



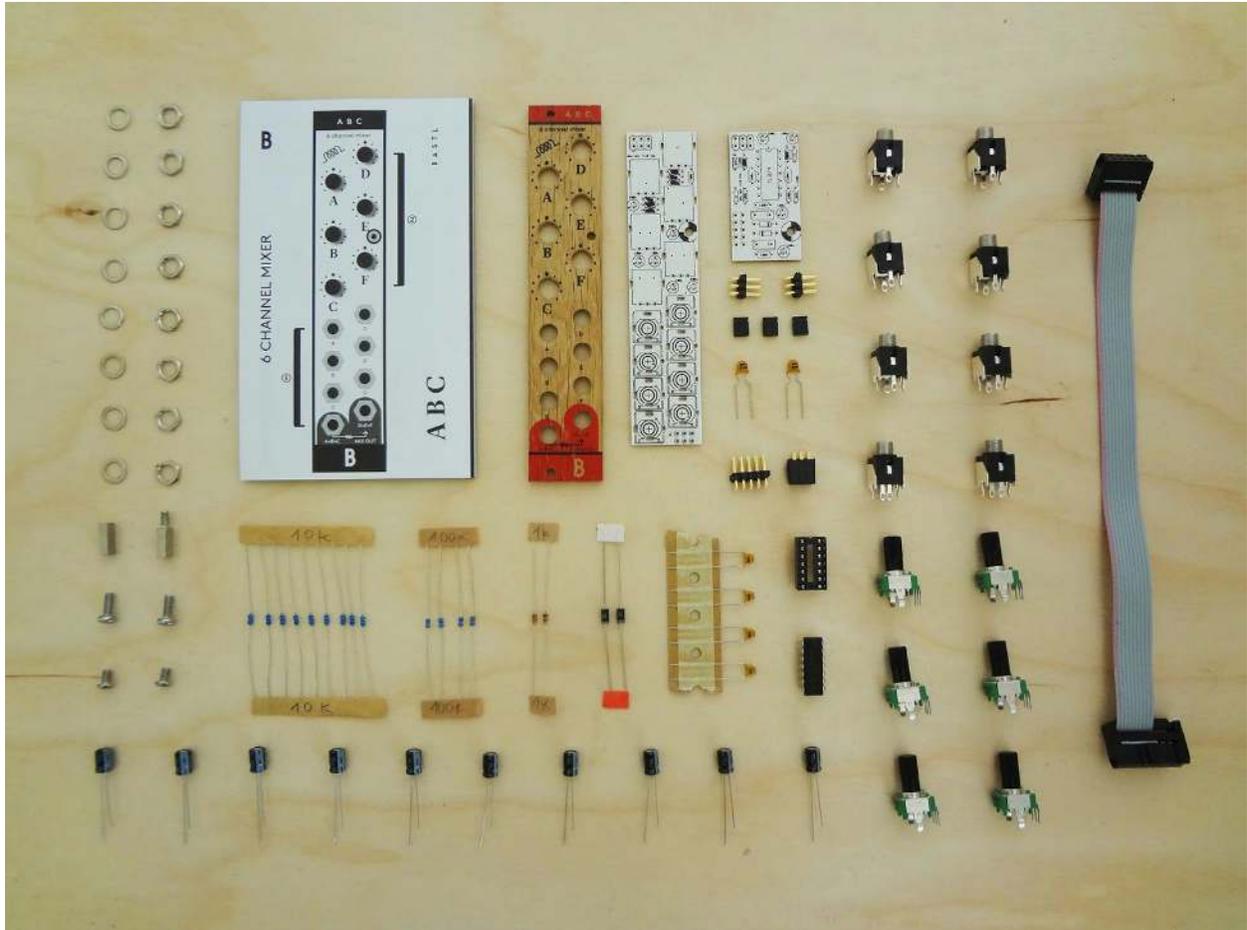
ABC V1.0 ASSEMBLY

Before starting this kit, prepare the following tools: Soldering iron (15-20W will do), flush cutters, no.2 hex screwdriver or allen key and phillips screwdriver. Also briefly go through this guide and make sure that you understand all the steps, if you are having any troubles don't hesitate to seek help at the forum. We suggest that you work in a clean and a well lit environment to avoid accidents or losing any of the small components.

IMPORTANT!

If you have never soldered before, check out this great [tutorial first](#).

And please check that your boards are the same version as this guide and that your kit contains the following items:



BOM – BILL OF MATERIALS

4 x 100k resistor	2 x 1N4007 diodes	1 x nut - nut spacer
9 x 10k resistor	6 x 100k log pots	1 x nut - screw spacer
2 x 1k resistor	1 x 2x11pin male pinheader	8 x jack washers
4 x 100nF capacitors	1 x 2x3 pin female pinheader	8 x jack nuts
10 x 10uF capacitors	2 x 100mA fuse	2 x 6mm screws
1 x TL74	1 x 14 pin DIL socket	2 x 8mm panel screws
8 x jack connectors	1 x ribbon cable 2x8	2 x PCBs
1 x front panel	3 x jumpers	

The ABC module consists of two boards, the top board is used for all of the UI components and the bottom board is the heart of the sound processing circuit.

BOTTOM BOARD

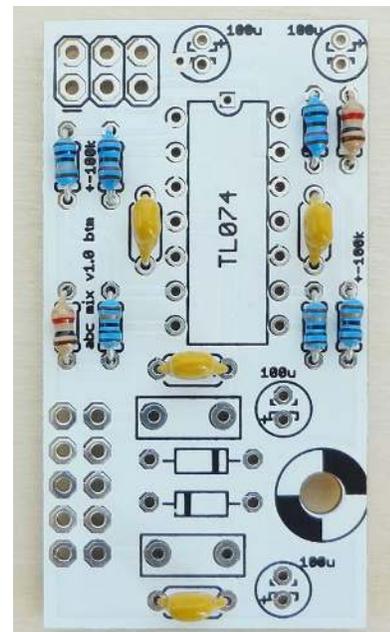
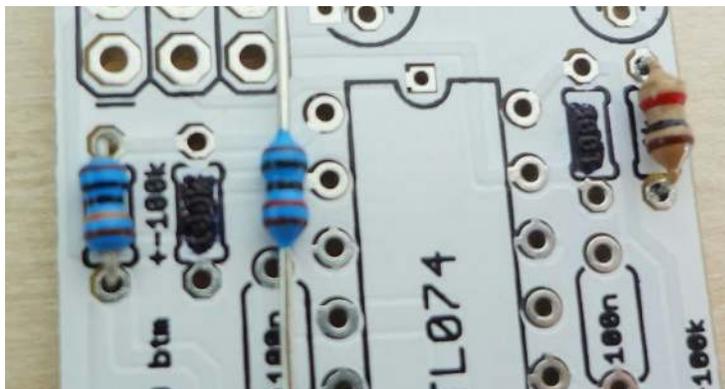
Lets start with the bottom board, with the shortest and smallest parts.

Take a strip of resistors and look up the values printed on the circuit boards. Start with the 100K, 10K and 1K. Place them through the board, solder them and clip off the excess leads.

ATTENTION!

Wrong label on the board. The crossed out 100K resistors on both boards are replaced with 10K. Your board should look like this (click on the images to enlarge):

Then add the capacitors, there are four 100n capacitors (marked 104). They might be in ceramic or polyester film package, don't worry they are not polarized.



Next place and solder the chip socket. Make sure that the notch is in the same direction as printed on the circuit board.

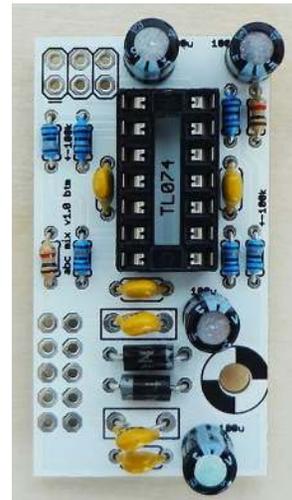
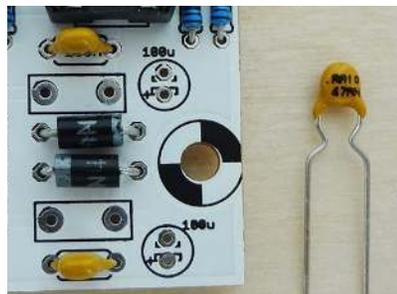
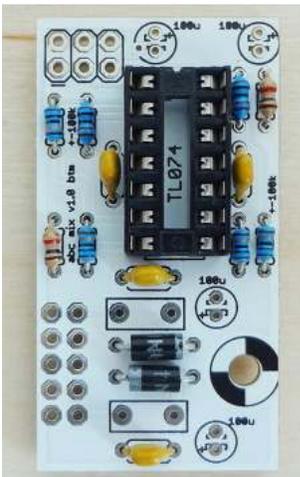
After that populate the board with the diodes and solder them in. Be careful though, diodes are **polarized!** Make sure that the marking ring on the diode body matches the marking on the circuit board. By now your board should look like this:

Move on to the protective fuses. They look quite similar to ceramic capacitors and are placed in the blank rectangular markings on the board besides the diodes.

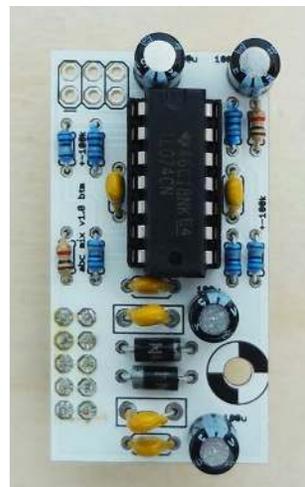
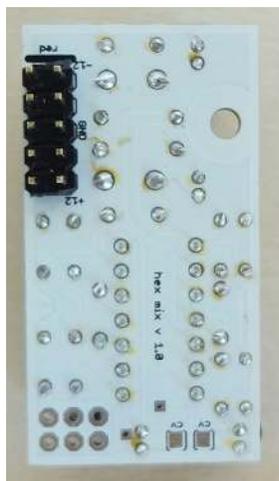
Now, it's time for more capacitors. Take the four 10uF electrolytic capacitors and place them on the board. Be careful, these are **polarized!** capacitors and should be soldered the **right way in**. There is a + marking on the circuit board that should match the long lead of the capacitor, the – side is also marked on the body of the capacitor with a white strip.

ATTENTION!

Wrong label on the board. All the 100uF capacitors on both boards are replaced with 10uF. Your board should look like this so far:



Take the 16 pin power connector and place it also on the back side of the board. It might be tricky to solder it straight, but you can place something like your cutter under the board to hold it level. Also first solder in just one of the pins, then take the board in your hand and re heat that pin while pressing down on the header to align it (be careful though, you don't want to touch the pin you are heating up) wait for it to cool and solder the rest of the pins.



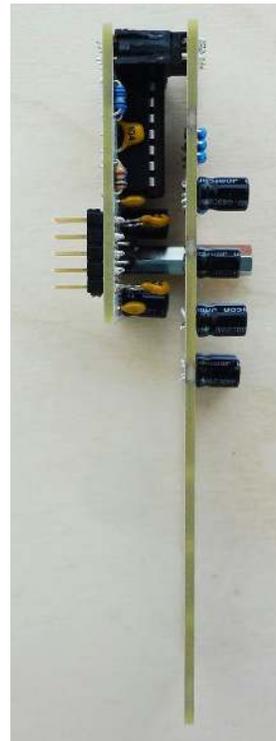
SUPER MEGA IMPORTANT!!! RESPECT CHIP POLARITY.

It's time to insert the IC chip into its socket. Again make sure that the notch on the TL074 chip is **facing the same direction** as the notch on the socket.

TOP BOARD

Now populate the top. Again start with the remaining resistors and solder them in. Like before all the crossed out resistors should be 10k. Next solder the remaining electrolytic capacitors. And like before, the values should be 10uF. Don't forget about their polarity.

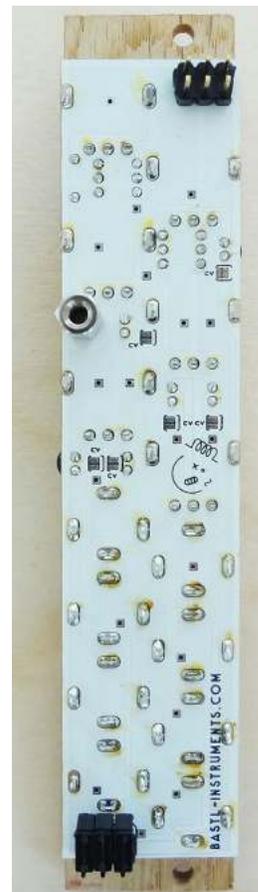
Next ensure that the headers are properly aligned, screw the hex screw and the standoff on bottom board. Place the female headers on button board with the male pins inserted. Now place top board, screw with the standoff screw and finally, solder the headers to both boards. Just like in the image below.



Solder also the small 3x2 jumper header at the back of the top board. And disconnect the two boards again.

Continue with the rest of the UI components, Place the four potentiometers to their respected places on the board. Push them well until they **sit absolutely flat on the board, but don't solder them yet.** Next, place the mono jacks on the board.

Again we want to make sure that all the components are properly aligned with the front panel, so take the standoff and place it in the opening. Check that all the components came through and then screw the wooden front panel with the second hex screw. Also secure the jacks to the panel with the washers and the nuts.



IMPORTANT!

Don't tighten the screws and jack washers too much as you may damage them!

Make sure that all the components are aligned and positioned well and then solder them.

Congratulations! You have made it through, now connect the second board again, place the jumpers in the position that you like and you are ready to enjoy your new module.

Before you connect anything, make sure that your system is disconnected from power. Also double check the polarity of the ribbon cable, the red cable should match the -12V rail both on the module and on the bus board!

TROUBLESHOOTING

did you plug it in?