MH BIAS 2

BOM

Designator	Notes	Qty	Value
C5, C6	Electrolytic Capacitor	2	22uF
D1,D2	Diode_THT:D_DO-35_SOD27_P2.54mm_Vertical_KathodeUp	2	1N5818
J1,J3, J4, J5	Connector_Audio:Jack_3.5mm_QingPu_WQP-PJ398SM_Vertical	4	Jack Socket
J2	Connector_IDC:IDC-Header_2x05_P2.54mm	1	Power Connector
R4, R5	Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P7.62mm	2	10R
RV1,RV2, RV3, RV4	Potentiometer_THT:Potentiometer_Alpha_RD901F-40-00D_Single	4	B100k
U1, U2	Package_DIP:DIP-8_W7.62mm	2	TL072
U1, U2	Turned pin headers 1x4 (in place of IC sockets)	4	
	Potentiometer Knobs	4	Knobs
	MH Eurorack BIAS 2 PCB (SMD Pre-soldered)		
	MH Eurorack BIAS 2 front panel		

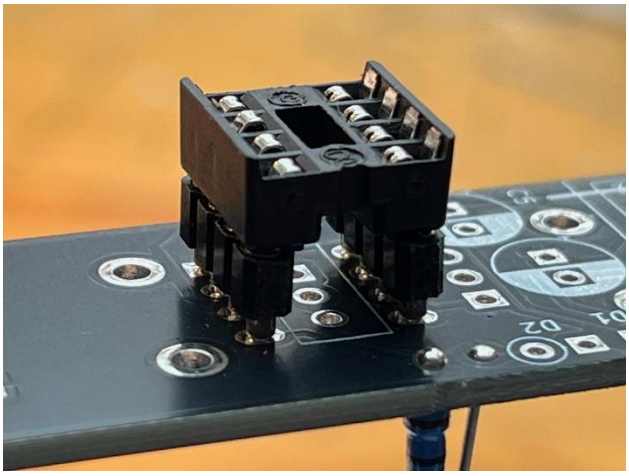
Note: A couple of techniques are implemented in the BIAS 2 design to allow higher component density (further info in the assembly guide).

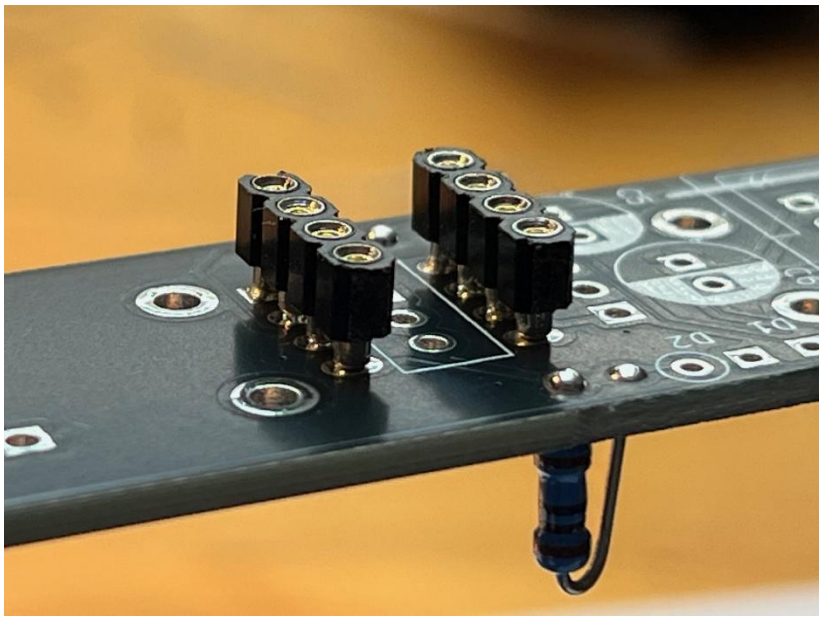
- Pin headers are used in please of IC sockets. This allows soldering of components under the ICs.
- The standoff lugs need to be removed from the potentiometers. This is to allow for components underneath them.

Assembly guide

On the front side of the PCB (the side without the MH logo), place and solder R4 and R5. These resistors and mounted vertically. Place the resistor (either way round) in the circled solder pad and bend the other leg over to go in the other solder pad.

Turning to the other side of the board (the side with the MH logo), place and solder the pin headers in the U1/ U2 positions. You can use an IC socket to makes sure these are soldered straight (See photo)

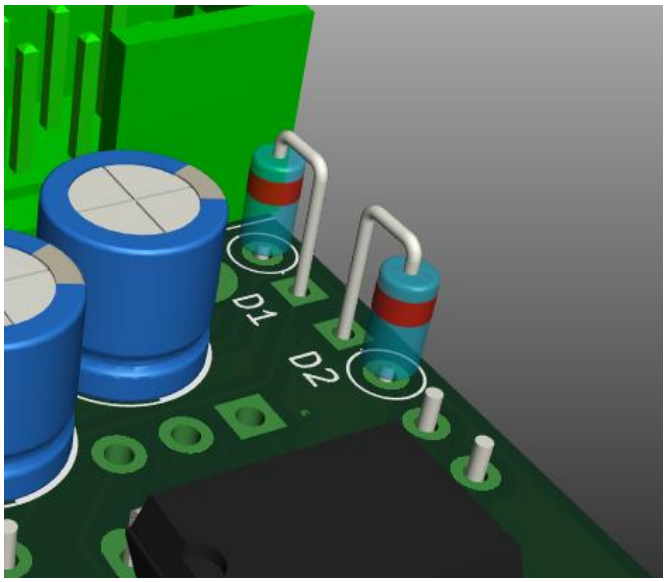




The 2 soldered pin headers form an IC socket

Place and solder the 2 diodes. These are mounted vertically. Take care with orientation – the body of the diode should be soldered into the circled solder pad with the stripe on the diode uppermost (away from the PCB). The other leg needs to be bent over to go in the adjacent square solder pad.

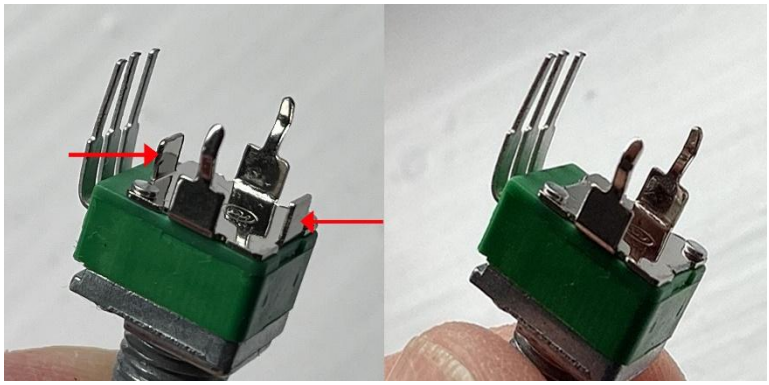
See illustration.



Place and solder the two electrolytic capacitors, C5 and C6. Take care with orientation the long leg of the capacitor must go in the hole marked '+' with the shorter leg going into the white semi-circle . The capacitor will also have a '-' marking on the short leg side.

Place and solder the 10 pin Power connector. If you are using a shrouded header make sure the notch lines up with the notch marking on the PCB.

Remove the stand off lugs from all four potentiometers. This is easily done by gripping the lug with a small pair of pliers and wiggling it back and forth a few times, The lug will come away from the potentiometer.



Place the potentiometers and jack sockets on the front of the PCB (Do not solder yet).

Put the front panel over the potentiometers and jack sockets. Secure with the jack nuts and pot nuts.

Check that the jacks are still aligned to the PCB and have not been rotated by tightening the nuts. If any of them are out of place loosen the nut adjust and re-tighten.

Solder everything making sure there is no gap between the front panel components and the PCB. The two main potentiometer legs next to the power header are particularly tricky to solder. If you don't feel confident in soldering them without damaging other components you can leave these unsoldered.

Insert the ICs in the pin headers making sure the notch on the IC lines up with the notch marking on the PCB. If there is no notch on the IC there should be a dot or circle in one corner. In this case, place the IC so that the dot/circle is nearest the notch on the PCB.

Push the knobs on to the potentiometer spindles.

Done! Connect the power cable, power up and enjoy your new module.