the rubber feet and place that over the hole. Place your index finger over the rubber foot so it keeps the nut from moving. Push and rotate the screw so it lines up with the nut and attaches securely. You may have to remove the screw and try again if it is not lined up with the nut correctly. Repeat this step for each of the feet.



You are done! Go make some tunes!

