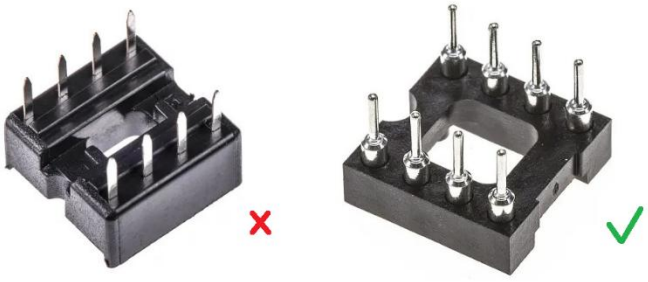
MH AMIX+

BOM

Designator	Notes	Qty	Value
C5,C6	Capacitor_THT:CP_Radial_D6.3mm_P2.50mm	2	22uF
D1,D2	Diode_THT:D_DO-35_SOD27_P2.54mm_Vertical_KathodeUp	2	1N5186
J1,J2,J3,J4,J5,J6	Connector_Audio:Jack_3.5mm_QingPu_WQP-PJ398SM_Vertical_CircularHoles	6	3.5mm Jack socket
J7	Connector_IDC:IDC-Header_2x05_P2.54mm_Vertical	1	Power connector
J8,J9,J10	Connector_PinHeader_2.54mm	4	1x2 male pin header
R1,R2	Resistor_0.25W_Vertical	2	10R
RV1,RV2,RV3	Potentiometer_THT:Potentiometer_Alpha_RD901F-40-00D_Single_Vertical_CircularHoles	3	A50k
	Potentiometer knobs	3	
SW1,SW2,SW3	Sub miniature toggle switch SPDT	3	SPDT
U1	Package_DIP:DIP-8_W7.62mm_LongPads	1	TL072
U1	IC Socket (turned legs)*	1	
	2 pin jumper	4	
	MH Eurorack AMIX+ Front panel & PCB (SMD presoldered)	1	

*The IC socket needs to accommodate potentiometer legs from the other side of the board so use a type which allows clearance and check the fit before soldering:



If you are unable to source a turned pin IC socket you can use any IC socket, but you may need to slightly shorten the potentiometer legs in position RV2. Check fit before soldering.

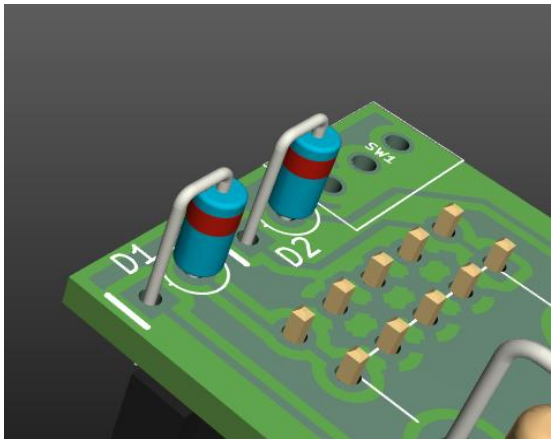
Assembly Guide

Starting on the front of the PCB (side without the MH logo):

Place and solder resistors R1 and R2. 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.

Place and solder the 2 diodes D1 and D2. These are also 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.

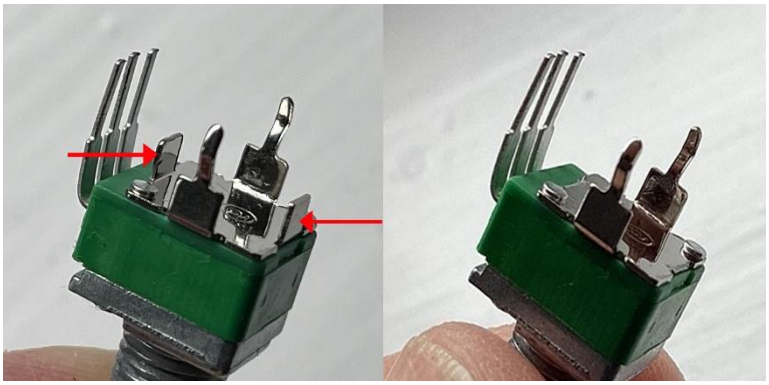


Turning to the other side of the board (the side with the MH logo), place and solder the two electrolytic capacitors, C5 and C6. Take care with orientation the long leg of the capacitor must go in the hole marked '+' with the shorter leg going into the white semi-circle. The capacitor will also have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

Place and solder the IC socket. The notch on the IC socket should be nearest the 'U1' label.

Place and solder the four 2-pin headers for jumpers and the 10-pin power connector. If you are using a shrouded power connector, make sure the notch lines up with the notch marking on the PCB.

Remove the stand off lugs from all the 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 easily come away from the potentiometer.



Put one nut on each on each of the three switches (this acts as a spacer)

Place the potentiometers, switches and jack sockets on the front of the PCB (Do not solder yet). Note pairs of jack sockets (J2+J3, J5+J6) share a solder pad for the ground leg. Push both legs through the same hole.

Put the front panel over the front panel components. Secure with the jack nuts, remaining switch 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 retighten.

Solder everything making sure there is no gap between the front panel components and the PCB.

Insert the IC in the IC socket making sure the notch on the IC lines up with the notch on the IC socket and 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 IC socket and the PCB.

Push the knobs on to the potentiometer spindles.

Select x5 gain on the mix channels if required by inserting jumpers.

Select the channel 1 to channel 2/3 normalisation if required by inserting jumper

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