ATOM: BUILD GUIDE

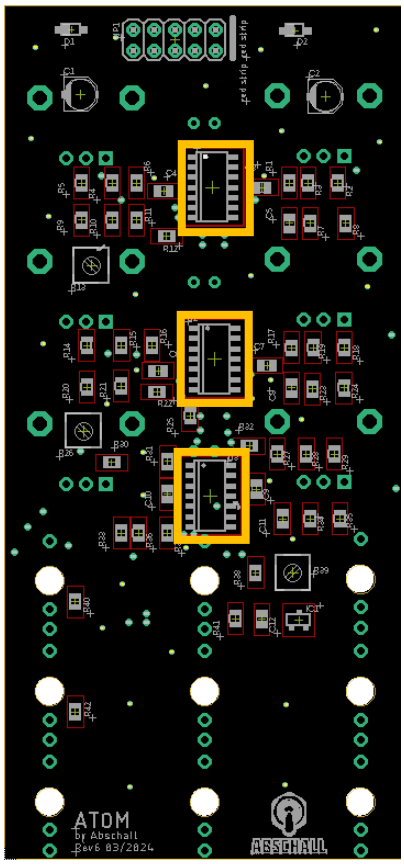
3x ATtenuverter Offset Mixer

Bill of Materials

QTY	VALUE	PACKAGE	NAME
		Resistors	
3	0	R0805	R12, R25, R38
3	330	R0805	R11, R22, R37
1	6k8 / 1k *	R0805	R41
3	9k	R0805	R7, R23, R34
11	10k	R0805	R2, R5, R8, R15, R18, R24, R29, R32, R35, R40, R42
12	100k	R0805	R1, R3, R4, R6, R14, R16, R17, R19, R27, R28, R30, R31
6	1M	R0805	R9, R10, R20, R21, R33, R36
3	2k	RTRIM3314J	R13, R26, R39
*	1k if using 10kB pots, 6.8 k if 100 kB pots (determines the Regulator delivered current), **100kB pots are preferred.**		
		Capacitors	
3	22p	C0805	C5, C8, C11
7	100n	C0805	C3, C4, C6, C7, C9, C10, C12
2	10u	PANASONIC_B	C1, C2
		ICs	
2	1N5819 Schottky	SOD123	D1, D2
1	LM4040A50IDBZR	SOT23	IC1
3	TL074	SO14	U1, U2, U3
		Through Hole Stuff	
1	2x5 Male Pin Header	2x5 Male Pin Header	JP1
6	100kB	ALPHA9MM	LVLA1, LVLB1, LVLA2, LVLA3, LVLB2, LVLB3
3		LED3MM	LED1, LED2, LED3
9	PJ301M	WQP-PJ301M-12_JACK	A1, A2, A3, B1, B2, B3, OUT1
6	Round Knuts M10	Round Knuts M10	
6	Mini Knobs		

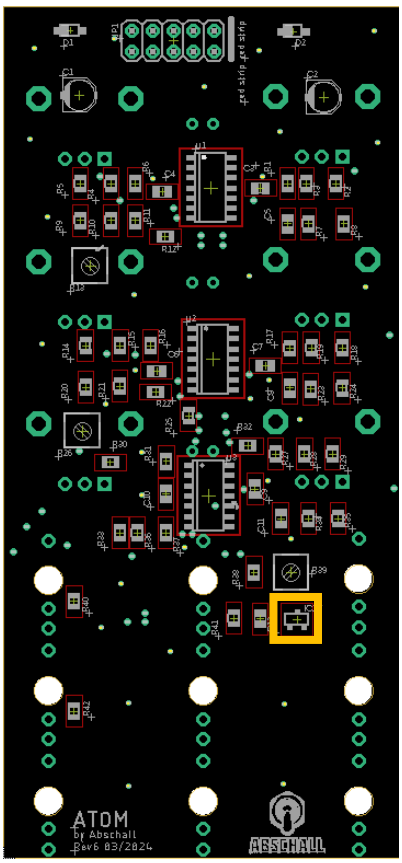
ABSCHALL

ICs

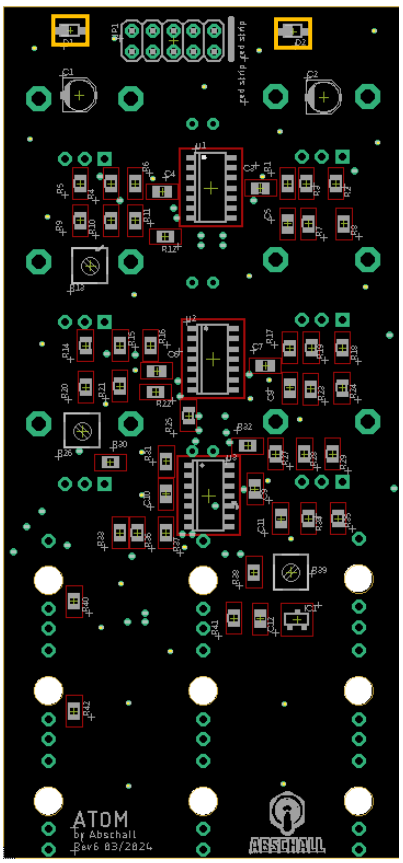


TL074: U1, U2, U3

Make sure that the angled side of the Op Amp is positioned left.



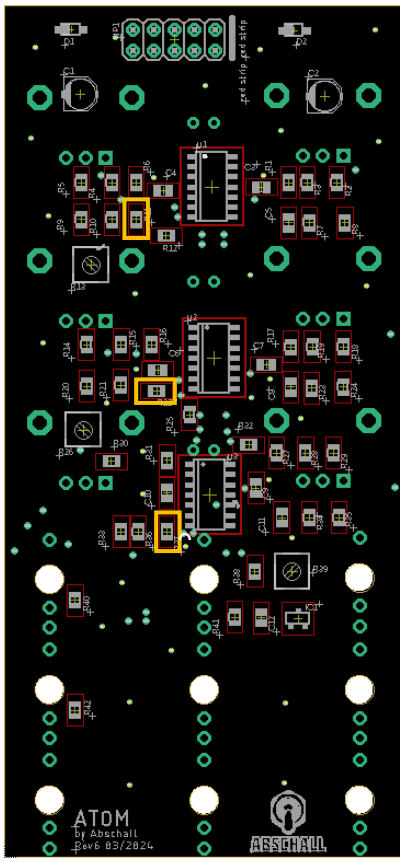
LM4040 5 V regulator: IC1



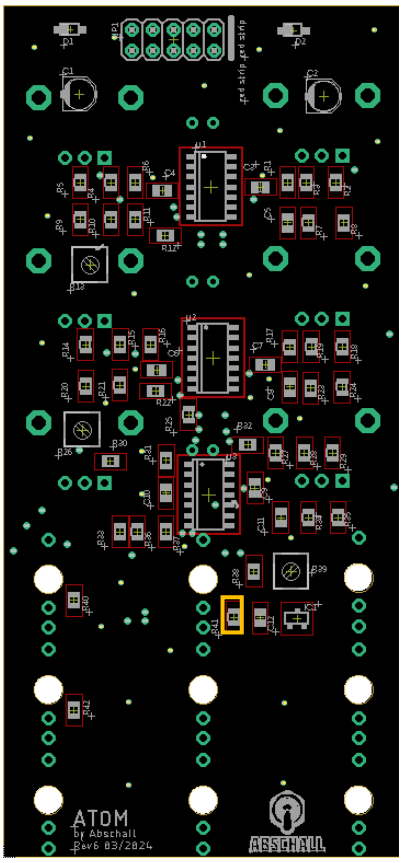
Schottky 5819: D1, D2

The 5819 Schottky diode has a small strip indicating the Cathode side. It corresponds to the white side of the silkscreen

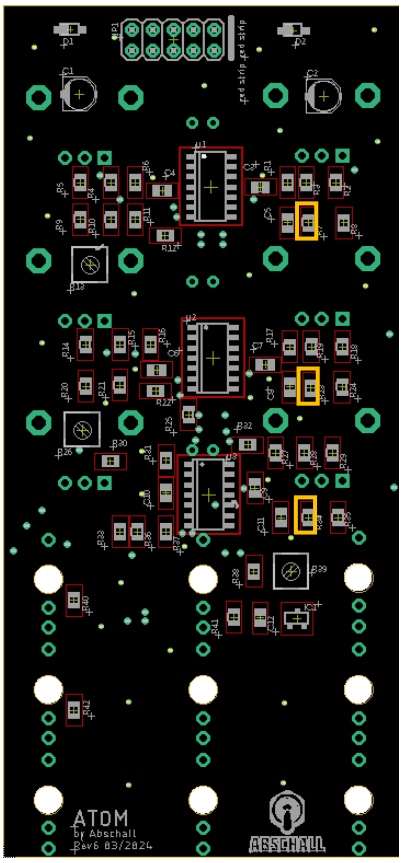
Resistors



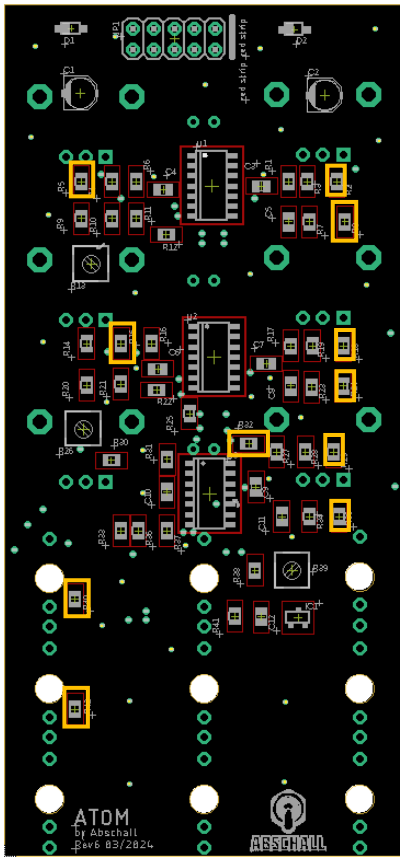
330 Ω: R11, R22, R37



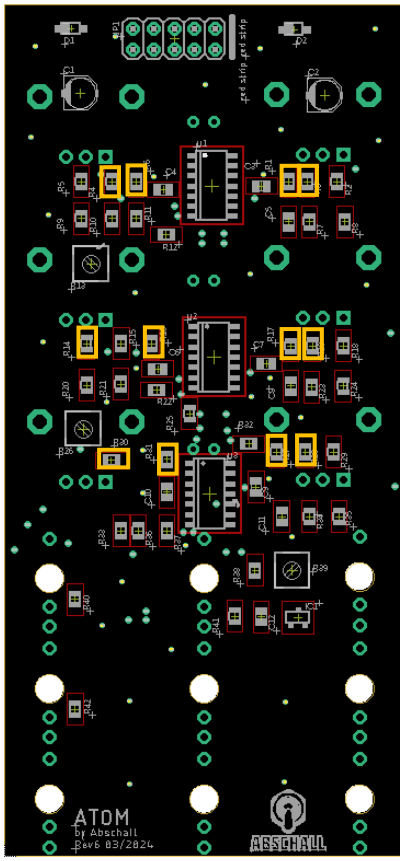
2 kΩ: R41



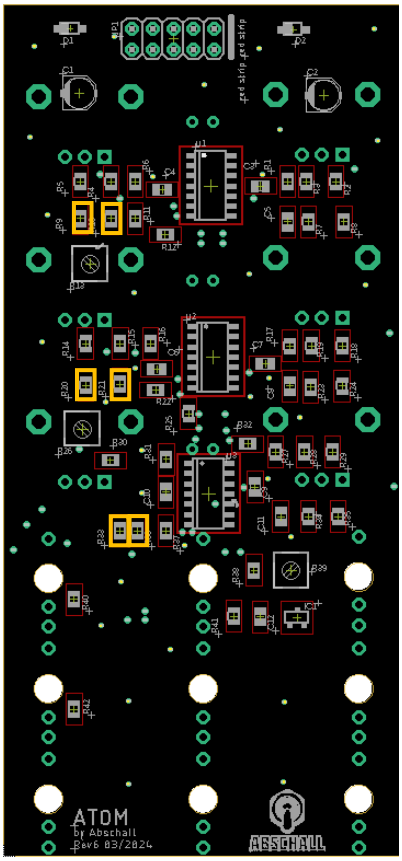
9 kΩ: R7, R23, R34



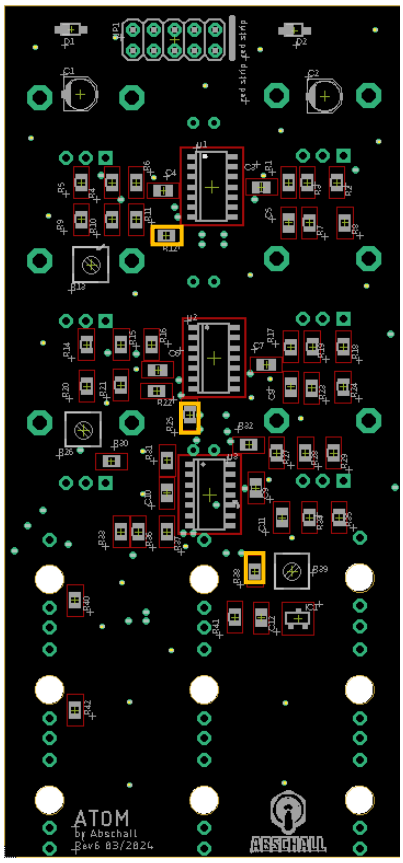
10 kΩ: R2, R5, R8, R15, R18, R24, R29, R32, R35, R40, R42



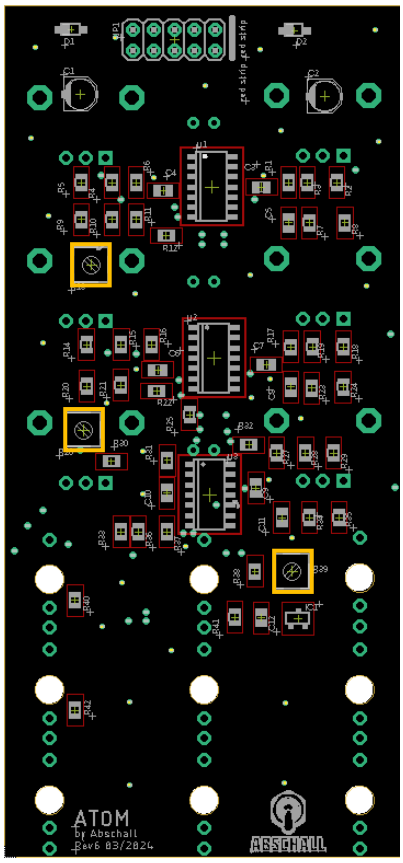
100 kΩ: R1, R3, R4, R6, R14, R16, R17, R19, R27, R28, R30, R31



1 MΩ: R9, R10, R20, R21, R33, R36

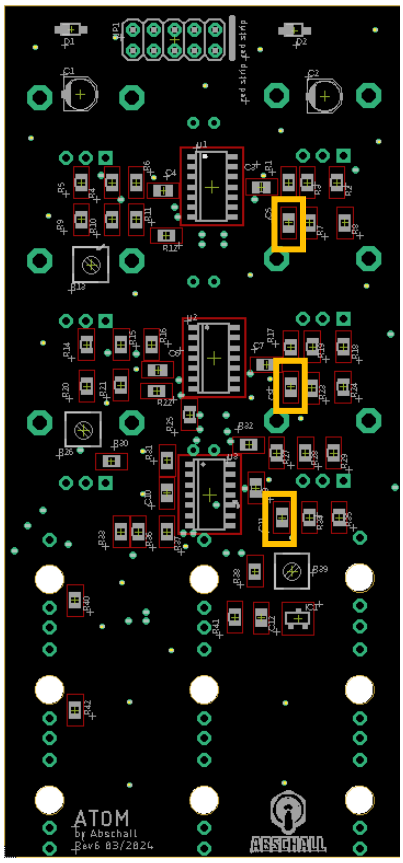


0Ω : R12, R25, R38

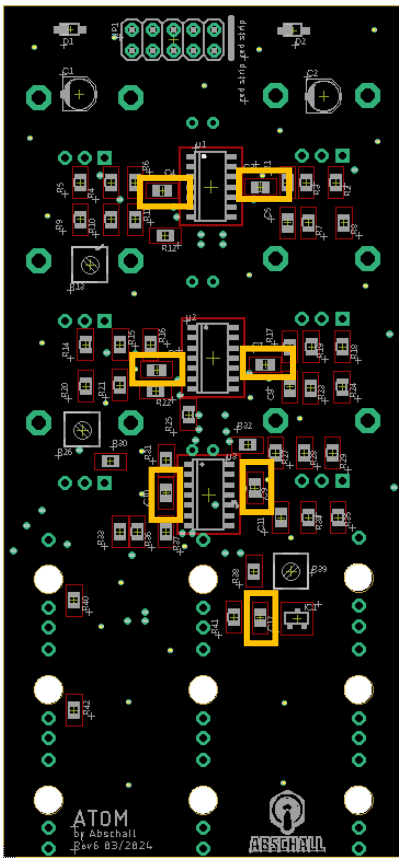


2k Trimmers : R13, R26, R39

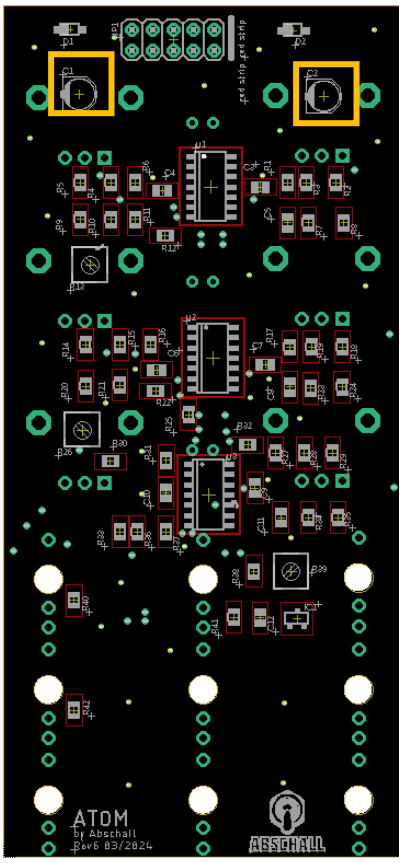
Capacitors



22pF: C5, C8, C11



100nF: C3, C4, C6, C7, C9, C10, C12



22uF: C1, C2 The capacitors have a black strip indicating the minus side.

Through Hole Stuff

1. Insert the 2x5 Male Pin Header through the PCB, turn the board around and solder the header in place.
2. Place the through hole components (LED's, potentiometers and Jacks) on the bottom face, but do not solder anything yet!
3. Place the Front Panel on top of PCB and make sure all the components are aligned properly.
4. Carefully tighten all the nuts of the jacks and potentiometers.
5. Turn the board around and make sure the sockets and potentiometers are pressed against the PCB before soldering the pins.

   DO NOT SOLDER THE LED LEGS YET!

   It is not necessary to solder the potentiometers holding pins (this facilitates potential pot replacement if necessary).

Press the LED until they touch the front panel at a right angle. Then solder one leg of each LED, make sure they are properly aligned against the bare PCB round number spots (the LED lights will shine through the PCB, so it is important that the LED's tops are aligned properly!). Then solder the second legs.

6. Place the knobs on top of T18 potentiometer shafts. Done!

Powering the module up

Before Powering the module up, make sure that there are no shorts between -12 V, GND and + 12 V, using a multimeter's continuity probe. Make sure one last time that the Op Amps, Diodes (D1 and D2) and Electrolytic Capacitors (C1 and C2) are soldered the right way round.

Connect the power ribbon cable: The red strip corresponds to -12 V power rail and must be connected in alignment with the white silkscreen line noted "red strip" of the PCB. If you have done a good soldering job, you should not hear a popping sound and see smoke coming from the components when powering the module…

Calibration

Both inputs of an ATOM channel are normalized to a precise 5V DC reference if no patch cable is plugged.

1. Leave both inputs A and B unconnected. Channel 1 is in offset mode.
2. Connect a cable in Channel 1's OUT Jack.
3. Using alligator clips, connect a voltmeter to the cable's unconnected male jack.
4. Set a Voltmeter capable of measuring at least 10V DC precisely, to DC read mode.
5. Turn both Bipolar Control Knobs of inputs A and B fully clockwise. Expect to observe a voltage around 10 V DC on the voltmeter, as both 5 V sources are combined.
6. Flip the board to locate the topmost trimmer (R13). Adjust the trimmer's 2k resistor using a small screwdriver until you read an exact +10.00 VDC on the voltmeter.
7. Proceed to the calibration of Channels 2 and 3, by **repeating step 1. To 6.** The trimmer of Channel 2 is R26 (middle) and the one of Channel 3 is R39 (bottommost trimmer).

It's crucial to note that if the output of Channel 1 remains unpatched, it will add to the sum of Channel 2. **To prevent this, insert a jack into the output of Channel 1 to break the internal connection.** The same procedure should be followed for Channel 2 when calibrating Channel 3.

Nice! You just successfully finished the build of ATOM!

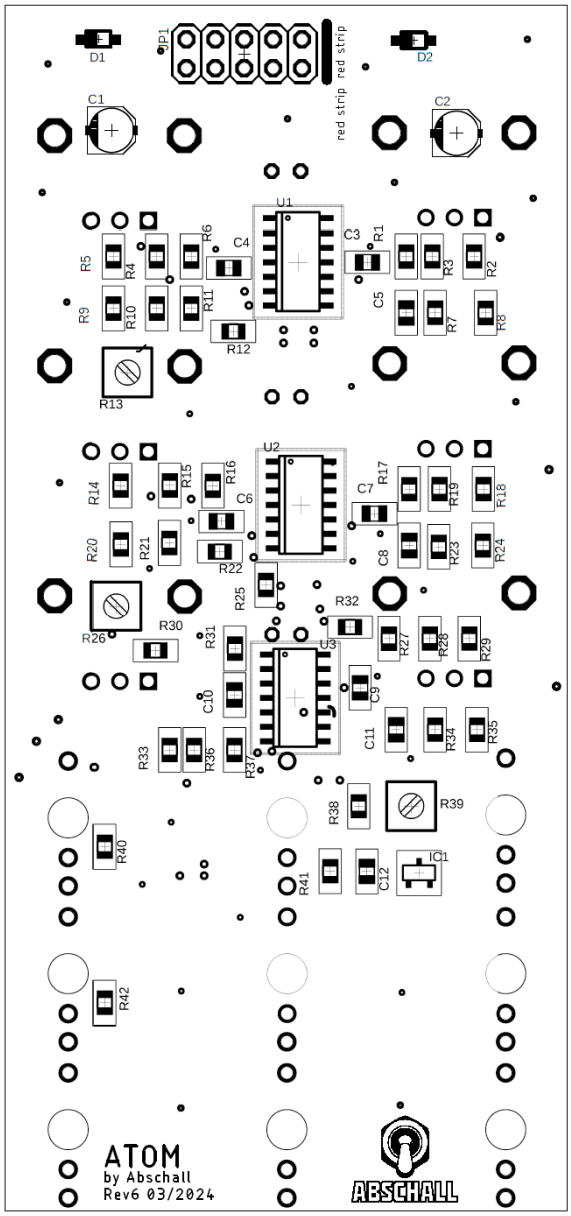


Figure 1 TOP Side Component Placement

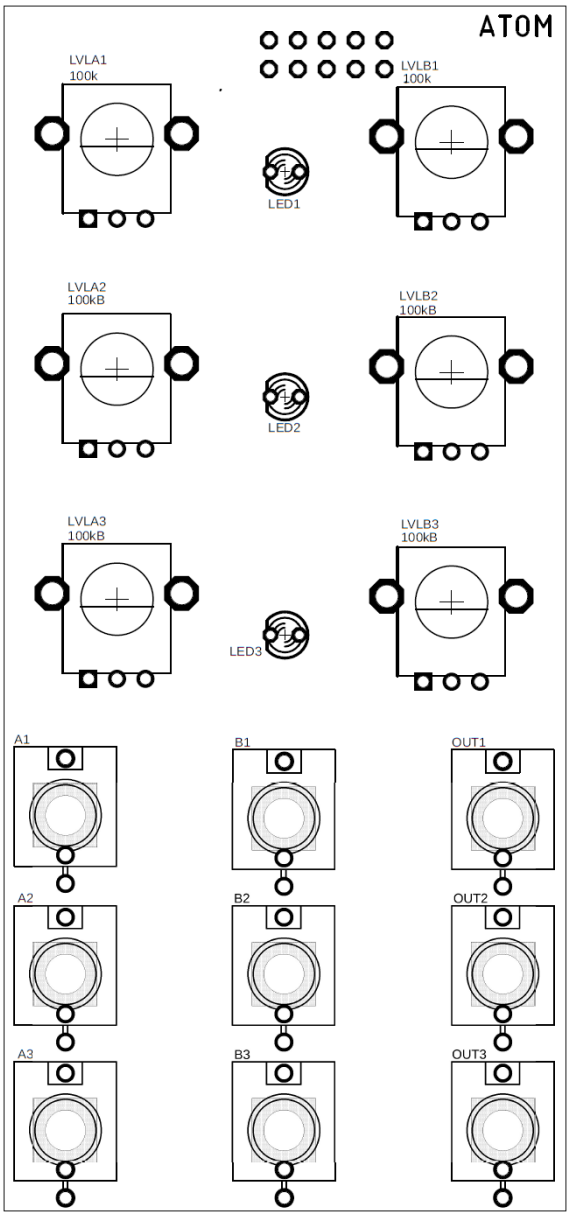


Figure 2 BOTTOM Side Component Placement