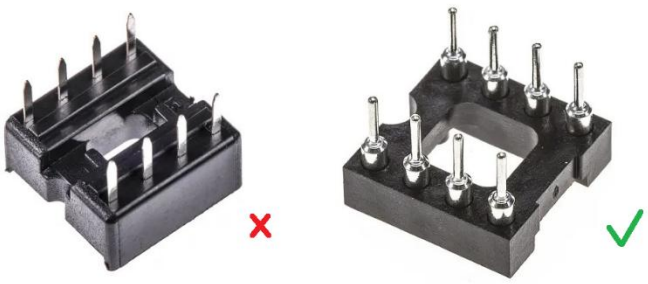
MH SMIX

BOM

Designator	Notes	Qty	Value
C1,C2,C3,C4,C5,C6	Ceramic Capacitor THT P2.50mm (104)	6	100nF
C7,C8	Electrolytic Capacitor – miniature P2.50mm	2	22uF
C9,C10,C11,C12,C13, C14,C15,C16	Capacitor_THT: Ceramic Disc P5.00mm (105)	8	1uF
D1,D2	Diode_THT P2.54mm_Vertical_KathodeUp	2	1N5818
J1,J2,J3,J4,J5,J6,J7,J8,J 10,J11	Connector_Audio:Jack_3.5mm (Thonkiconn) Vertical +jack nut	10	~
J9	Connector_IDC:IDC-Header_2x05_P2.54mm_Vertical	1	Power Connector
R1,R2	Resistor_THT:R_Axial P7.62mm_Horizontal	2	10R
R3,R4,R5,R6,R7,R8,R9, R10,R11,R12,R13,R14, R15,R16,R17,R18,R19, R20,R21,R22,R23,R24	Resistor_THT:R_Axial P7.62mm_Horizontal	22	100k
R25,R26	Resistor_THT:R_Axial P7.62mm_Horizontal	2	1k
RV1,RV2,RV3,RV4,RV5, RV6,RV7,RV8	Potentiometer THT: AlphaSingle_Vertical +Nut	8	A50k
SW1,SW2,SW3,SW4,S W5,SW6,SW7,SW8	Sub miniature toggle switch SPDT ON-OFF-ON with 2x nuts	8	SW_SPDT_321
U1,U2	IC DIP-8_W7.62mm	2	TL072
U1,U2	IC Socket (turned legs)*	2	
	Pot Knob	8	

*The IC sockets need to accommodate components soldered beneath them so allow clearance:



Assembly Guide

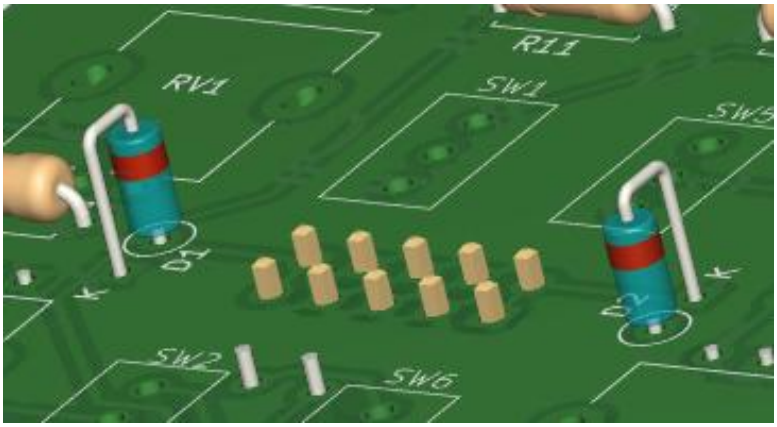
Starting on the front of the PCB (side without the MH logo):

Place and solder resistors R1 to 4, R11, R12. Orientation is not an issue with resistors. Note these are different values – Check BOM

Place and solder C1 to C6. Again, orientation is not an issue with these components.

Place and solder the 2 diodes D1 and D2. 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:



Turning to the other side of the board (the side with the MH logo), place and solder resistors R5 to R10 and R13 to R26. Note these are different values – Check BOM

Place and solder C9 to C16. Again, orientation is not an issue with these components.

Place and solder the two electrolytic capacitors, C7 and C8. 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 sockets (U1 and U2). The notch on the IC sockets should line up with the notches on the PCB. Note that the IC sockets need to be 'turned leg' type – see BOM

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

Screw one nut onto each of the switches – these act as spacers.

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

Put the front panel over the potentiometers, switches and jack sockets. Secure with relevant nuts.

Check that the jacks are still aligned to the PCB and have not been significantly 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 ICs in the IC sockets making sure the notch on the ICs line up with the notch on the IC sockets and the PCB. If there is no notch on the ICs there should be a dot or circle in one corner. In this case place the ICs so that the dot/circle is nearest the notch on the IC socket and the PCB.

Push the knobs on to the potentiometer spindles.

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