PDX Synth DIY Workshop
Mutated: Ripples Build Guide

Mission Statement

Knowledge IS Power.

It is the intention of this workshop to show you that it is not beyond your ability to successfully solder surface mount DIY projects and that it can actually be easier than working with through hole, in practice. With a little background knowledge, and some initial guidance, we aim to empower you to take on further projects independently with confidence.

This workshop also seeks to connect like-minded builders to form a support network, because when we work together and share our knowledge we're able to accomplish more.

The Heat is on.

Contents of kit:

1. This build guide
2. PCB silkscreen blowup for placement reference (Appendix I)
3. The Bill of Materials (hereafter referred to as the BOM)(Appendix II)
4. Ripples PCB
5. Ripples panel
6. One bag of hardware including
    - 9 ea. PJ301-m jacks and mounting nuts
    - 3 ea Sifam knobs and end caps
    - 3 ea Alpha 9mm potentiometers, 10k B (linear) 15mm D-shaft
    - 1 ea 5k trimmer horizontal adjustment, 25 turn.
    - 2 ea 47 uf electrolytic capacitors 25V rated
    - 1 ea 4.7 uf electrolytic capacitor 16V rated
    - 1 ea 10 pin power header
    - 1 ea 16 pin to 10 pin Eurorack power connector
7. One Bag of surface mount components labeled
7. One coil of 0.02" *Kester* "245 no-clean" solder (63% tin 37% lead)
7. One pair of curved tweezers
7. One flux pen
7. De-soldering braid

Designed by Olivier Gillet 2015, version 4, cc-by-sa
Width: 8 HP
Current draw: +12: 35mA, -12: 35 mA

The Mutated: Instruments Ripples filter is a multi-mode analog filter heavily inspired by the Roland SH-101 and Jupiter-6 filters. Available filter modes are 2 pole low pass (12db/ octave roll off), 4 pole low pass (24db/ octave roll off) and band pass (12db/ octave roll off). Ripples also includes a built in VCA fed from the the output path of the 4 pole low pass output. All filter modes self-oscillate and the 4 pole outputs a clean sine wave. Ripples has built in loudness compensation when resonance is increased to avoid the amplitude drop common to 4 pole analog low pass filters.

Build instructions:

We're going to build Ripples in several stages: the surface mount parts, the through-hole parts, and the final assembly.

The surface mount parts include ICs, resistors, diodes, and capacitors (which come in polarized and non-polarized varieties).

The through-hole parts are the pots and jacks, plus the power headers and trimmers.

Surface Mount Parts (SMD)

ICs

See appendix III for a guide on soldering SMD ICs

Beginning at the top edge of the module identify the pads labeled IC1 and locate one of the TL072 ICs from the kit. For this module, all the ICs will go onto the board oriented so that the writing is right-side up and readable and any indentation or silk-screened edges are oriented to the left.

If you have any parts that are improperly aligned after soldering the initial pin just take the tip of the iron and reheat the solder on the pin and use the tweezers to realign the IC before continuing to the opposite corner's pin. This will save effort in the long run.

Once this first pin looks good, solder the opposite corner pin in place and proceed to solder the remaining pins one by one.

Finish populating the rest of the ICs top to bottom one by one using this method, making sure to double check that each one is the correct part and is placed in the correct orientation, before beginning to solder.

(**Wait to do the voltage reference IC7 for now**. We'll get to that one soon).

Passive components:

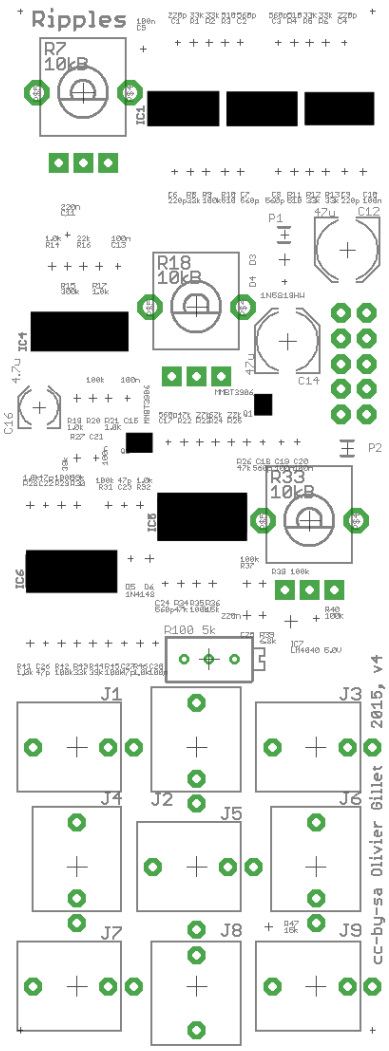
See appendix III for a guide on placing passive SMT components.

Resistors

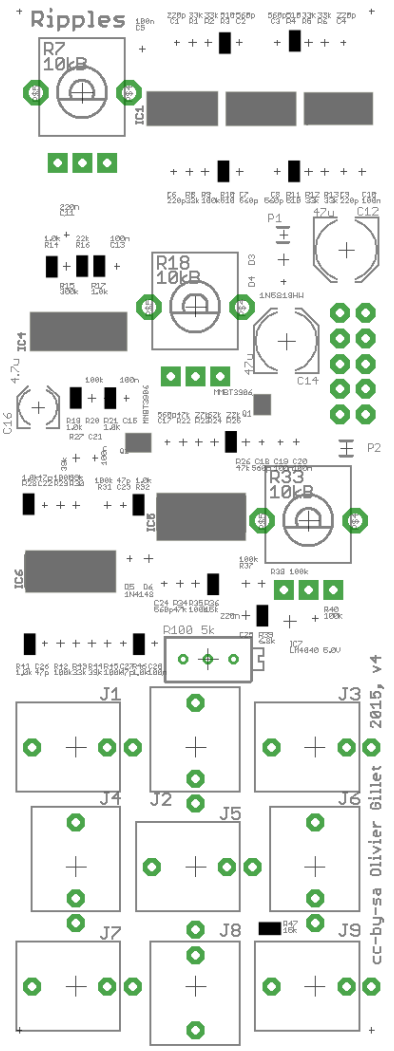
We're going to do the resistors in a couple of stages: values 0 ohms to 25k ohms, 25k ohms to < 100k ohms, and > 100k ohms.

We recommend tinning one pad of each component per stage before you start placing - this helps things go quicker, as you're tinning, then placing, instead of going back and forth.

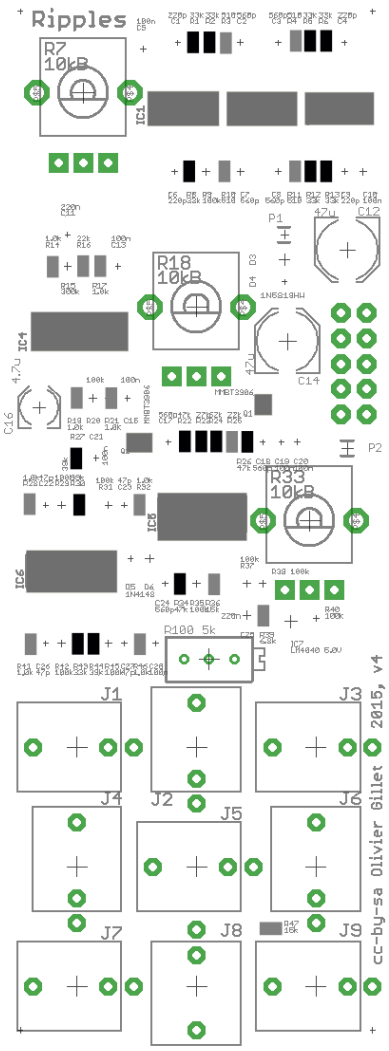
ICs

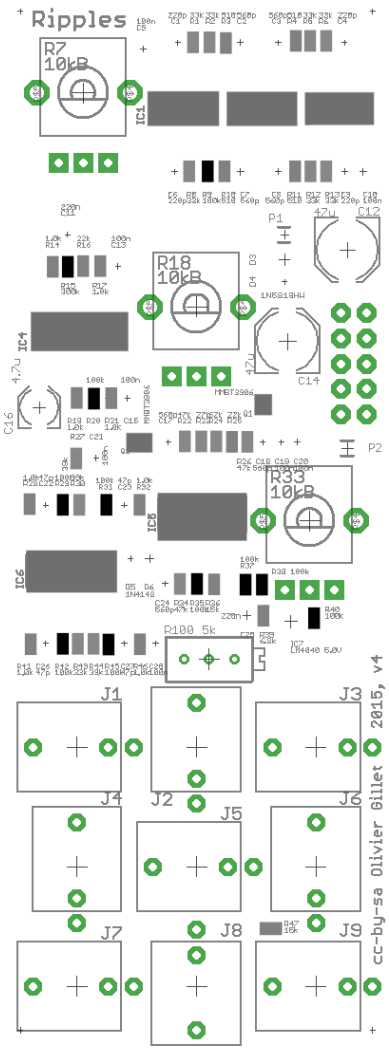


Resistors with values under 25k Ohms



Resistors btwn 25k and 100K Ohms / Resistors 100K Ohms and greater

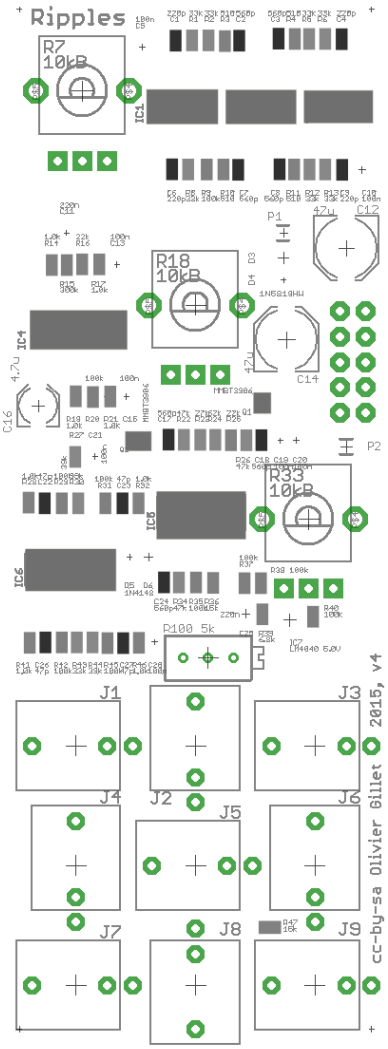




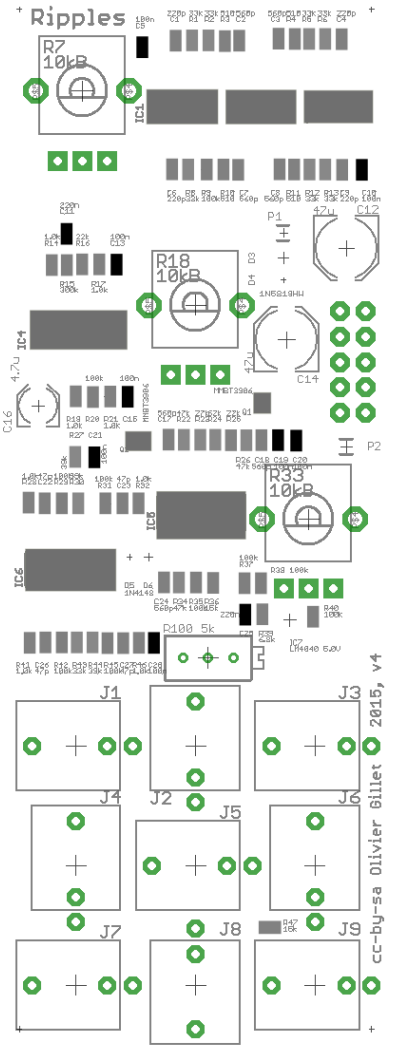
Non-Polarized Capacitors

Next we'll do the non-polarized capacitors. Like the resistors, we're going to do these in a couple of stages.

47p, 22p, and 560p Capacitors



110n and 220n Capacitors



Fuses, diodes, voltage reference and transistors

First place the resettable fuses P1 and P2. These are non-polarized meaning, they can go either way, so feel free to express yourself.

Next populate the diodes, D3 and D4. **THESE ARE POLARIZED - the greyish stripe on the top of the component should be aligned with the white stripe silk-screened on the pcb.**

(NB D1 and D2 are listed as DNP meaning do not populate.)

Next identify Reference Voltage IC7. This is an IC of the SOT23 (**S**mall **O**utline **T**ransistor) form factor. It is a smaller rectangle with 3 legs, one placed centrally along its length, two at opposing ends of the opposite edge. Place it like the larger ICs in the first step, starting with the center leg.

Follow the same procedure with the PNP transistors Q1, and Q2, located above IC5 to the right and left. Note that these share the same package type as the voltage reference.

Electrolytic Capacitors

The last surface mount parts are the electrolytic capacitors. These look like little cans. There is a black stripe up the side of the can *opposite* the narrow point of the base, which indicates the negative lead.

These are POLARIZED parts and go one way on the board.

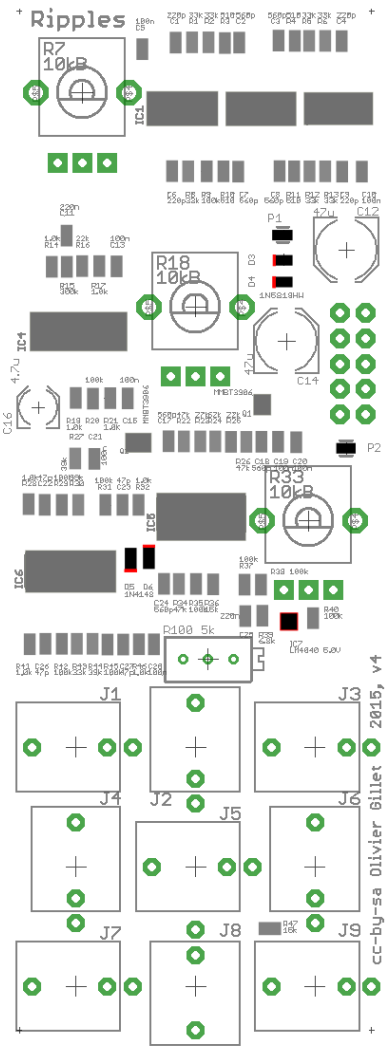
(Don't over-stress this - just line them up with the silkscreen).

While holding one of the caps in place with the tweezers, flux the pad nearest to the iron and use the 'wet tack' method (more info in Appendix III), transferring melted solder from the tip to the pad and pin.melting solder to the tip and then transferring it to the pin.

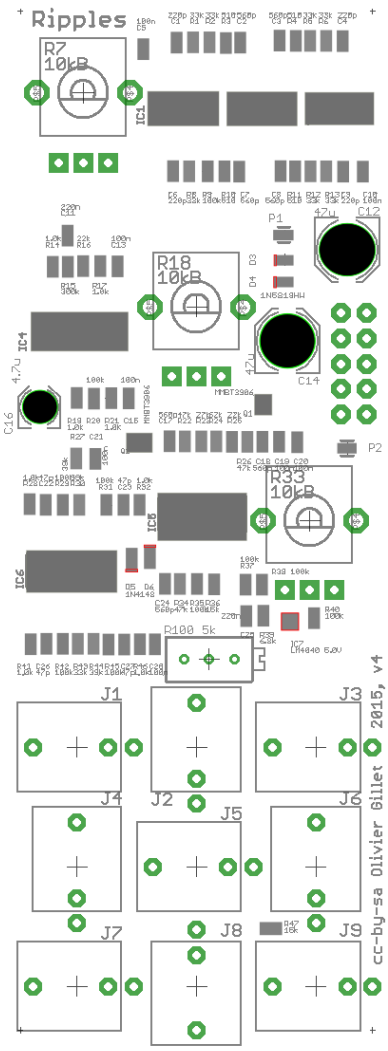
The 2 larger 47uf capacitors,C12 & C14, are placed with the negative stripes facing the away from each other, toward the top and bottom edge of the module. (Or just follow the silkscreen).

The smaller 4.7uf capacitor, C16,goes to the left, just below the other two. (Unlike the other two electrolytic capacitors, this one is non-polar. This makes no difference for our build, because of the way the packages are shaped, but it's interesting!)

Fuses, Diodes, Regulators



Electrolytic Capacitors



Once all three electrolytic capacitors are placed, we are done with the hard stuff! Great job team!

Through Hole components

Trimmer Pot

Place the trimmer through the board so the adjustment screw aligns to the right (as per the silkscreen), then solder the 3 pins from the back of the board.

Ten Pin Power Jumper

Populate the 10 pin power jumper (JP1) so that the short ends of the pins are down, and then tack one pin in place with solder and check for alignment, before proceeding to the last 9 pins. If the jumper isn't aligned properly, you can push some of the unsoldered pins towards the board while reflowing the tacked pin and it should pop into place. **This will be the only part populated on the back side of the board**.

Pots and Jacks

Remove the mounting nuts and populate all 3 of the 10k potentiometers, but **do not solder any of the pots**. Notice they have a pair of kinked larger legs. These should snap onto the pcb and hold it in place until it is soldered. You may find that you need to spread the kinked legs apart if they are too snug to snap easily onto the board. This can be done with needle nose pliers. Once the pots are snapped on, begin populating the jacks, noting that the orientation alternates by 90°.

Do not solder any of jacks at this point. Instead once populated, put the front panel in place on the module and install the mounting nuts for the pots as well as the jacks, tightening them with your fingers . You may need to adjust the position of the components relative to the PCB to get everything to sit flush with the panel. As long as the pins remain in contact with the pcb holes this is not a problem, and it is worth noting that this step is a worthwhile habit to get into whenever building a DIY Eurorack module.

Check power pins for shorts

Now we will take a moment to check the module for short circuits. Using a multimeter on the continuity setting, place either of the probe leads between the middle 3 pairs of pins (the ground pins) on the 10 pin connector, and with the other lead, touch the top (positive 12 volts power) and bottom (negative 12 volts power) pair of pins.

Next touch one lead to the top pins and the other to the bottom pins.

Finally, repeat this process with the multimeter leads swapped (so if you had positive lead to ground, go negative to ground, and vice versa).

This detects any faults that might otherwise be masked by reverse power protection diodes or similar circuit features (it's often redundant… but it's good to be sure).

If the multimeter beeps during this process, this indicates either a short... or that you may have touched the leads together inadvertently. It is best to try this process a couple times to ensure you get consistent results before going to town troubleshooting. If indeed there are shorts, first check to see that the diodes orientation is not reversed.

If that was not the issue the next step is to look is for correct orientation on ICs, along with a thorough examination for bridged pins. Use a microscope or jeweler's loupe, if available.

Solder it up
Once the panel is snug in place and you have checked for shorts, flip the module over and solder all jacks and pot pins from top end to the bottom. The larger holes on the pots should be filled in with solder to lend structural support, but do not have any electrical properties.

Testing and calibration

Check once more for shorts before connecting power and testing the module for functionality.

Testing
Begin by setting the FREQ knob fully counter clockwise, and RES to about 12 o'clock. Connect an oscillator to the input and connect the LP4 output to your monitors, making sure to set the volume to a conservative level. Sweep the FREQ knob clockwise - the oscillator should become audible and change in timbre to unfiltered as it gets to the top of the cutoff range. Next turn the FREQ back down and increase the RES to full. Slowly increase the FREQ to about 10%. A deep sine wave should become audible, increasing pitch all the way through the audible range around 90%. Connect the LP2 and BP outputs and repeat the process. LP4>VCA should be identical to LP4 until a patch cable is inserted into the gain input, at which point the output at LP4>VCA should become muted. A CV signal should increase the amplitude at this output.

Connect an LFO to the 3 modulation inputs and make sure they all function then proceed to calibration.

Calibration

1. Set the FREQ knob to 10 o'clock (exact position does not matter).
2. Set the RES knob to its maximum position. Disconnect any input signal or FM modulation CV.
3. Connect the note CV output of a well-calibrated keyboard interface or MIDI-CV converter to the FREQ input.
4. Listen to the tone at the LP4 output. Adjust the trimmer resistor at the back of the circuit board until the musical intervals played on the keyboard are correctly reproduced (the actual note values do not matter, but when playing an octave, it must sound like an octave)

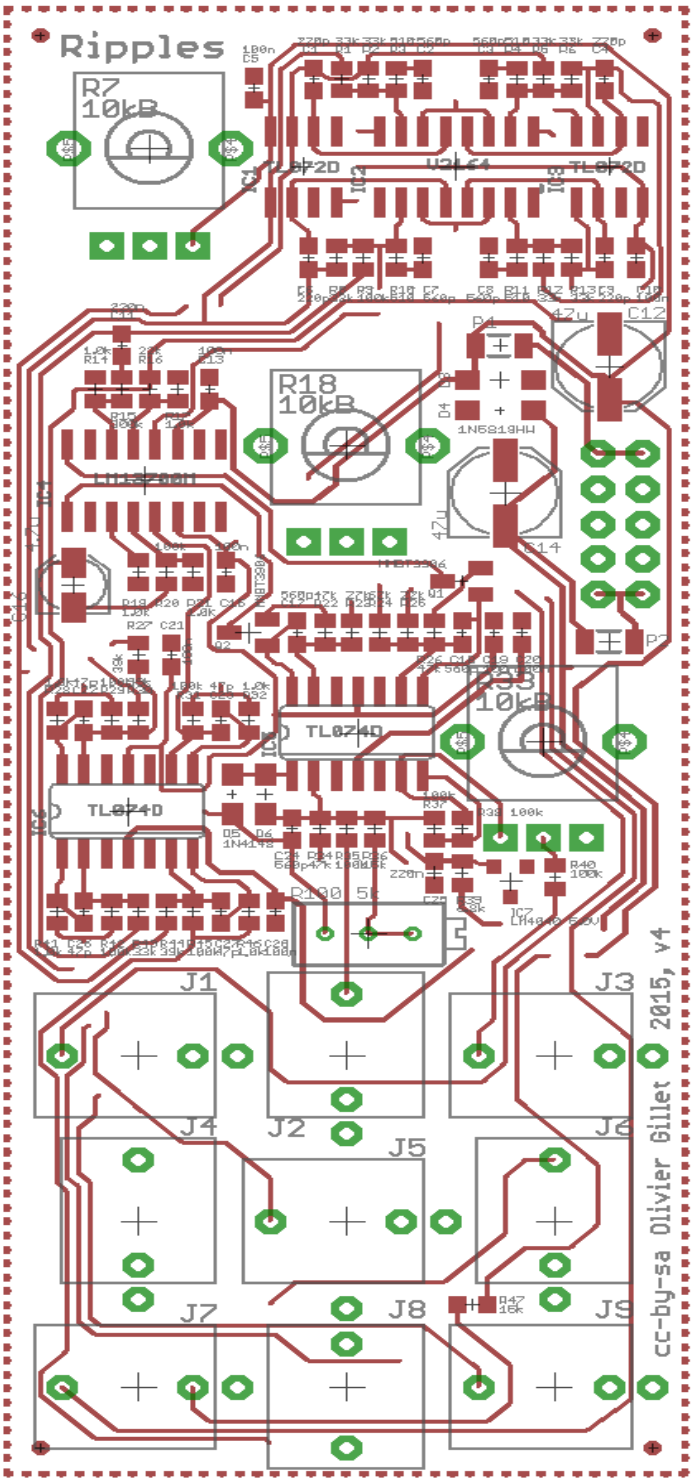
Note that the filter is designed to track well over 4 octaves, but is not temperature-stabilized.

Installing the knobs

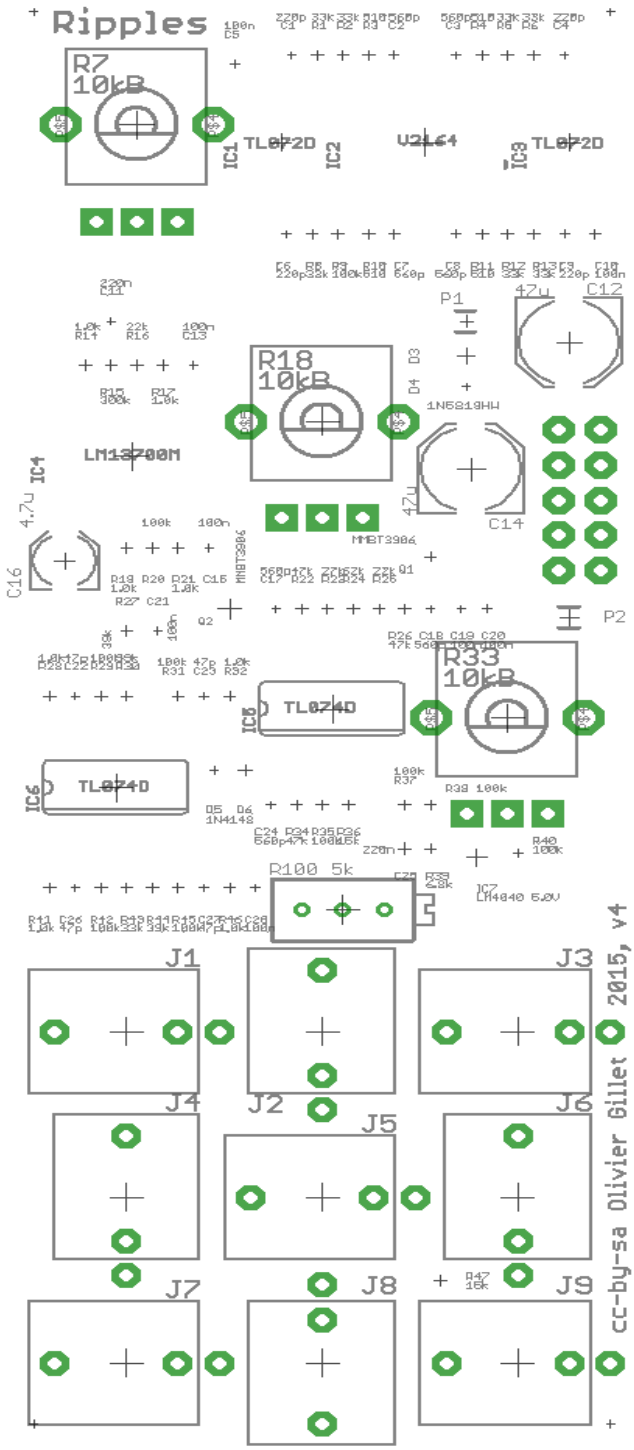
Finish tightening the nuts on the panel with needle nose pliers. Install the knobs by placing the colored end caps in place and pushing them gently down onto the shaft of the potentiometer.

Congratulations! You have successfully completed a surface mount DIY Eurorack module.

Appendix I
silk-screens for reference



Appendix I (continued)



Appendix II

Bill Of Materials

Index	Qty	Description	Specs	Value	Package	Ref. Mouser
SMT parts						
R3, R4, R10, R11	4	Resistor, 1%		510	0603	667-ERJ3EKF5100V
R14, R17, R19, R21, R28, R32, R41, R46	8	Resistor, 1%		1.0k	0603	667-ERJ3EKF1001V
R39	1	Resistor, 1%		6.8k	0603	667-ERJ-3EKF6801V
R36, R47	2	Resistor, 1%		15k	0603	667-ERJ3EKF1502V
R16, R25	2	Resistor, 1%		22k	0603	667-ERJ3EKF2202V
R23	1	Resistor, 1%		27k	0603	667-ERJ3EKF2702V
R1, R2, R5, R6, R8, R12, R13, R43	8	Resistor, 1%		33k	0603	667-ERJ3EKF3302V
R27, R30, R44	3	Resistor, 1%		39k	0603	667-ERJ3EKF3902V
R22, R26, R34	3	Resistor, 1%		47k	0603	667-ERJ-P03F4702V
R24	1	Resistor, 1%		62k	0603	667-ERJ-P03F6202V
R9, R20, R29, R31, R35, R37, R38, R40, R42, R45	10	Resistor, 1%		100k	0603	667-ERJ3EKF1003V
R15	1	Resistor, 1%		300k	0603	667-ERJ3EKF3003V
C22, C23, C26, C27	4	Capacitor, ceramic	>= 16V	47p	0603	81-GRM185C1H470GA01J
C1, C4, C6, C9	4	Capacitor, ceramic	>= 16V, Tol < 2%	220p	0603	81-GRM185C1H221GA01D
C2, C3, C7, C8, C17, C18, C24	7	Capacitor, ceramic	>= 16V	560p	0603	81-GRM185C1E561JA01D
C5, C10, C13, C15, C19, C20, C21, C28	8	Capacitor, ceramic	>= 16V	100n	0603	81-GRM188R71E104JA1J
C11, C25	2	Capacitor, ceramic	>= 16V	220n	603	81-GRM188R71C224KA1J
C16	1	Capacitor, electrolytic NP	>= 16V	4.7u	Panasonic B	647-UWP1C4R7MCL
C12, C14	2	Capacitor, electrolytic	>= 25V	47u	Panasonic D	667-EEE-FK1V470P
P1, P2	2	Resettable fuse	>= 15V	>0.1A	1206	576-1206L035/16YR
D1, D2	DNP					
D3, D4	2	1N5819HW diode			SOD123	621-1N5819HW-F
D5, D6	2	1N4148 diode			SOD323	512-1N4148WS
IC1, IC3	2	TL072 dual op-amp			SOIC8	595-TL072CD
IC2	1	V2164 quad VCA			SOIC16	
IC4	1	LM13700 dual OTA			SOIC16	926-LM13700MNOPB
IC5, IC6	2	TL074 quad op-amp			SOIC14	595-TL074CD
IC7	1	LM4040 Shunt Vref	A, B or C grade	5.0V	SOT23	595-LM4040C50IDBZR
Q1, Q2	2	PNP transistor MMBT3906			SOT23-BEC	771-MMBT3906T/R
PTH parts, top side						
R7, R18, R33	3	10k linear pot, 15mm shaft				
J1, J2, J3, J4, J5, J6, J7, J8, J9	9	Vertical jack connector				
R100	1	Trimmer resistor, 25 turns, horizontal adjust.		5k		81-PV36X502C01B00
PTH parts, bottom side						
JP1	1	2x5 male header, 2.54mm pitch				649-67996-410HLF
PCB specifications						
PCB		39.4 x 106.8 ; 2 layers				

Appendix III:

Surface Mount Soldering Guide

This is a guide to soldering surface mount projects (**S**urface **M**ount **T**echnology / **S**urface **M**ount **D**evices) with a specific focus on the Ripples project, designed as a supplement to the build guide above.

ICs

Almost all of the integrated circuits in this build are of the footprint type SOIC, or **S**mall **O**utline **IC**. These are rectangular with an even spacing of pins arranged down two opposing long edges of the chip. They resemble little flat bugs or a miniature Andes mint with legs.

To solder a SOIC package chip, begin by placing it on the board with a pair of tweezers, oriented with Pin 1 (usually indicated with a dot) on the appropriate pad.

For Ripples, Pin 1 will always be the lower leftmost pad (so the writing on the chip will be readable left to right versus the silkscreen), but always consult a schematic if you are unsure.

One method is to apply flux to the pin, melt a little solder to the tip of the iron, and transfer the solder to the pin using the tip of the iron. This method is called ‘wet tacking’, and it is easier to do with a small chisel tip iron.

Another method, which we recommend for pin one, is to apply a small amount of solder directly to the pad and line the IC up with the tweezers, then reflow the solder and slide the pin into place. It is important not to remove the iron until it has had a chance to heat up the pin ensuring a good solder bond.

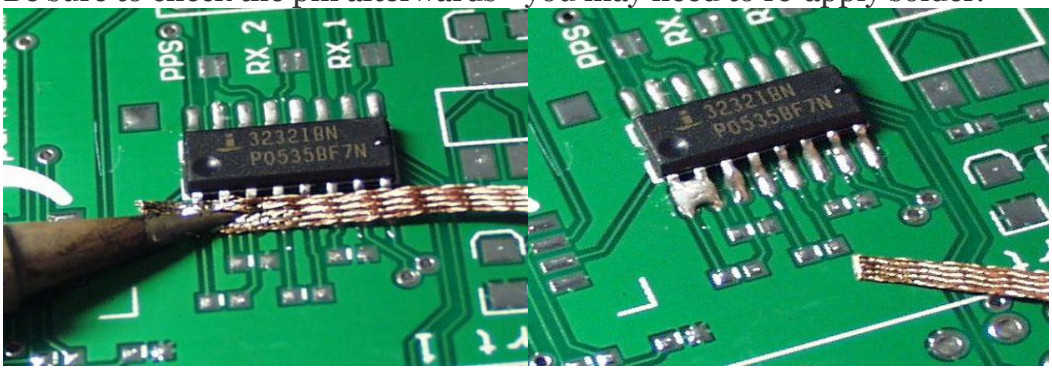
Once Pin 1 is complete, proceed with the rest of the pins, starting with the opposite, by heating the pin and pad with your iron and adding solder until a clean joint is formed.

Check to ensure that the chip has enough solder and has maintained alignment before proceeding. An ideal solder joint will have a fillet* shaped like an exponential curve, sweeping up from the pad toward the top edge of the pin and extending around to the back end of the pin. The solder should be fairly evenly distributed, with no blobs or bridges to other pins.

*The word ‘fillet’ describes the pooling solder melted to the pin and the pad and is pronounced “fill-it”.

Bridged Pins
In the instance that two pins get bridged, there are a number of solutions:

1. Try to add flux and reheat the bridge. The flux will encourage the bridged solder to re-flow onto the individual pins more thoroughly, eliminating the bridge.
2. You may be able to heat the solder and use a solder pump to suck off excess solder from the bridge. (Adding a little bit more solder first can help, as the greater the mass, the longer the solder stays molten).
3. Desoldering braid can be useful for absorbing excess solder from a bridged area on an IC. Starting with a fresh portion and a tinned iron, lay it lengthwise across the bridged pins (be sure to hold a few inches back as copper transfers heat!). Place the long side of the tip directly onto the braid above the bridged pins and allow heat to transfer into the copper braid.
   The braid will absorb the solder and turn silver - you may need to move to an unused portion of braid if there's a lot of solder to clear. (Flux can help as well). Be sure to check the pin afterwards - you may need to re-apply solder.



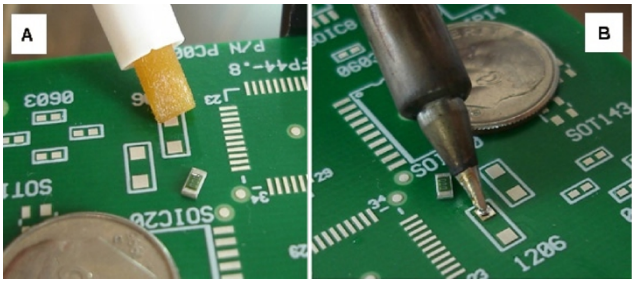
4. If a SMD hot air rework station is available you may be able to paint the pins with flux similarly to method 1 and re-heat the chip with hot air to achieve similar results.

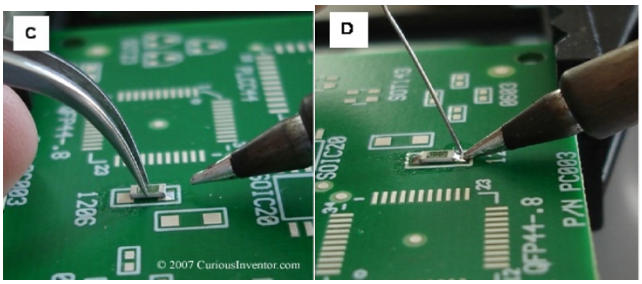
When checking for bridges, don't forget to check the schematic for connections between pads - it may not be a bridge, but an intentional connection. (This is not the case with Ripples: all IC pads should be disconnected from each other).

Passive Components

Though there are many ways to achieve success soldering small SMD components, the best method we have found is as follows:

1. Find the pad you intend to populate next and arrange the board so the solder pads are parallel to the edge of the table in front of you.





2. Dab a small amount of flux on the pad nearest the hand with the iron.
3. Apply heat to the pad and flow a small amount of solder to the pad with your other hand. This is called tinning the pad. It only needs a small amount.
4. Now flux the other pad.
5. Take the component out of the cut-tape packaging (we usually tap it out on a piece of white paper in a well lit location) and carefully pick it up across it's breadth with tweezers making sure to keep it aligned with the pads and any silk-screened orientation indications (flat, right side up with any writing on the top) then place it next to the tinned pad, continuing to hold it in place with tweezers.
6. Apply heat to the tinned pad, reflowing the solder, and slide the component into place on the tinned pad, heating them both for a moment to ensure proper bond. Remove heat and allow to cool before releasing with the tweezers.
7. Check the joint for the proper fillet and alignment and reheat if you need to re-align.
8. Once it looks good, flip the board 180° and heat the other pad and apply just enough solder to form a nice fillet. Allow it to cool and examine before proceeding.

Troubleshooting passive components

You may end up with a problem wherein you need to remove a part placed improperly after affixing both sides, fortunately that is not that difficult to fix. The best method is to flood the joint with extra solder so that it stays molten longer, quickly heat both sides and lightly push the component aside with the iron. Then, cleaning one of the pads with soldering braid before re-attaching it correctly, follow the above instructions from step 5.

The next best way is to lay the iron horizontally along the side of the component attempting to desolder both sides simultaneously, and sweeping it away from the pads with the iron as it melts off. We wouldn't use this method on components with a plastic base, like an electrolytic capacitor, or a toggle button.

Electrolytic Capacitors

Start by placing some flux on the pad. Get the capacitor into position, then melt some extra solder onto the iron's tip. Using the 'wet tack' method described above, reflow the solder onto the pad and pin.

Maintain heat for a moment to ensure good solder flow, but not long enough to melt the shield-shaped plastic base. Ensure the cap maintained alignment and reflow the pin if it needs to be adjusted with the tweezers, before proceeding to rotate the PBC, then add flux and solder the other pin.

In a pinch, you can also treat the pins like the legs of an IC and use a similar technique to get them onto the board (starting by tinning one pad). In practice, this can be tricky to pull off without damaging components, especially on a more-populated board.