

THE QU

The Qu is a 10hp Phaser and sound mangler inspired by Love Tone Doppelganger phaser pedal from the 90's and the works of Turkish sci-fi writer CM Kosemen¹ (who also kindly did the artwork for the panel). If you are unfamiliar with his work I suggest before even building this to check it out.

The Qu module has an input, two outputs (which you can use for stereo dudites, one is clean and the other one is distorted and thus slightly louder as well), two cv inputs (cv to the left is normaled to the right when only using one input). There are two switches, one for removing the dry signal from the circuit (creates vibrato type effects in phaser mode, does

¹ https://www.youtube.com/watch?v=imNtSPM3-r4&ab_channel=AltShiftX

weird stuff in Qu mode). The other switch, labeled Qu is to switch the Qu on or off. When on the Qu are attacking, mangling and destroying the sound, sometimes trying to mimic the input but ultimately failing and instead morphing it into a "new species" (just like in "All Tomorrowa). The right cv input will have a slighter more impact on the signal then the left in Qu mode but both will effect the signal. There are three pots on top, two for feedback and one for Mix. Mix is important in Phaser mode since the phaser effect relies on a dry signal being present. Also phaser sounds are usually more pronounced with more complex sounds: chords, rich waveforms, drums or vocals. Don't expect much action by inserting a straight sine from Bristol Bloodhound (the wave folded out though: HELLO!) The feedback pots are for the Qu mode, it will affect the sound depending on incoming signal and other setting. All knobs and even the wet switch will interfere in Qu mode. Have fun exploring thresholds with this one. Play around. Have fun.

Build guide:

Building this is pretty straight forward since all SMD components are pre-soldered:

Place the power connector according to the silk screen and solder it in. Make sure no joints are touching.

The I usually do the LDR and LEDS on the back panel. There is nothing wrong or right here as long as the LED's (I prefer flat top ones than rounded to get a closer fit). Not that LED D11 has two LDRs "hugging" it. You have several options here, one on top, the other on the side of the LED or both on the side of the LED. I've done both and both works equally well. Just make sure no LDR legs are touching each other.

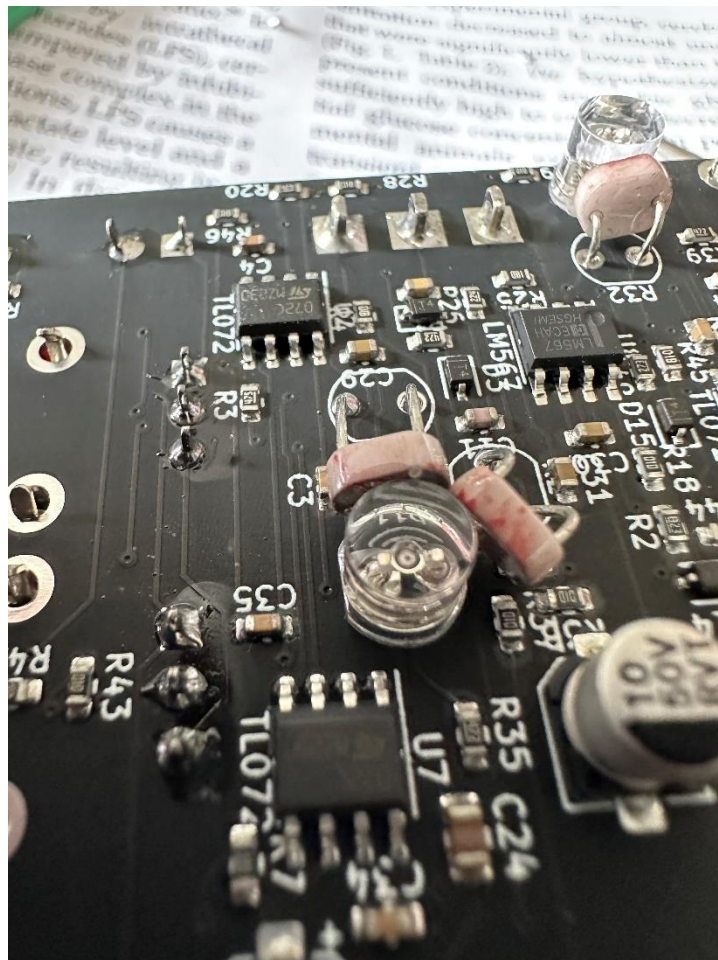


Figure 1: This image is suggesting how to position your LDRs with the corresponding LED

I then usually place the potentiometers first and solder only one pin from the top of the board, watch out touching the SMD components with your iron in crowded areas. That way they will not fall off when we turn the PCB. Then I do the same with all jacks, just soldering the ground connector (the one that is sticking out form the jack and has a square pad).

Double and triple check that all solder joints on the connectors look good and that all pins are properly soldered in. If you have some type of magnification that usually helps.

I then place the switches but don't solder any pads on those, let them wiggle but don't fall out. I then place the tactile switch try make it as straight to the board as possible without the actual button placed.

Mount the panel, align the switches so they are straight, pots and jacks as well, insert the LEDs for the front panel, make them peep out beneath the panel. Solder everything in. Double check, plug it in, voila!

Also, the holes for this panel might be 0.5mm off on one side (first version). If you are having problems mounting it, simply start with one side (up or down), insert a screw but don't tighten it too much, go to the other side, do the same, back to the first side fully tighten and go back and do the same. That way the panel will slowly align itself to you rails. I haven't had any problems using these panels at all in my Aturia racks but I've heard from people using other rails that it might be an issue. If everything else fail you can use a small rounded file and make top or bottom holes 0.5mm larger since the panel is in pcb material this should be very easy to do.

BOM

Designator	QTY	INFO	Product link
J1-J5	5	Thonkiconn jacks	https://www.thonk.co.uk/shop/thonkiconn/
RV1-5	5	100k Lin Pot T18 9mm Alpha	https://www.thonk.co.uk/shop/alpha-9mm-pots-vertical-t18/
SW2-3	2	ON-ON SPDT	https://www.taydaelectronics.com/mini-toggle-switch-spdt-on-on.html
For RV-1-4	5	Must be Black	https://www.thonk.co.uk/shop/micro-knobs/
SV1	1	10pin eurorack header	https://www.taydaelectronics.com/2x40-pin-2-54-mm-double-row-pin-header-strip.html
R11, R15, R24, R32, R41	5	GL5537 - LDR	https://www.tinytronics.nl/shop/en/sensors/optical/light-and-color/gl5537-ldr-light-sensitive-resistor
D8-D10	2	3mm Led White (Kosemen prefers Purple)	https://www.taydaelectronics.com/led-3mm-white-water-clear-ultra-bright.html
D6-D9	4	5mm LED Red flat top (rounded might work as well)	https://www.ebay.com/itm/254834860808?var=554725598366

