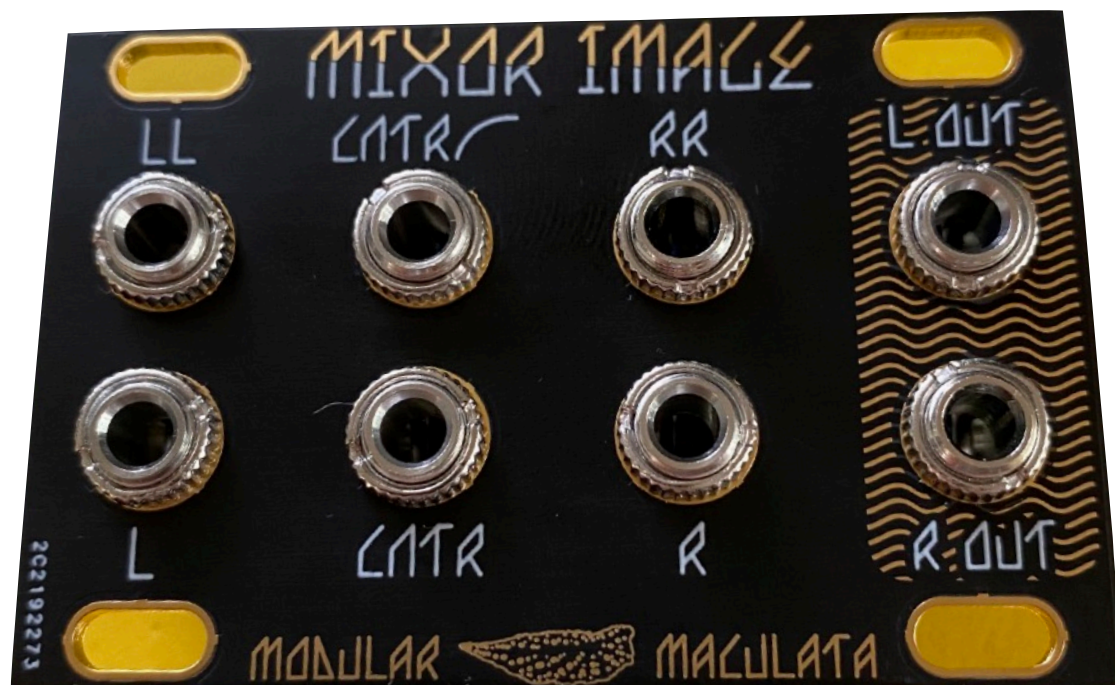# MODULAR MACULATA
MIXOR IMAGE — BUILD GUIDE

Please be Green and read me on your Screen

PREAMBLE

Thanks for supporting Modular Maculata!

This Build Guide is specifically for the **Mixor Image** module and assumes that you are beginning with all parts required and some basic DIY experience.

This guide will not teach you how to solder if you haven't done it before. There are plenty of excellent tutorials online to check out if this is new to you.
That said, while the layout of the boards for this module are quite tight, getting a working module shouldn't be too difficult even for beginner DIY'ers.

Have a read through this build guide before beginning and then simply follow it step by step and you should finish up with a working stereo panning mixer module!

This module is built upon the bones of the 'Nearness' minimal panning mixer designed by Jesper Särnesjö. Nearness is a product of the *////////* community.

The changes that make the Mixor Image a unique variation of the idea behind 'Nearness' are as follows:

- This module has been re-designed to suit the Intellijel 1U format behind a 12hp panel, and be made with entirely through-hole components

- Resistor values have been changed throughout to give a different stereo image with constant power panning inputs set at -35°, -15°, 0°, 15°, 35°

- An additional centre input with a 150hz high pass filter has been added so as not to muddy the centre of the stereo field

- Stereo in & out *link* connectors have been added allowing for chaining of multiple modules or linking to other manufacturers' stereo modules without patch cables

In accordance with the wishes of the designer of the work that inspired this, and in the spirit of community development, this work is licensed under a Creative Commons Attribution-ShareAlike 4.0 International License.

A GitHub repository for this module and associated resources is maintained at:
https://github.com/modular-maculata/mixor-image

BOM

QUAN.	VALUE	BOARD REFERENCE(S)	PART SIZE/DIMENSIONS
1	180K	R1	1/4W 1% Metal Film Resistor
2	102K	R2, R13	1/4W 1% Metal Film Resistor
2	620K	R3, R12	1/4W 1% Metal Film Resistor
2	113K	R4, R11	1/4W 1% Metal Film Resistor
2	220K	R5, R10	1/4W 1% Metal Film Resistor
4	110K	R6, R7, R8, R9	1/4W 1% Metal Film Resistor
8	100K	R14, R15, R16, R17, R18, R19, R20, R21	1/4W 1% Metal Film Resistor
2	1K	R22, R23	1/4W 1% Metal Film Resistor
1	5.6nF	C1	Any Small (=/<4mm lead pitch), Non-Polarised Capacitor **Min. Rating 35V**
2	22pF	C2, C3	Any Small (=/<3mm lead pitch), Non-Polarised Capacitor **Min. Rating 35V**
2	22uF	C4, C5	5x11mm Radial Electrolytic Capacitor (2mm lead pitch) **Min. Rating 35V**
4	100nF	C6, C7, C8, C9	Any Small (2.54mm lead pitch), Non-Polarised Capacitor **Min. Rating 35V**
2	1N5819	D1, D2	DO-41 1A 40V Schottky Diode
2	TL072	U1, U2	8-Pin DIP
2	IC Socket	U1, U2	8-Pin DIP Socket
8	Jack Sockets	CEN1, CEN2, LIN1, LIN2, LO1, RIN1, RIN2, RO1	PJ398SM "THONKICONN" 3.5mm Jack Sockets
2	Pin Header	J1, J3	1x3 2.54mm Pin Header
2	Pin Socket	J2, J4	1x3 2.54mm Pin Socket
1	Power Header	PWR1	2.54mm 10-Pin (2x5) Boxed Header Connector (Standard Eurorack boxed power header)
2	Pin Header	STIN1, STOUT1	2.54mm 3-Pin Molex Wafer Connector

ASSEMBLY

Begin by gently separating the two boards.

It may help to score along the tiny holes on each side of the board with a craft knife before attempting to snap them apart.

If necessary, sand or file any excess PCB material away so you end up with two separate boards as shown in the pictures below:
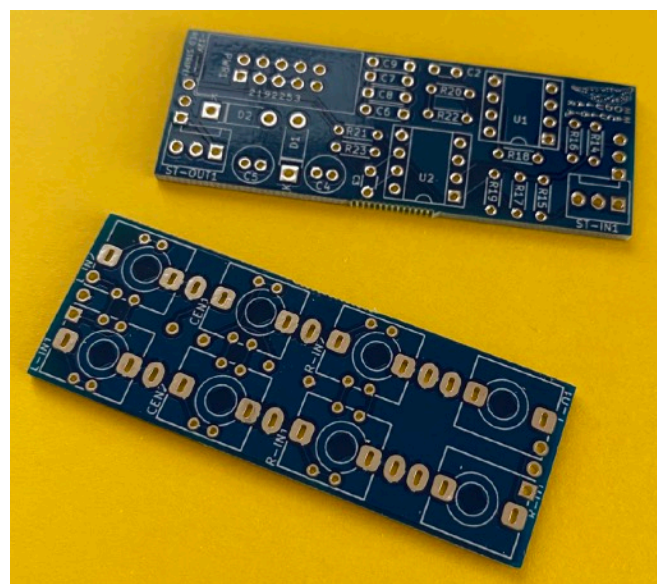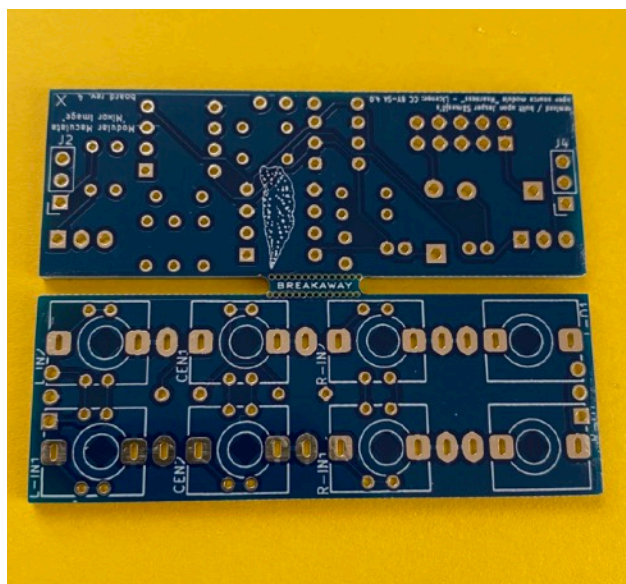
Next, place the appropriate resistor values for R1 through to R13 on the jack board, **and solder them.**
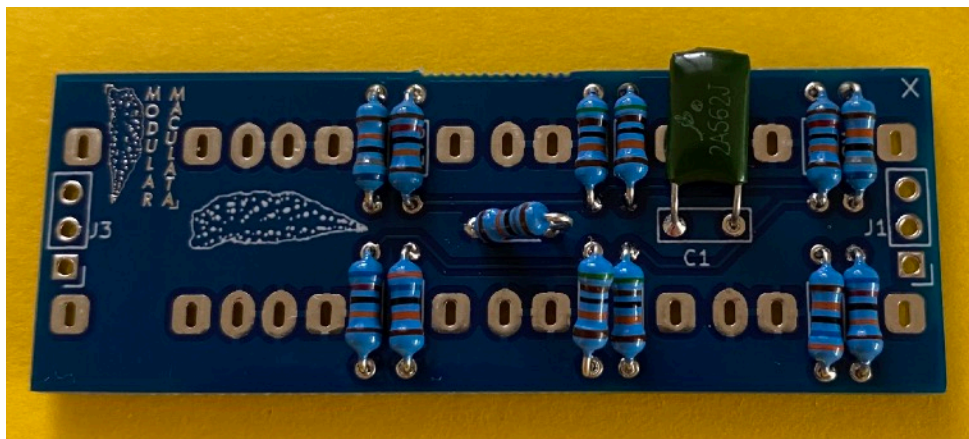
Depending on the size of the resistors you're using, they may have to sit a little angled to fit the lead pitch as you see in the photo below:

31/03/2022
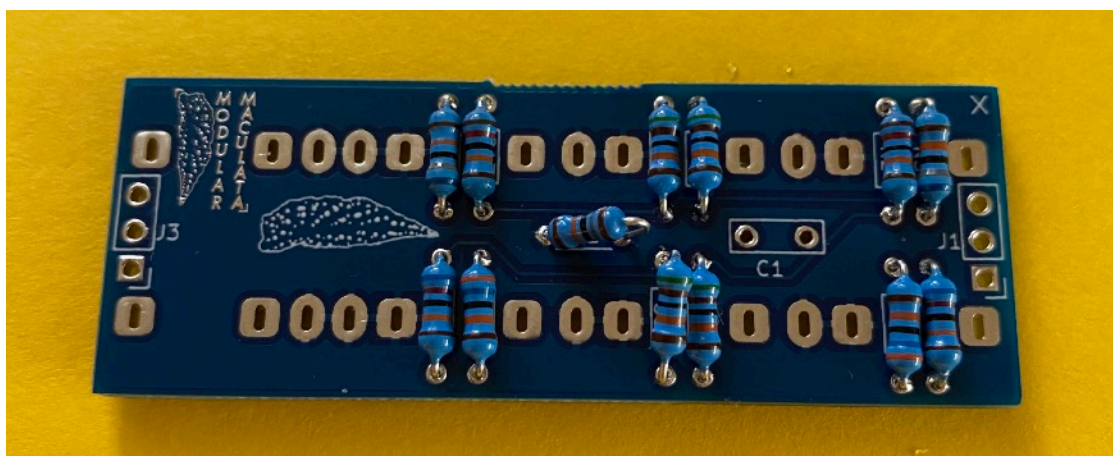
Once those resistors are all soldered in, install capacitor C1.
It will help later if you leave enough lead length on the capacitor so that it can be easily bent over and lie close to flat if needed.


Once you have the resistors in place, prepare the 2.54mm Pin Headers so that you have **two that are each 3 pins long.**

Depending on how your Pin Headers came, you may have to cut these lengths out of longer rows. This is easily done with a pair of side cutters.

Once you have the right lengths ready to go, place and solder them one by one in the appropriate positions for J1 and J3 on the jack board.

Ensure that each header is soldered with the pins as close to straight up as you can get them - This will make joining the two PCB's together much easier.
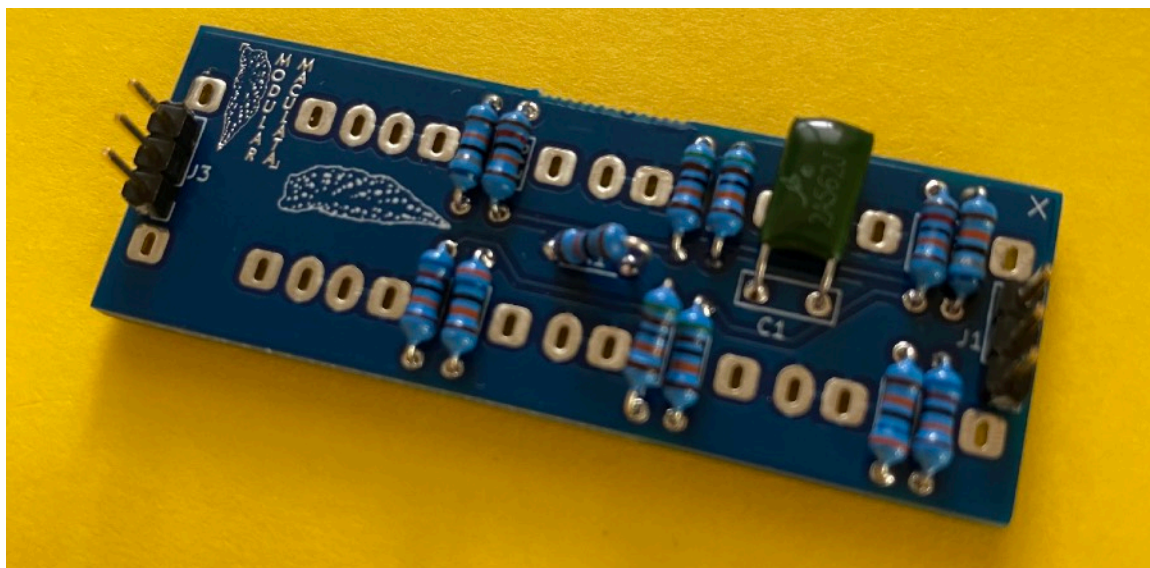
Next, flip the board over and (If you haven't already) cut the excess resistor legs and any significant bumps left behind from soldering the Pin Headers so that the solder joints are not too tall.
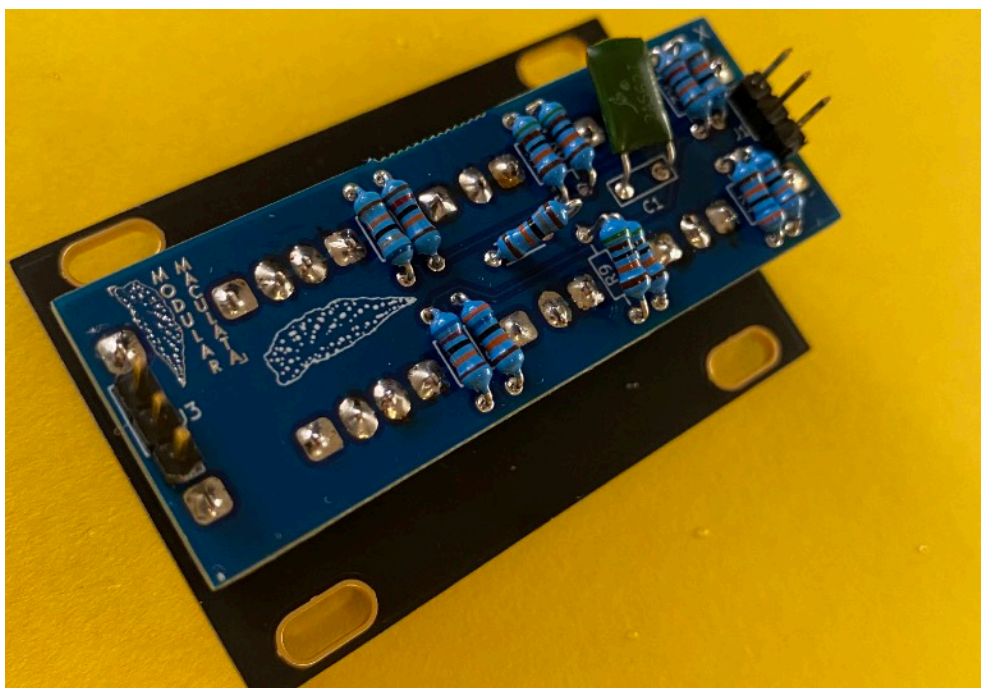
Place all 8 jacks sockets in the board but *DO NOT SOLDER THEM YET!*
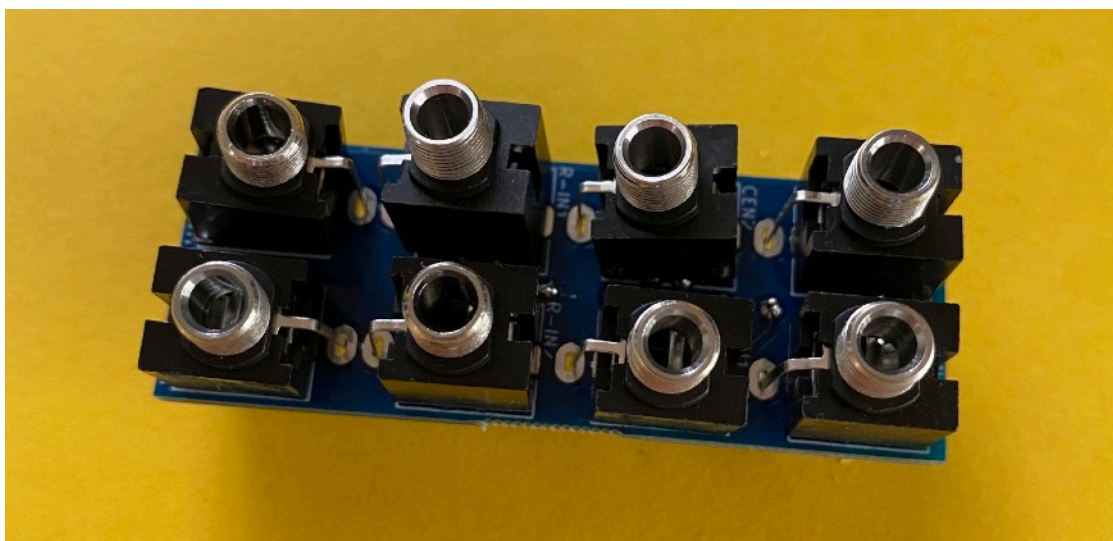
With all the jack sockets in place, mount the front panel and use the socket nuts to hold the panel and the board together.
Only tighten the nuts gently - **They do not need to be very tight yet!**

Now, f*lip the panel face down and solder all the jack sockets in place.*


You can now loosen the nuts, remove the panel, and make sure nothing has moved in the process. If the panel fits back on easily, **Well Done!**

If the panel doesn't fit back on quite right, see if you can identify any obstructions. You may need to re-solder and tweak the alignment of one or more of the jack sockets.

**WE CAN NOW PUT THE JACK BOARD AND PANEL TO ONE SIDE
AND FOCUS ON FINISHING THE BOTTOM BOARD.**

Start here by placing and soldering the **eight 100K resistors in place for R14 through to R21**. Cutting the excess leg off after checking the solder joint will help make the work easier as we go on.
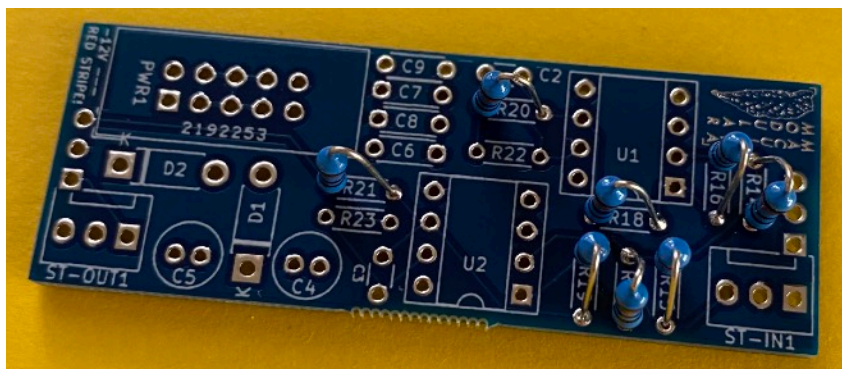
Once all the 100K resistors are soldered in place, we can continue and do the same for the **two 1K resistors in place for R22 and R23**.
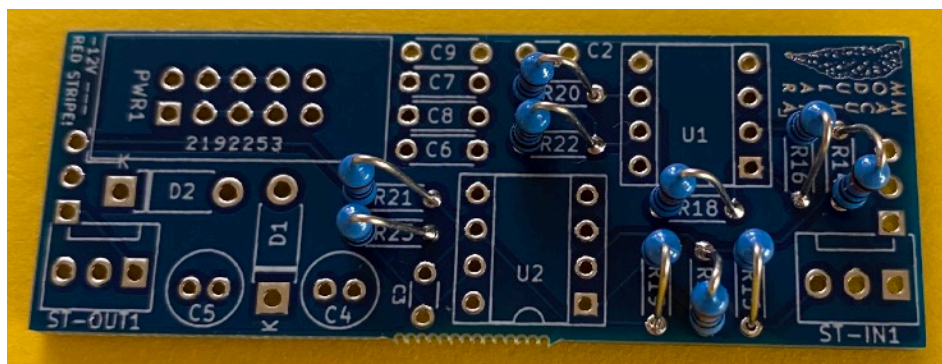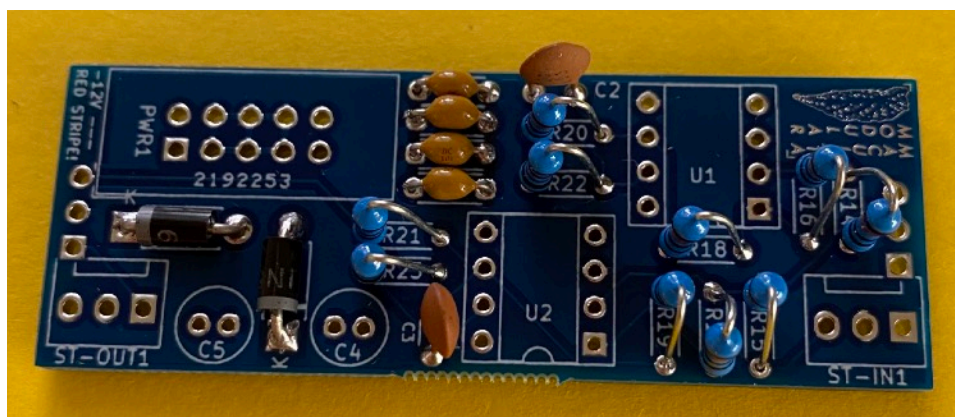

With all the resistors done, we can place and solder both D1 and D2.
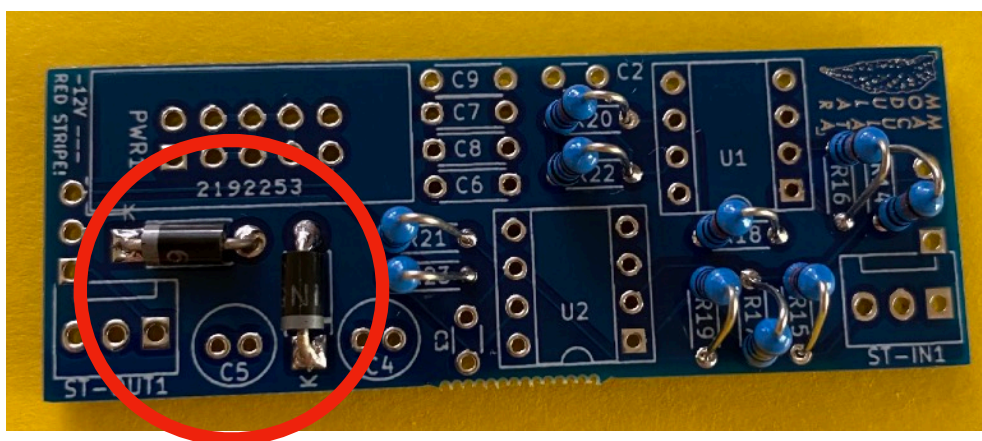
*Pay close attention to orientation of the diodes - If they go in the wrong way, your module **WILL NOT WORK!***
Refer to the photo above for confirmation you've got them round the right away.

Next, we can place and solder all the non-polarised capacitors C2, C3, and C6, C7, C8 and C9 according to the BOM.

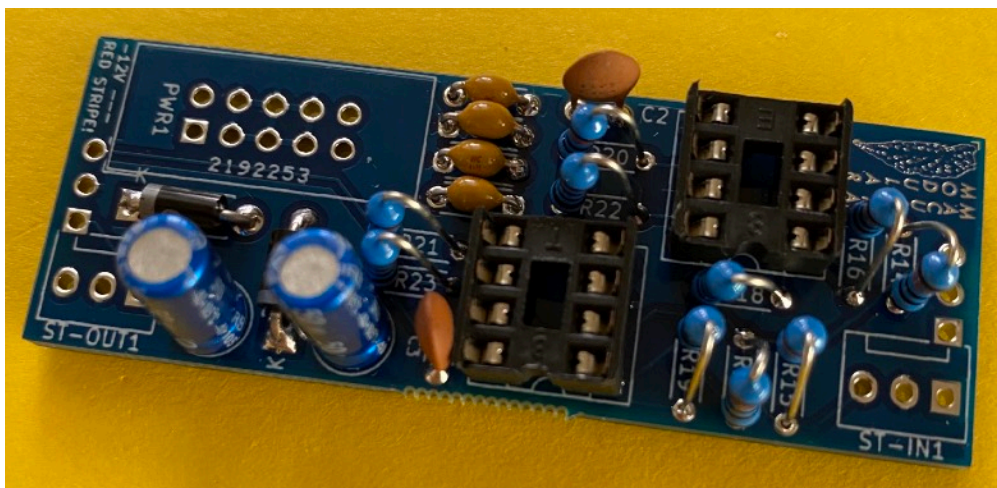Next, place and solder the Electrolytic Capacitors in position for C4 and C5.
Again, **watch the polarity on these!** The stripe down the side of the capacitor denotes the 'Negative' leg. This should face away from the + sign on the board.

***If in doubt, compare your placement to the photo below before soldering:***
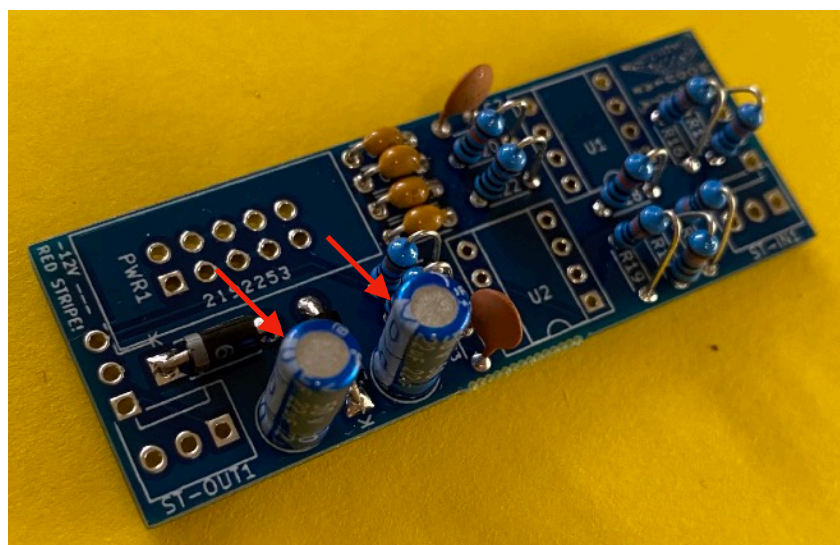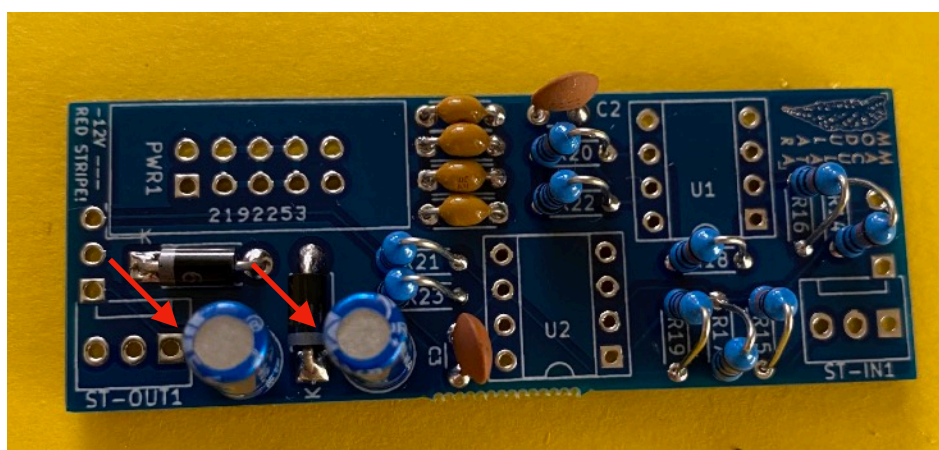
Next, place and solder the two IC Sockets.

…And the Power Header!

This is another part that needs to go round the right way - If it is not positioned as shown in the photo below (With the side opening facing to the centre of the board), You might connect a power ribbon around the wrong way and cook the module!
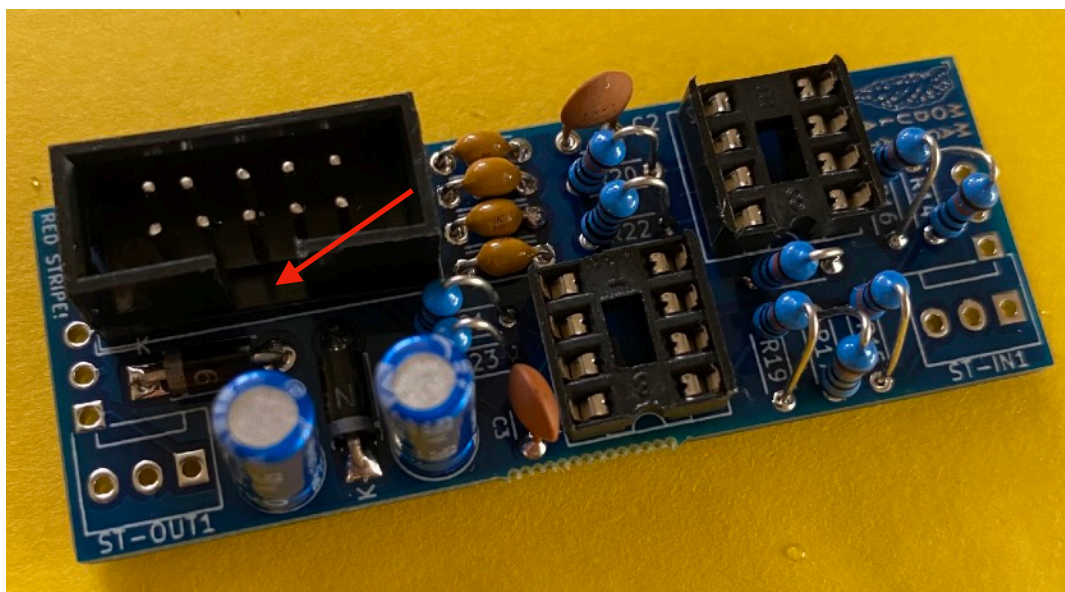
Once you've finished that, prepare the 2.54mm Pin Sockets. We need the same set of these as we did for the Pin Headers on the upper board - **two that are 3 pins long.**
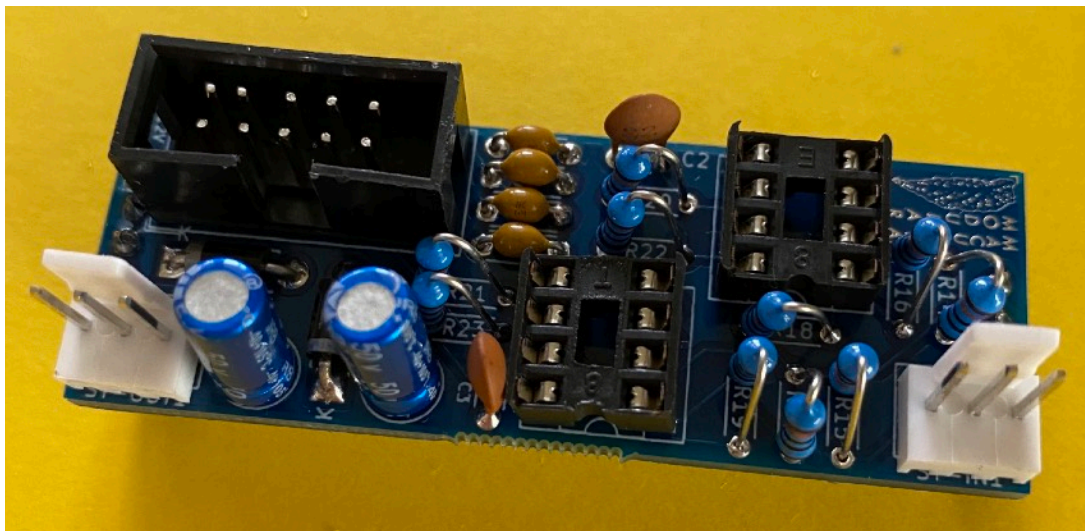
Place and solder these in position for J2 and J4.
Note that these are installed on the opposite side of the board from the rest of the components.
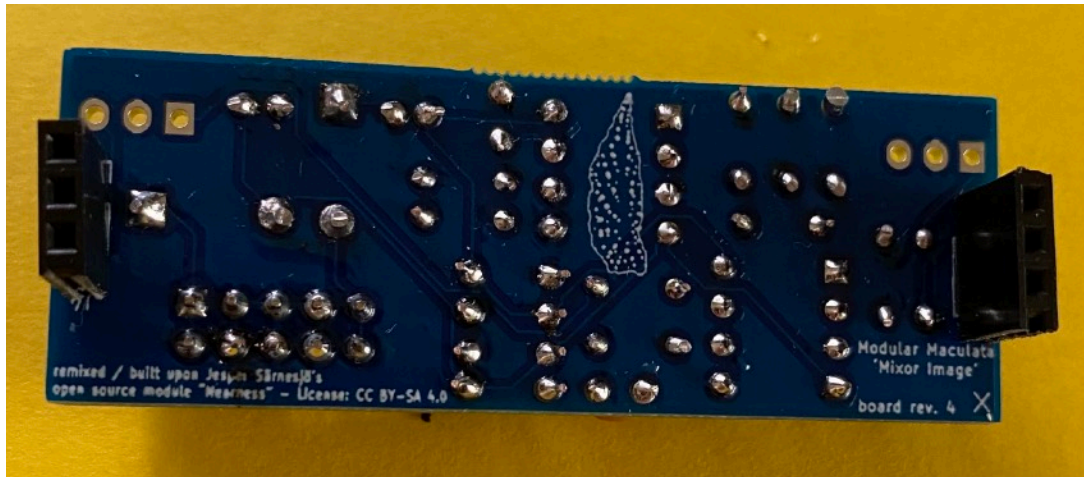
The final parts that need to be soldered are the Stereo Link connectors.

Place the Wafer Connectors so that they match the silkscreen below them. Once soldered in they should look like this:

Before we go any further, have a look over your board for any obvious issues - Bad solder joints, component legs touching each other, any burned or damaged parts, etc.

If your board looks good then you can install the IC's in their sockets, ***making sure that they are oriented correctly as shown on the PCB silkscreen and in the photo below!***
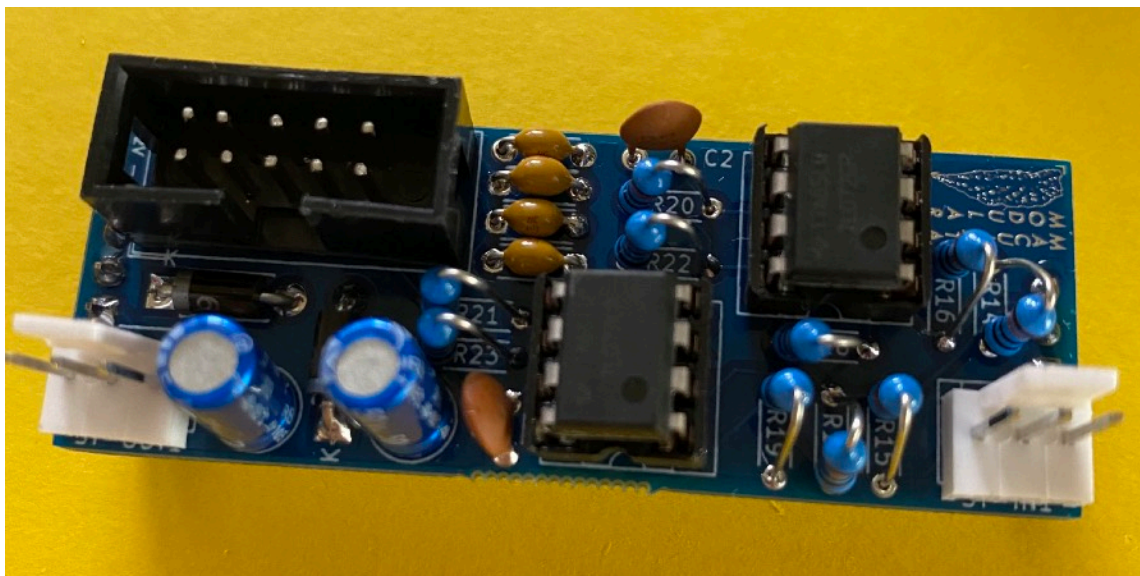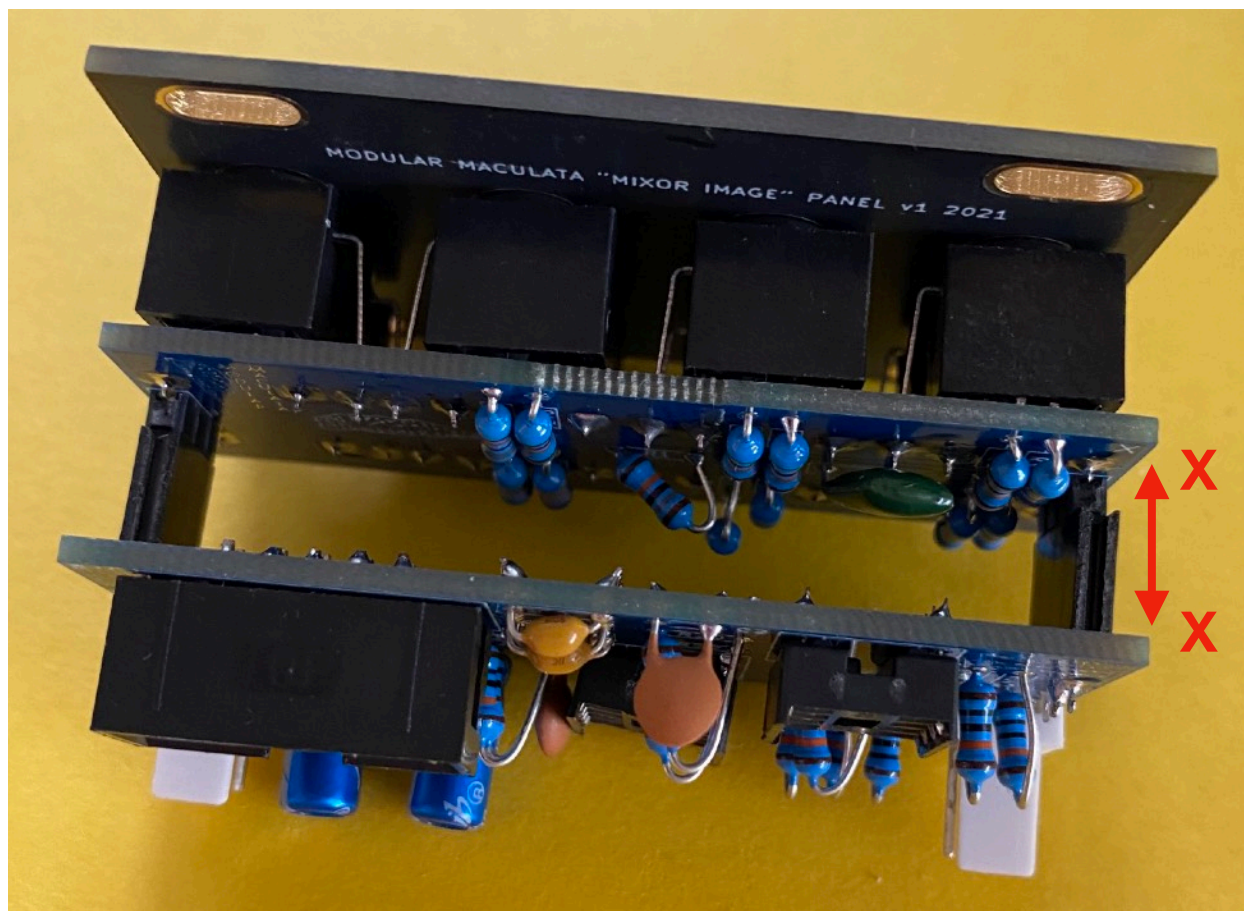

You can now take the top and bottom board and join them together by aligning the Pin Headers and Sockets and ***lining up the two 'X's on each board as shown in photos below!***

Have a good look from different angles to make sure everything fits well and no parts are touching each other that shouldn't be.
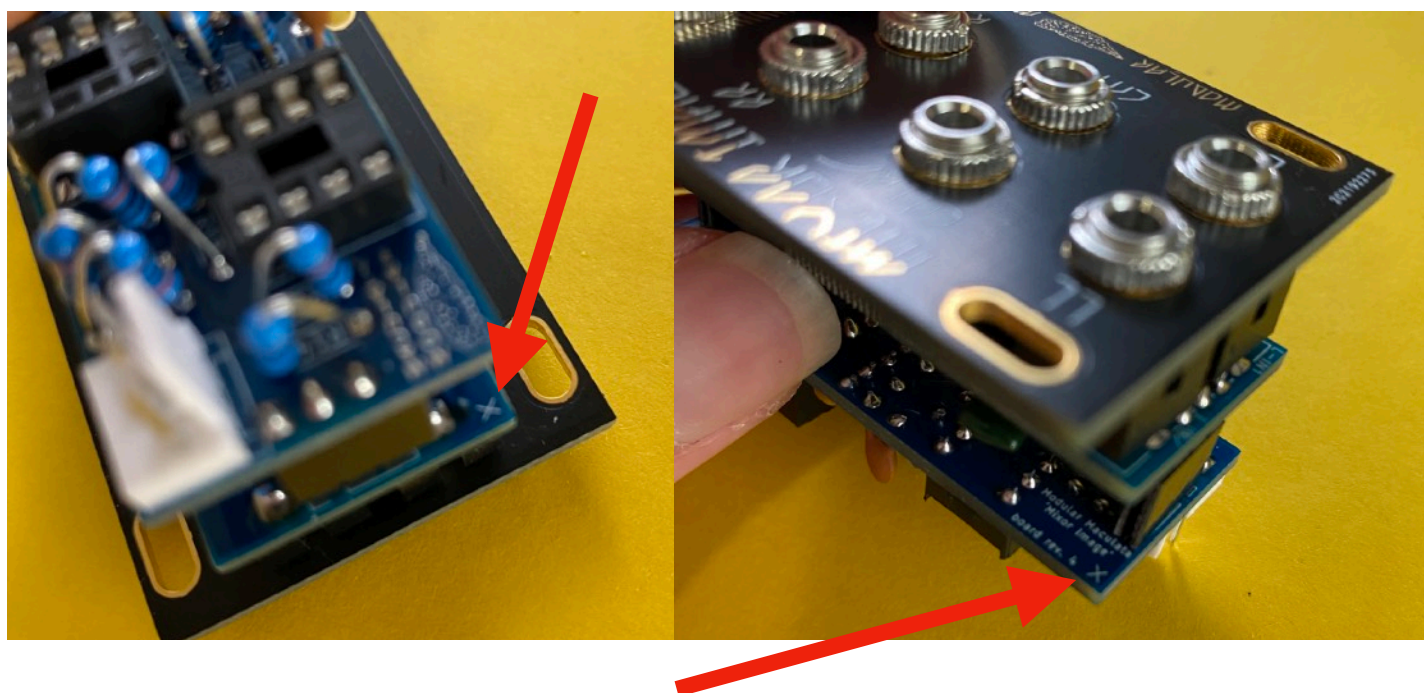
If everything looks good then **Well Done!**

Before powering the module up for the first time, it is a good idea to test for continuity between the +12V, GND, and -12V pins on the power header. **If there is a short circuit between any of these this needs to be fixed before powering up the module.**

Unfortunately, Modular Maculata is unable to provide DIY support beyond this build guide.
There are plenty of guides and lots of advice online about troubleshooting and tracing problems with DIY modules.
If you have an issue take a minute to think about it, do some research and then work calmly and methodically to resolve it *BEFORE POWERING UP YOUR MODULE!*

Once everything is looking good to go, pop it in your rack and start it up!

Congratulations on finishing this build and enjoy the Mixor Image module that you built ☺

Feel free to get in touch with us at modular.maculata@gmail.com if you have any questions.

As stated above, unfortunately we cannot provide DIY assistance beyond this guide, but we'll always do our best to help out where we can with general support.