

# THC Build Document

V1



**GMSN!**  
design | build | repair

## Intro

This is a version of the incredibly popular Mutable Instruments Clouds utilising Through Hole rather than SMD components.

Components are split across two PCBs, Control and Interface.

The Control PCB has SMD pads for the STM32 and WM8731 chips. There is an option to have these presoldered and for the STM32 to come preflashed.

The Interface board matches the same dimensions as the original Clouds, however the In Gain pot hole may need to be widened to 9mm to allow for a different model of pot. All the analogue circuitry is mounted on the interface board.

The hardware will run the same firmware as the original as well as alternative firmwares such as [Parasite](#)

There are 2.54mm mounting holes and provisions for MTA-100 and MTA-156 power connectors for formats other than Eurorack. The module will accept +- 15v with no modifications.

The module is available as a PCB set, with the options of components, presoldered SMD chips, and preflashed STM32.

An SMD stencil is also available to assist with applying solder paste for the STM32 and WM8731.

Please note this product is in no way connected to or supported by Mutable Instruments.

### Through Hole Clouds BoM

Component	Value	Qty	Interface PCB	Control PCB
ARM Microcontroller	STM32F405RGT6	1		IC5
Audio Codec	WM8731	1		IC8
Diode	1N4002	2		D1, D2
Resistor	100k	8	R21, R24, R29, R32, R38, R39,	R12, R15
Resistor	10K	4		R13, R14, R16, R17,
Resistor	180k	1	R36	
Resistor	1k	2	R61, R62,	
Resistor	200k	5	R22, R23, R28, R33, R37,	
Resistor	27k	2	R57, R58,	
Resistor	2k2	2		R40, R41
Resistor	33k	3	R25, R26, R35	
Resistor	33R	1	R1	
Resistor	39k	4	R48, R49, R50, R59	
Resistor	3k3	2	R46, R47	
Resistor	470R	1	R30	
Resistor	47k	1	R34	
Resistor	4R7	1		R11
Resistor	5k6	6	R51, R52, R53, R54, R55, R56	
Resistor	66k5	2	R27, R31	
Resistor	68R	8	R3, R4, R5, R6, R7, R8, R9, R10	
Ceramic Capacitor	100nF	19	C5, C15, C16, C23, C38, C39, C45, C46	C3, C8, C14, C17, C18, C25, C26, C29, C30, C32, C34
Ceramic Capacitor	15pF	4	C40, C42	C6, C7
Ceramic Capacitor	1nF	6	C9, C10, C11, C20, C21, C22,	
Ceramic Capacitor	1uF	3		C12, C28, C31
Ceramic Capacitor	220pF	2	C43, C44	
Ceramic Capacitor	2u2	2		C19, C24
Electrolytic Capacitor	10uF	5		C33, C35, C36, C37, C41
Electrolytic Capacitor	47uF	5		C1, C2, C4, C13, C27
Crystal	8Mhz	1		Q3
Transistor	2N3904	2		Q1, Q2
DIL Socket	8 Pin	3	IC3, IC9, IC10	
DIL Socket	14 Pin	1	IC4	
DIL Socket	16 Pin	1	IC1	
Voltage Reference	LM4040DIZ	1	IC6	
Voltage Regulator	UA78M33CKCS	2		IC2, IC7
PCB Header Male	1x6 2.54mm	1	JP1	
PCB Header Male	2x5 1.27mm	1	JP2 (MINI JTAG)	
PCB Header Male	1x3 2.54mm	1	JP6	
PCB Header Female	1x3 2.54mm	1		JP6
PCB Header Male	2x4 2.54mm	2	JP3, JP5	
PCB Header Female	2x4 2.54mm	2		JP3, JP5
PCB Header Male	2x5 2.54mm	2	JP4, PWR1	
PCB Header Female	2x5 2.54mm	1		JP4
Push Button LED Switch	LP40A1PBCT	1		SW1
Switch SPST (6mm PCB)	E-switch TL1105SPF250Q	4	SW2, SW3	SW4, SW5
Switch Cap	E-switch 1RBLK	2	SW2, SW3	
Alpha 9mm Vertical Linear Pot	10KB	6	R18, R19, R20, R43, R44, R45	
3.5m Jack	PJ-301-M-12	12	J1, J2, J3, J4, J5, J6, J7, J8, J9, J10, J11, J12,	
Stereo 50KB Pot	PTV112-4420A-A503	1	R42RA	
Dual Colour LED	WP59EGW	4	LED1, LED2, LED3, LED4	
8bit Shift Register	SN74HC595N	1	IC1	
Op Amp	LME49720	2	IC9, IC10	
Op Amp	MCP6004	1	IC4	
Op Amp	MCP6002	1	IC3	
Knob	6.35mm Round Shaft	6	R18, R19, R20, R43, R44, R45	
Knob	D Shaft	1	R42RA	
Heat Sink		1		IC2
PCB Spacers	12x2.54mm	4		
IDC Connector	16 Pin	1		
IDC Connector	10 Pin	1		
Ribbon Cable	16 core	1		
Screws	M3	4		
PCB Set		1	Interface PCB	Control PCB

## Tools

In order to build the Pure VCO you will need the following tools:

- Soldering Iron
- Solder Wire
- Snips
- Multimeter
- Oscilloscope

If this is your first kit, it is highly recommended that you either start with Synth A, and some of the other Pure Modules.

Synth A is just £10 and available from the [GMSN! Shop](#).

The Synth A Build Doc can be found in the [GMSN! Support Documentation](#).

If you are new to soldering, this is a good point to check out the [GMSN! Soldering Video](#).

## Populating the PCB

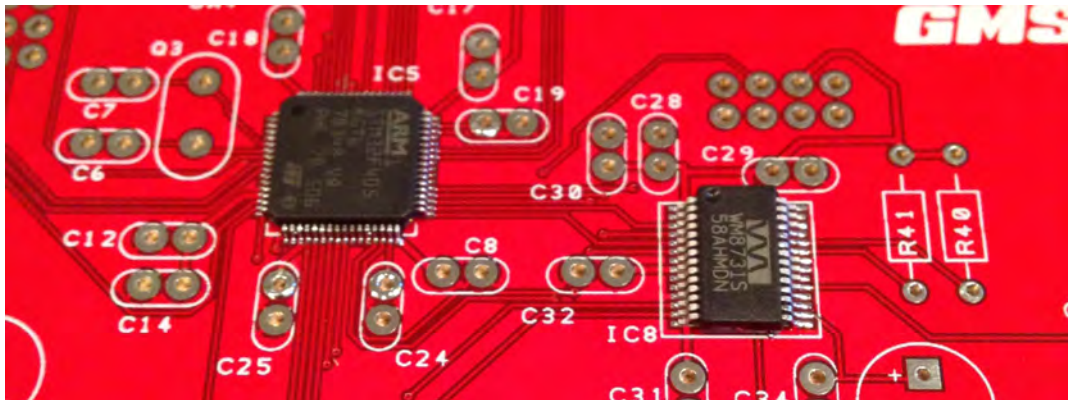
### Overview

Populating the PCB is best done starting with the lowest components in height, moving up to the highest. This helps to keep the board stable whilst soldering.

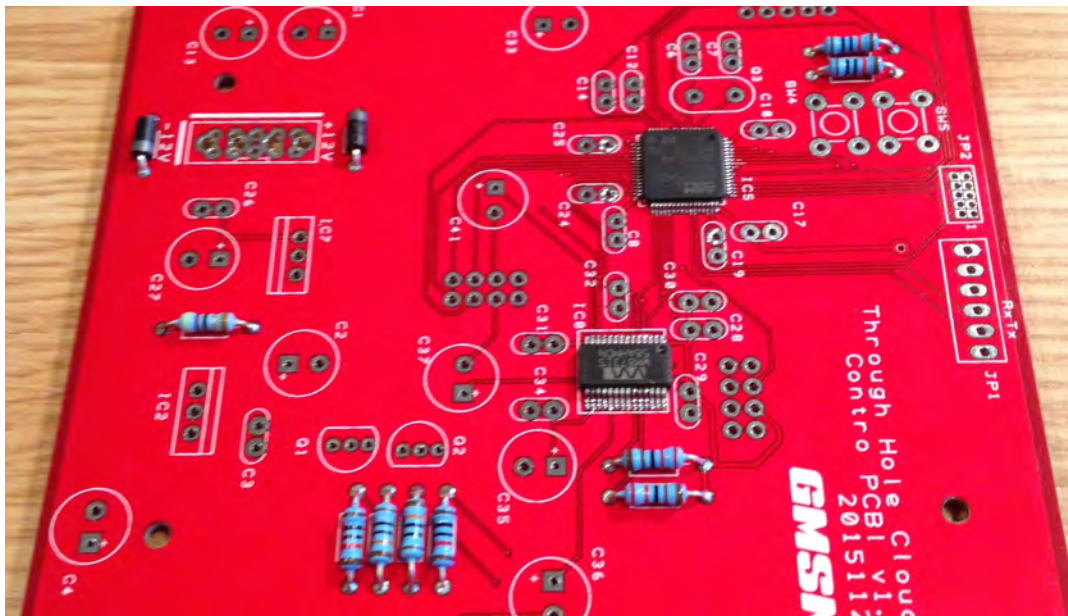
### SMD Chips

If your PCB comes with pre-flashed and soldered chips, you can skip this step.

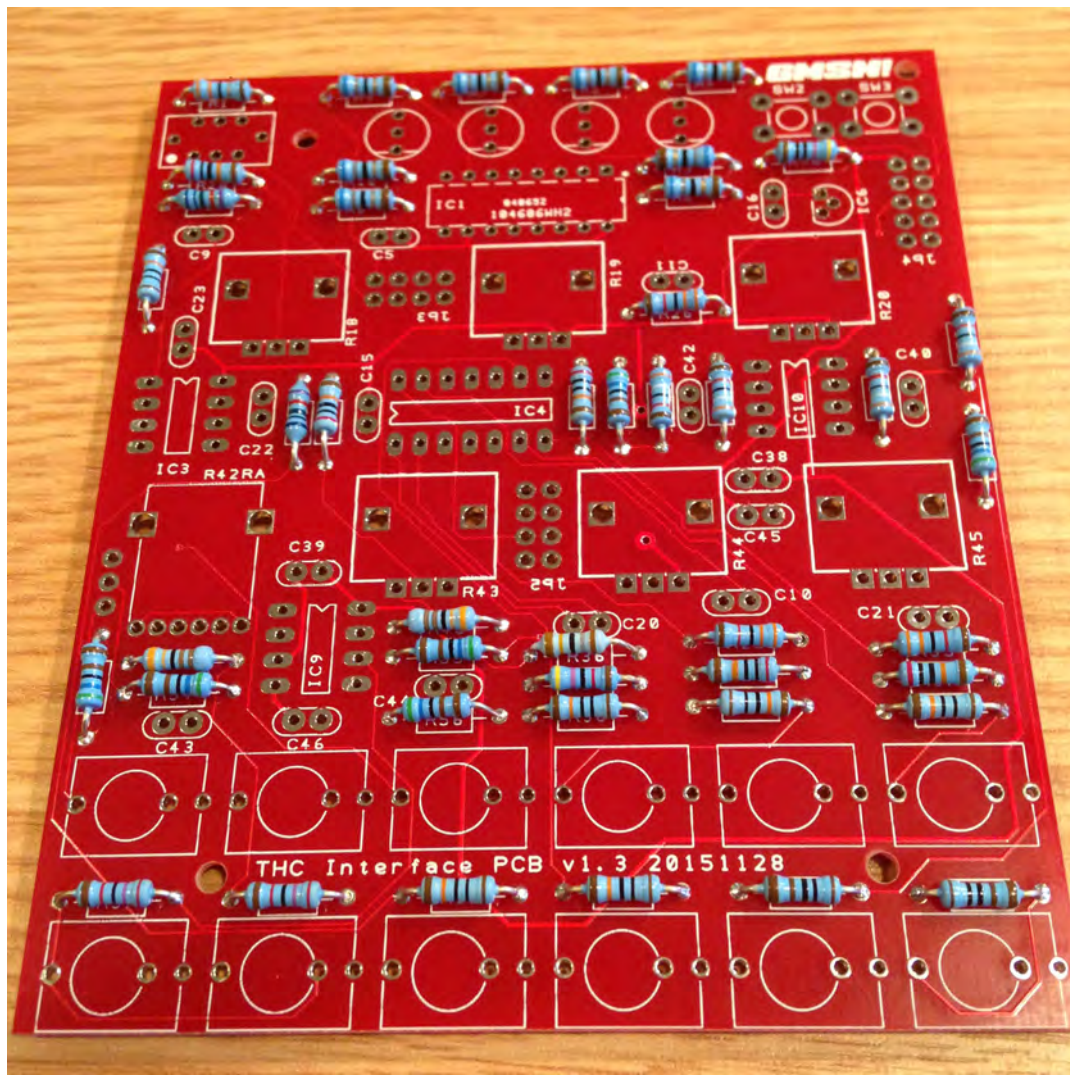
Solder the STM32 and WM8731.



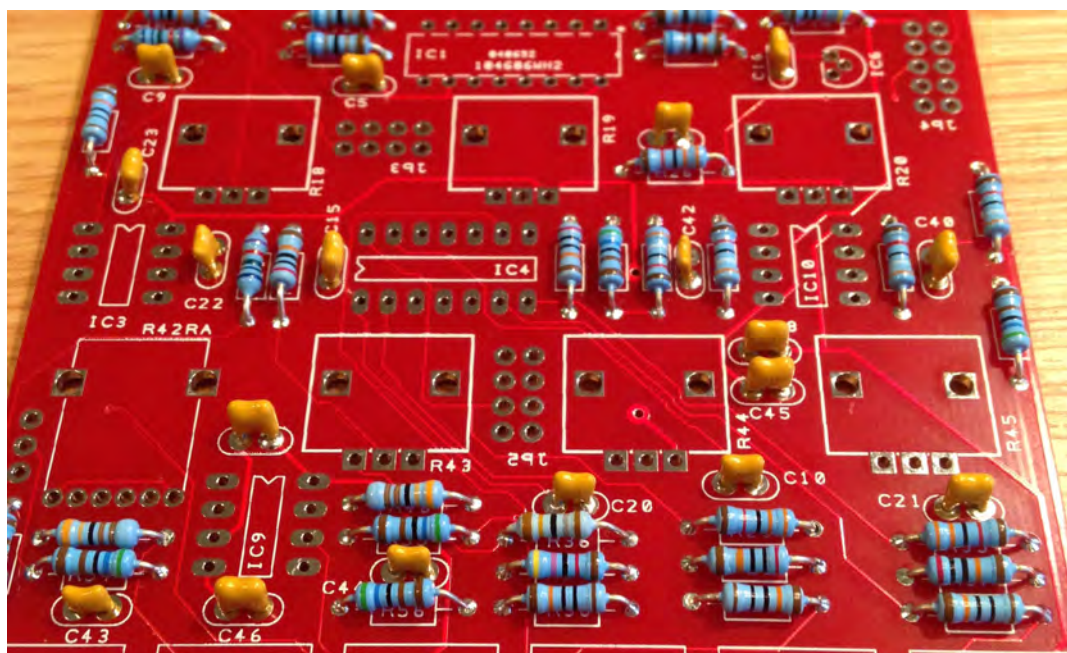
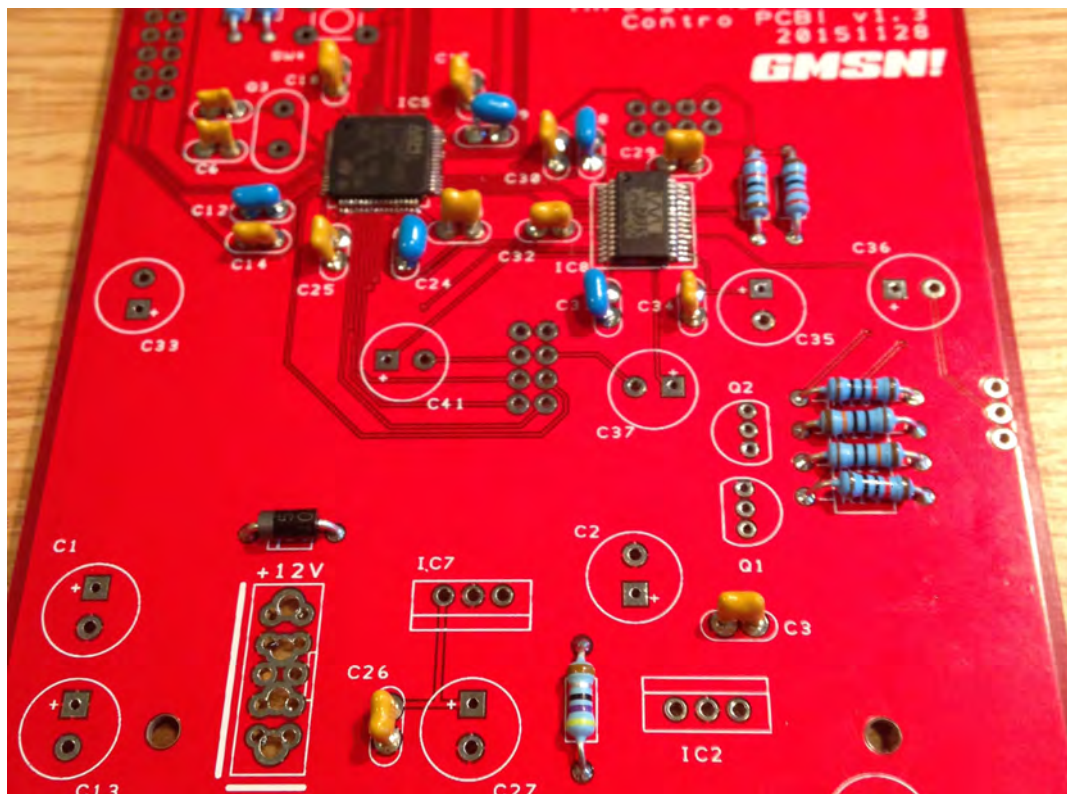
### Diodes and Resistors





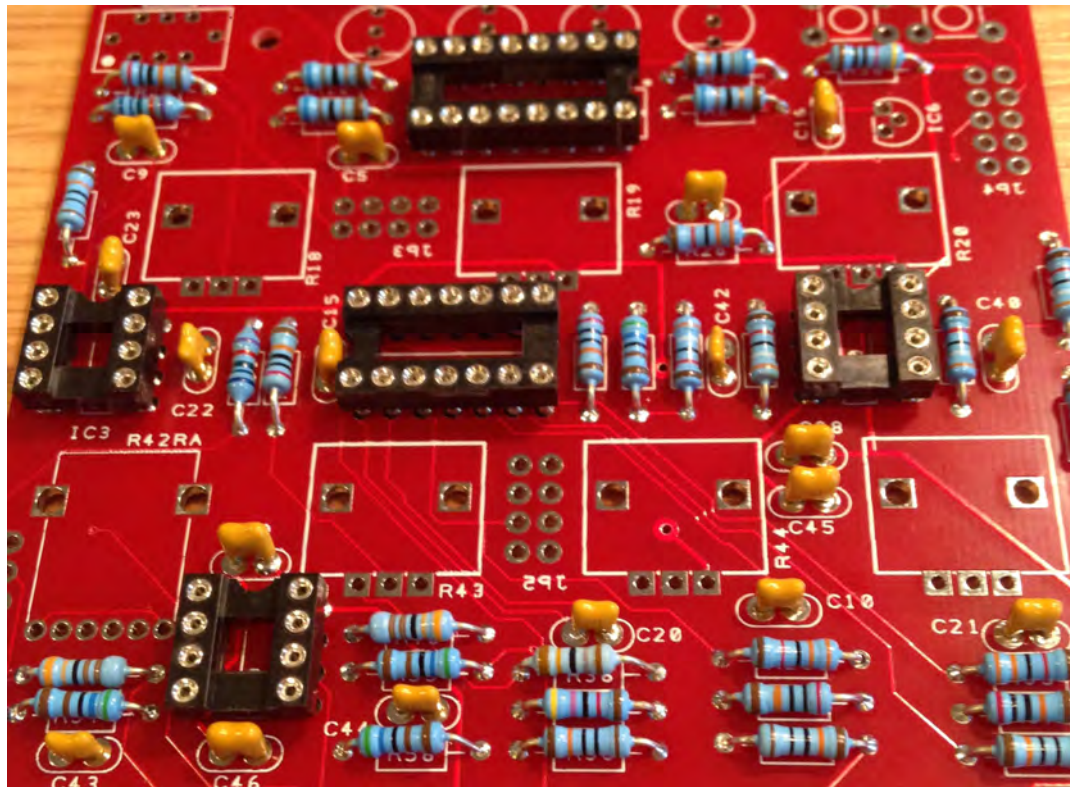


## Ceramic Capacitors

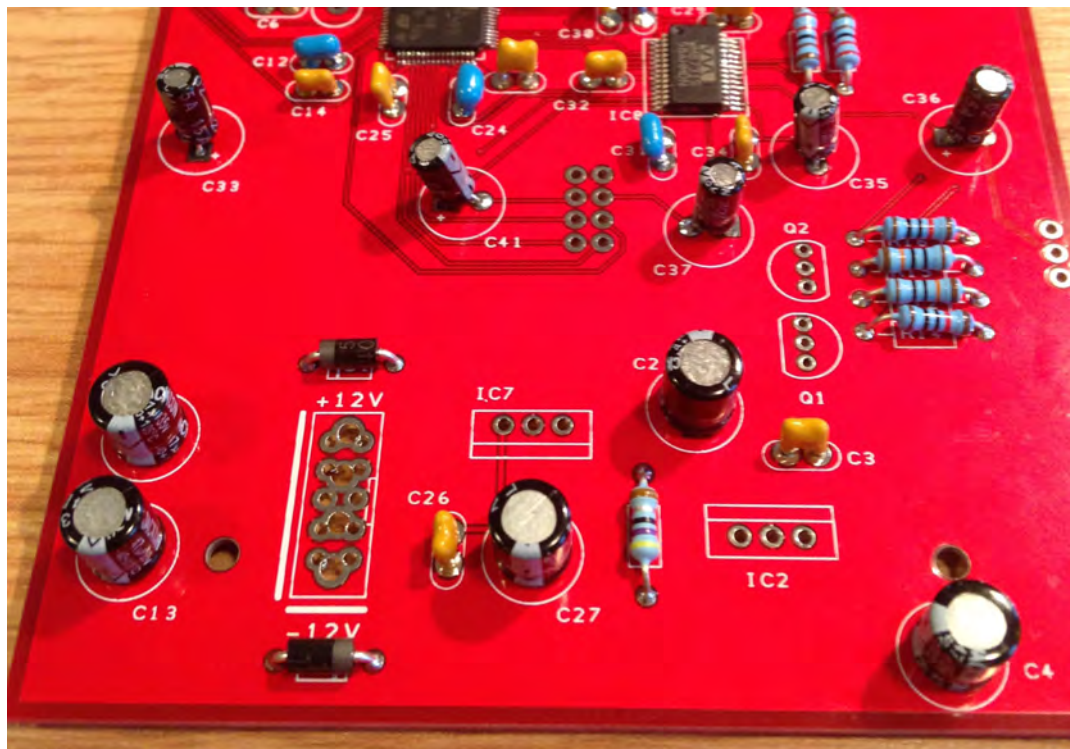




## DIL sockets

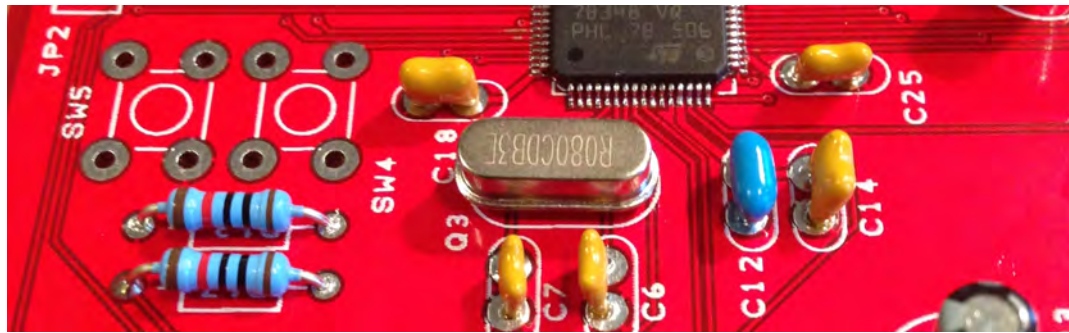


## Electrolytic Capacitors

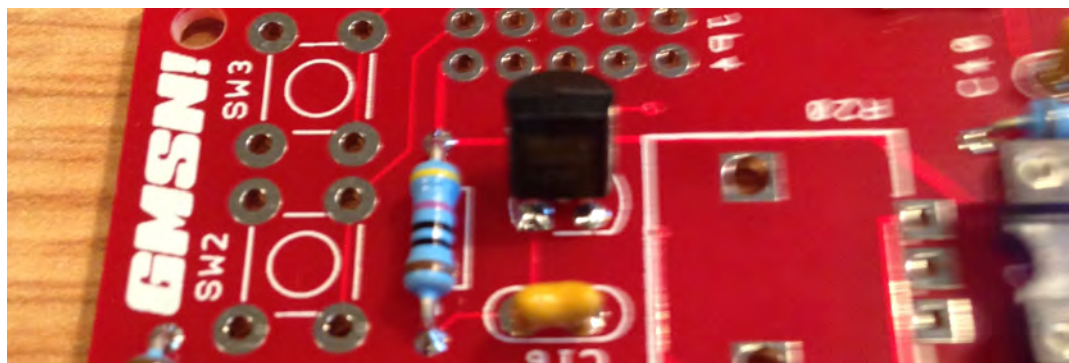




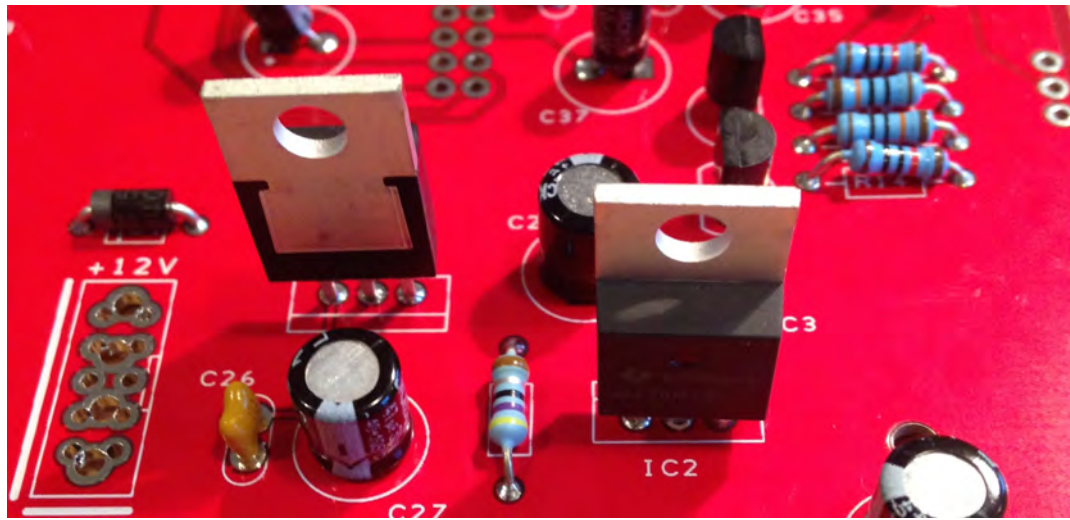
## Crystal



## Transistors and Voltage Reference

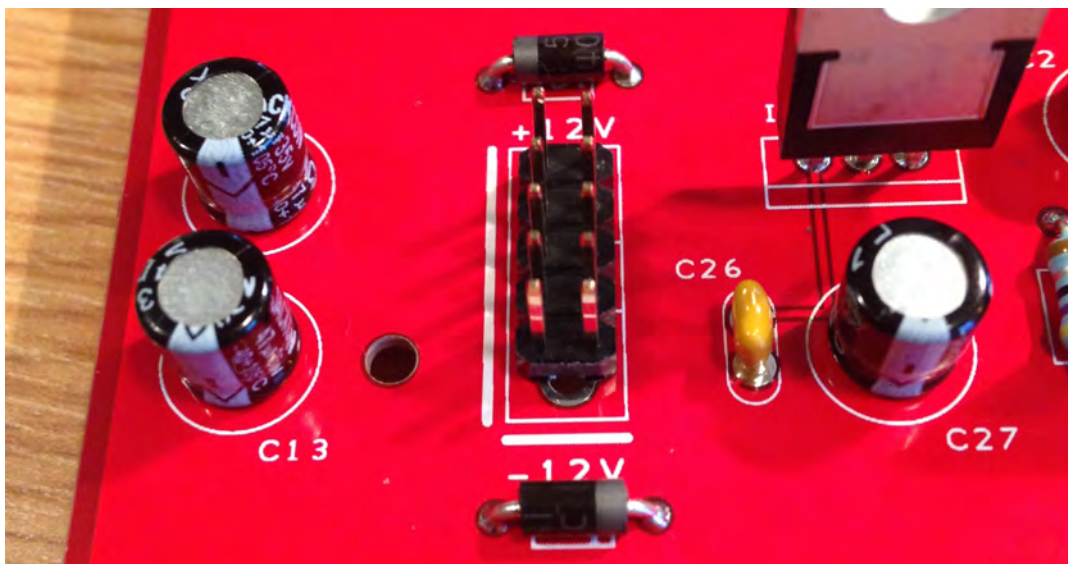
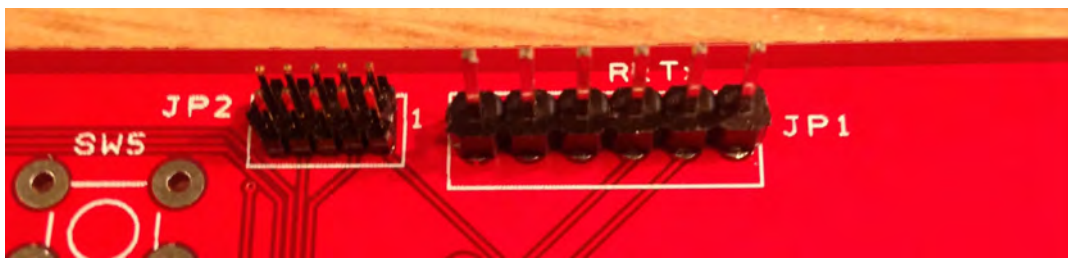
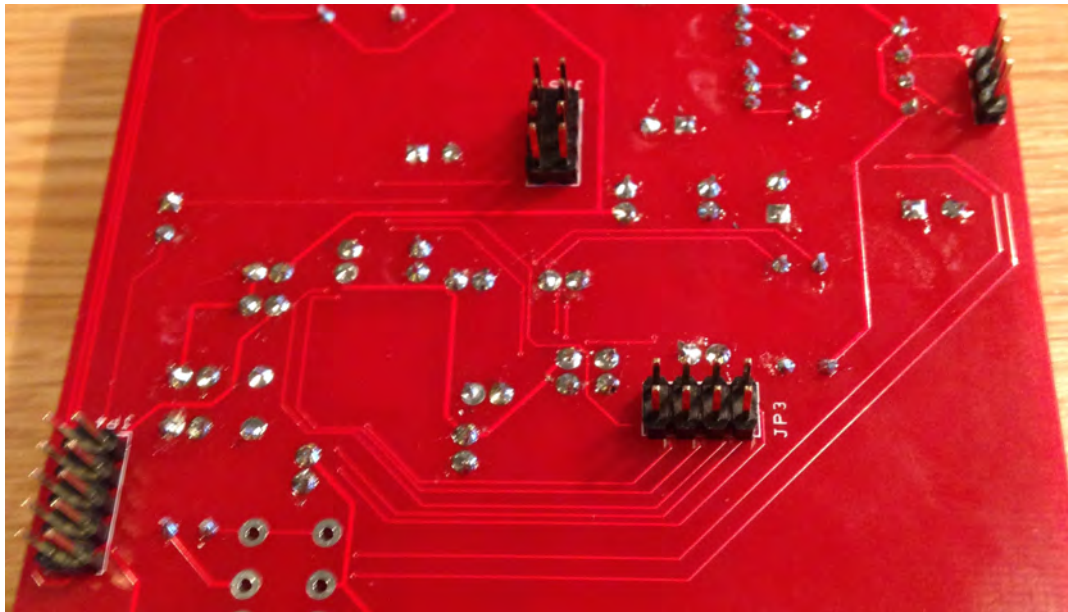


## Voltage Regulators

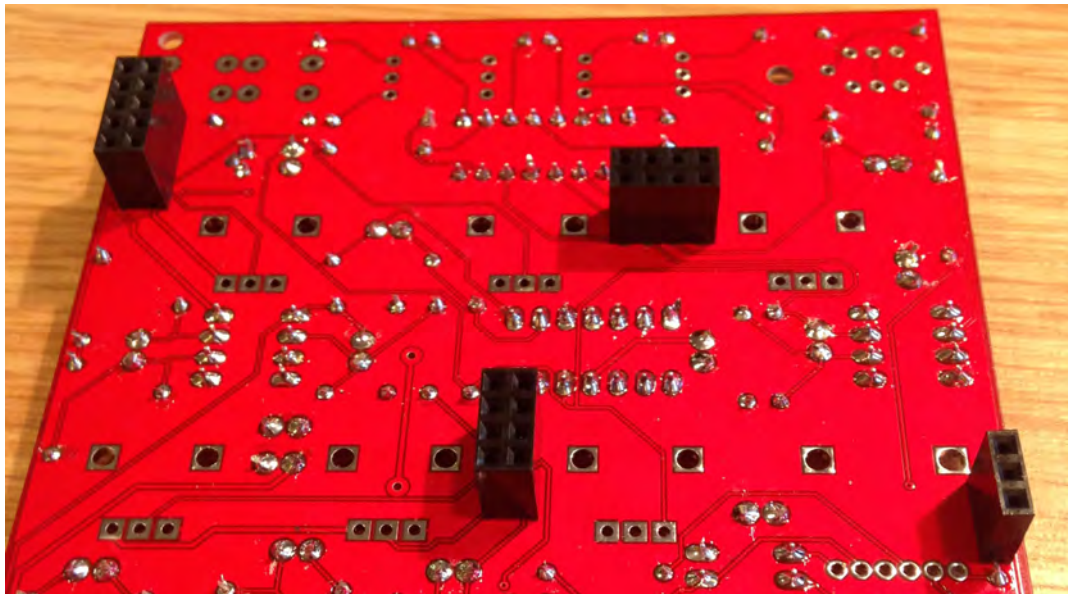


## PCB Headers and Power Connector

Watch out for solder bridges on JP2!

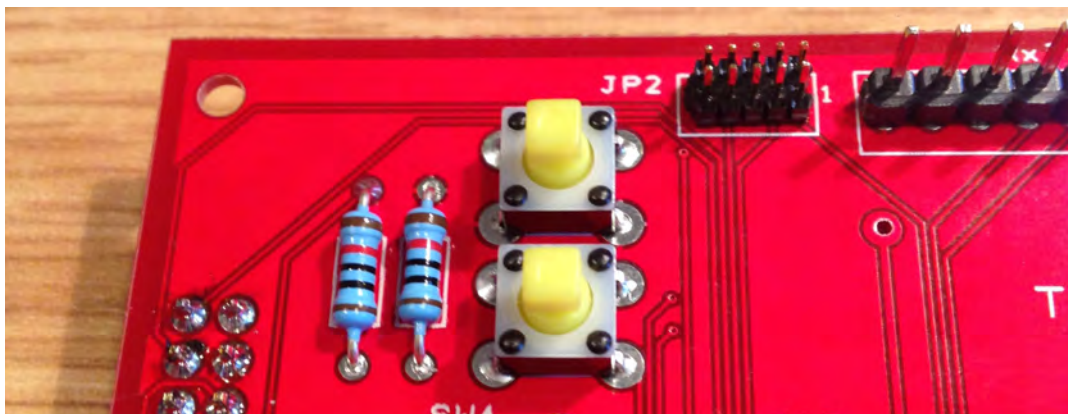






### Switches

The switches will only fit in one way.

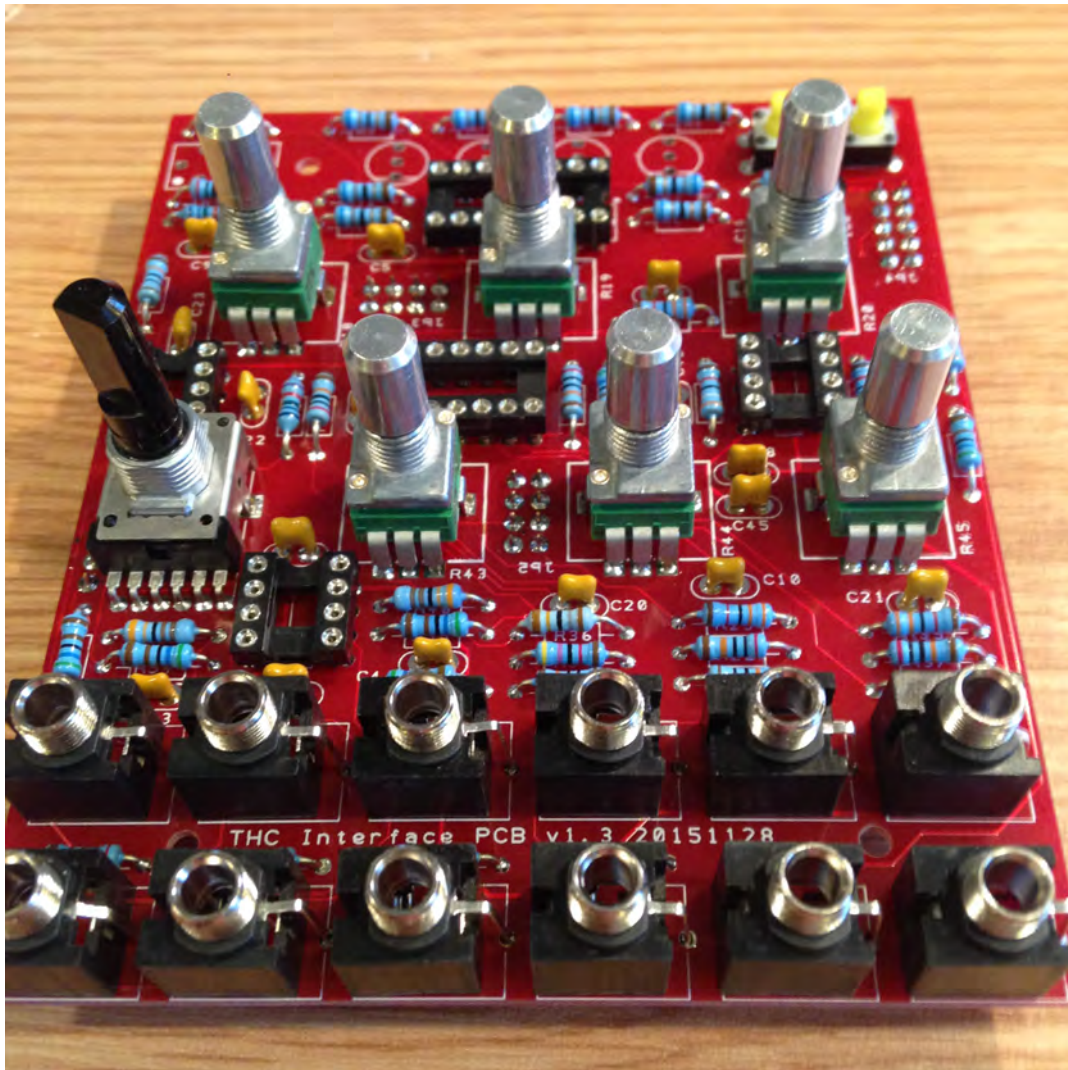


## Pots and Jacks

The old trick here is to mount everything to the PCB and panel, then solder.

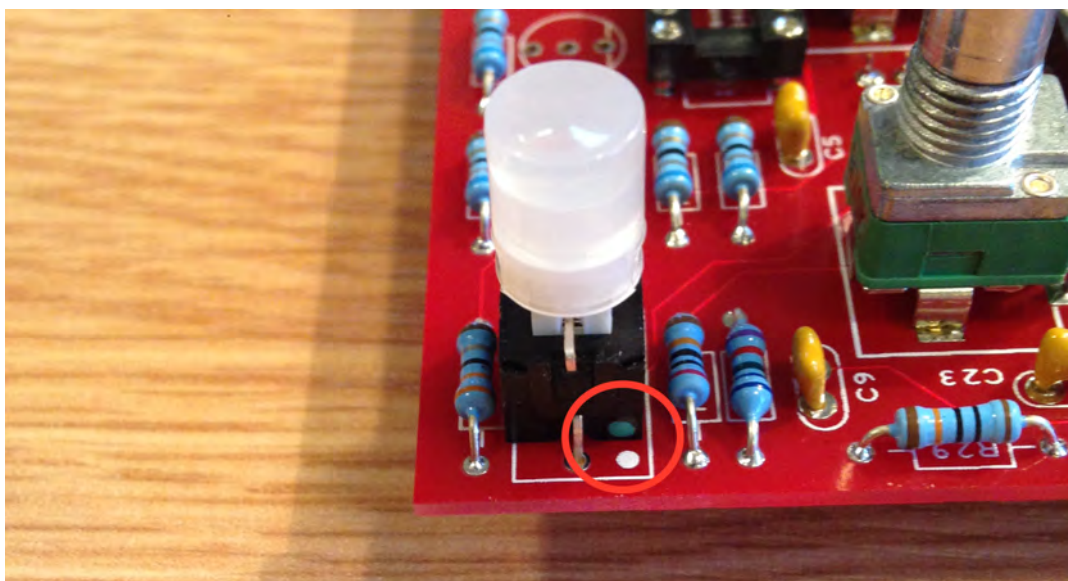






### LED Switch

Make sure the dot on the body matches the dot on the switch.



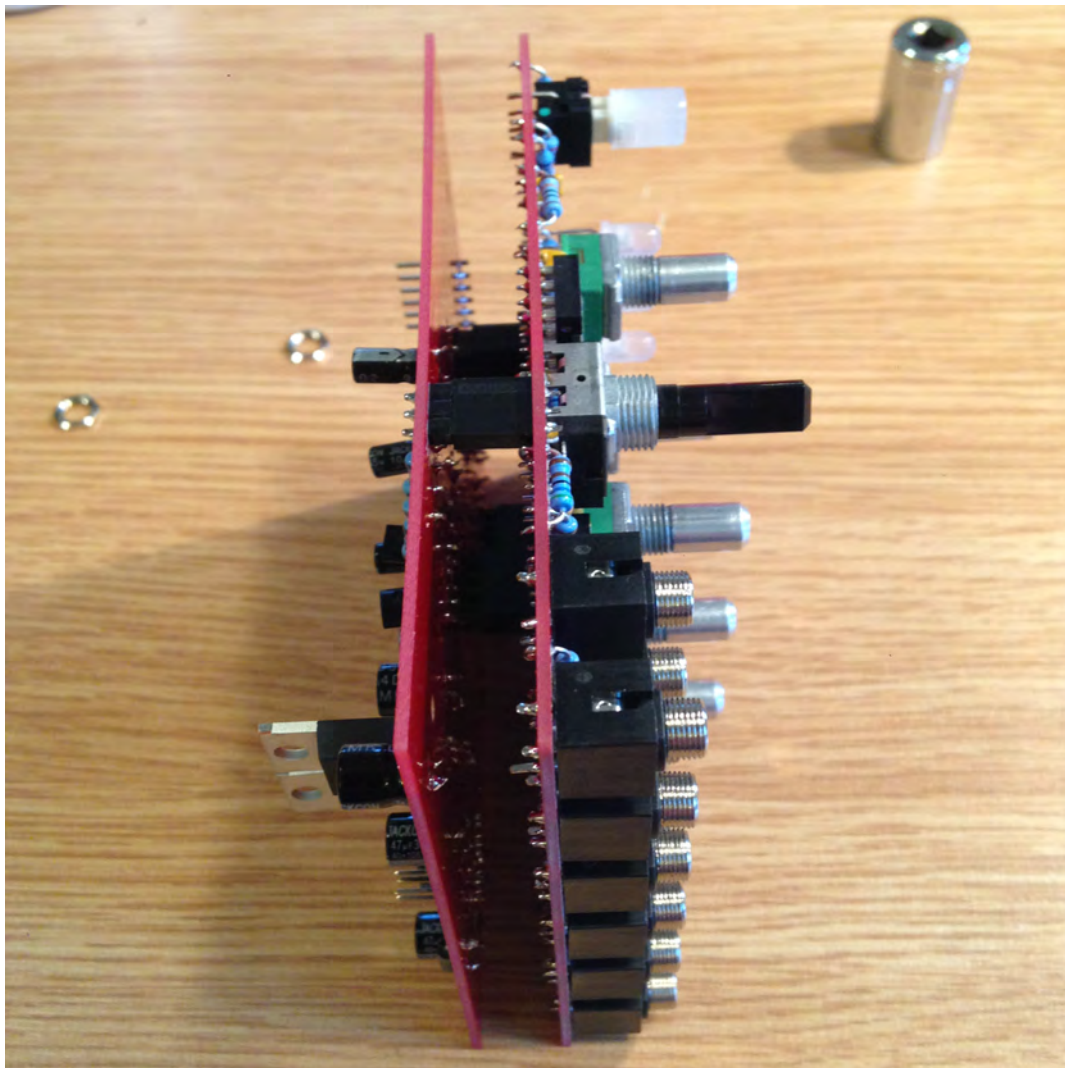


## LEDs

Fit to the panel before soldering.

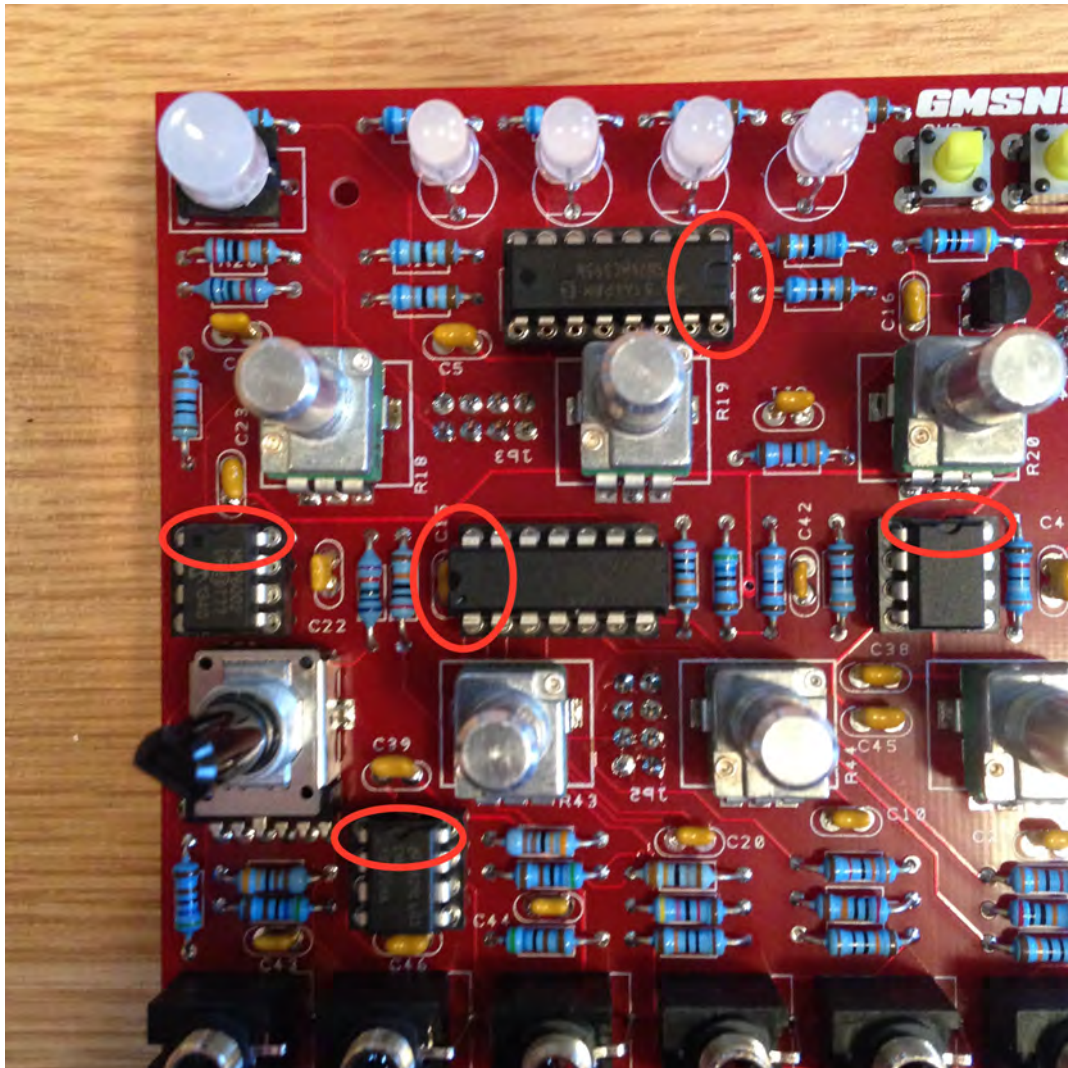


Connect the two boards together

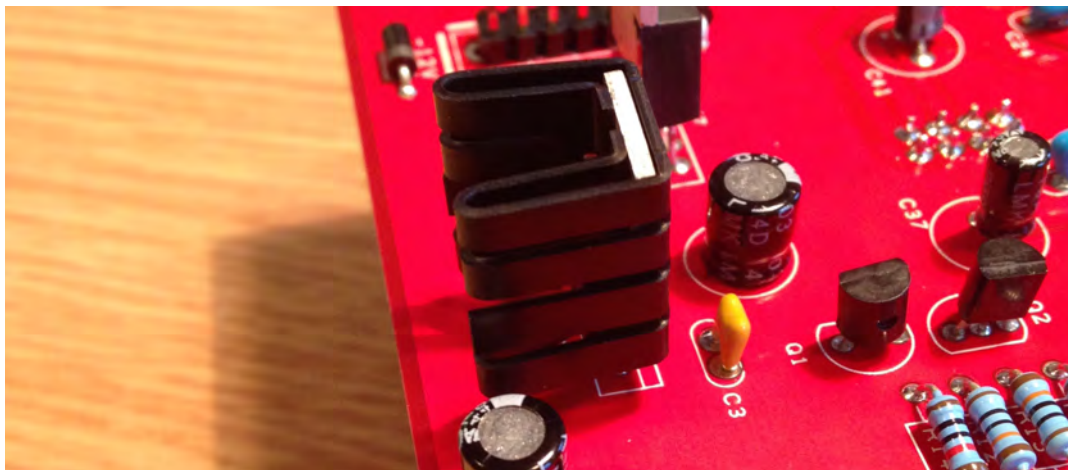


## DIL Chips

Ensure they go in the correct way round.



## Heat Sink

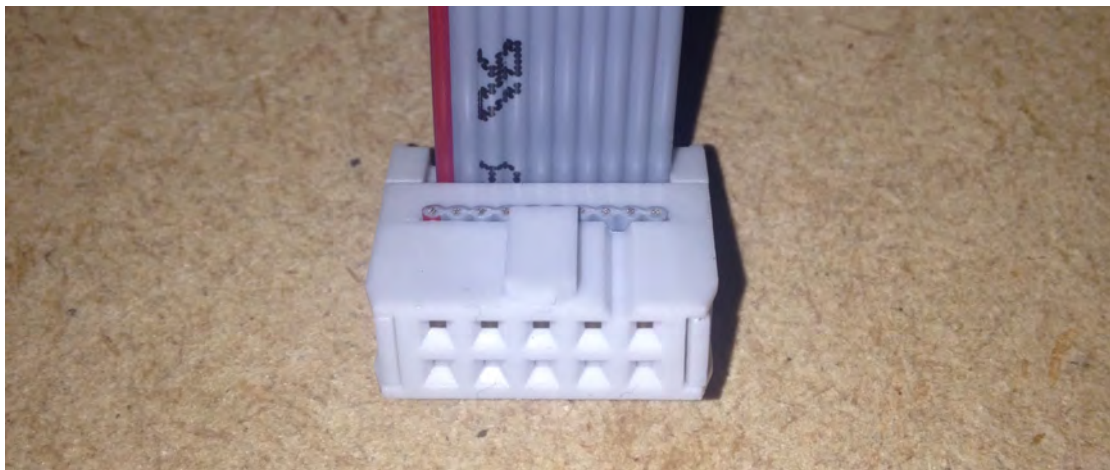




### Switch Caps



### Power Cable



To connect the IDC to the cable, align the end of the cable with the red stripe on the correct side. Place in a vice and tighten until the IDC has clicked closed.





## Power On

When you power on the THC, the four LEDs should light in turn.

Pressing the Freeze LED should cause it to toggle on and off.

Pressing the Blend button should cause the four LEDs to light in turn.

From this point on the THC will perform exactly the same as the Mutable Instruments Clouds.

The Clouds manual can be found here: [Clouds Manual](#)

## Calibration

The Pitch Pot and Pitch CV In both need calibrating and can be done using the following routine from the MI website. This will ensure unity pitch is at the 12 o'clock position of the Pitch Pot.

1. Disconnect all CV inputs.
2. Connect the note CV output of a well-calibrated keyboard interface or MIDI-CV converter to the V/OCT input.
3. Press the Load/save button, and while you hold it down, press the **Blend parameter/Audio quality** button. The first 2 LEDs will blink in orange.
4. Input a CV corresponding to a C2 note (1V).
5. Press the **Load/Save** button. The four LEDs will blink in orange.
6. Input a CV corresponding to a C4 note (3V).
7. Press the **Load/Save** push-button.
8. Calibration is done!

## Fault Diagnosis

To be perfectly honest, it's very rare for a circuit to work perfectly first time. If you are having problems there can only be a few things wrong:

- Bad solder joint
- Solder joint shorting to another
- Incorrect component placement
- Component placed the wrong way round

If you have had to unsolder several times, there's a chance the tracks have been lifted.

## ***Reflow***

By far the commonest issue is joints soldered incorrectly. Simply apply the hot soldering iron to all the joints to melt the solder. Hold the iron on for a couple of seconds to let the solder suck into the joint. Once you've done this retest.

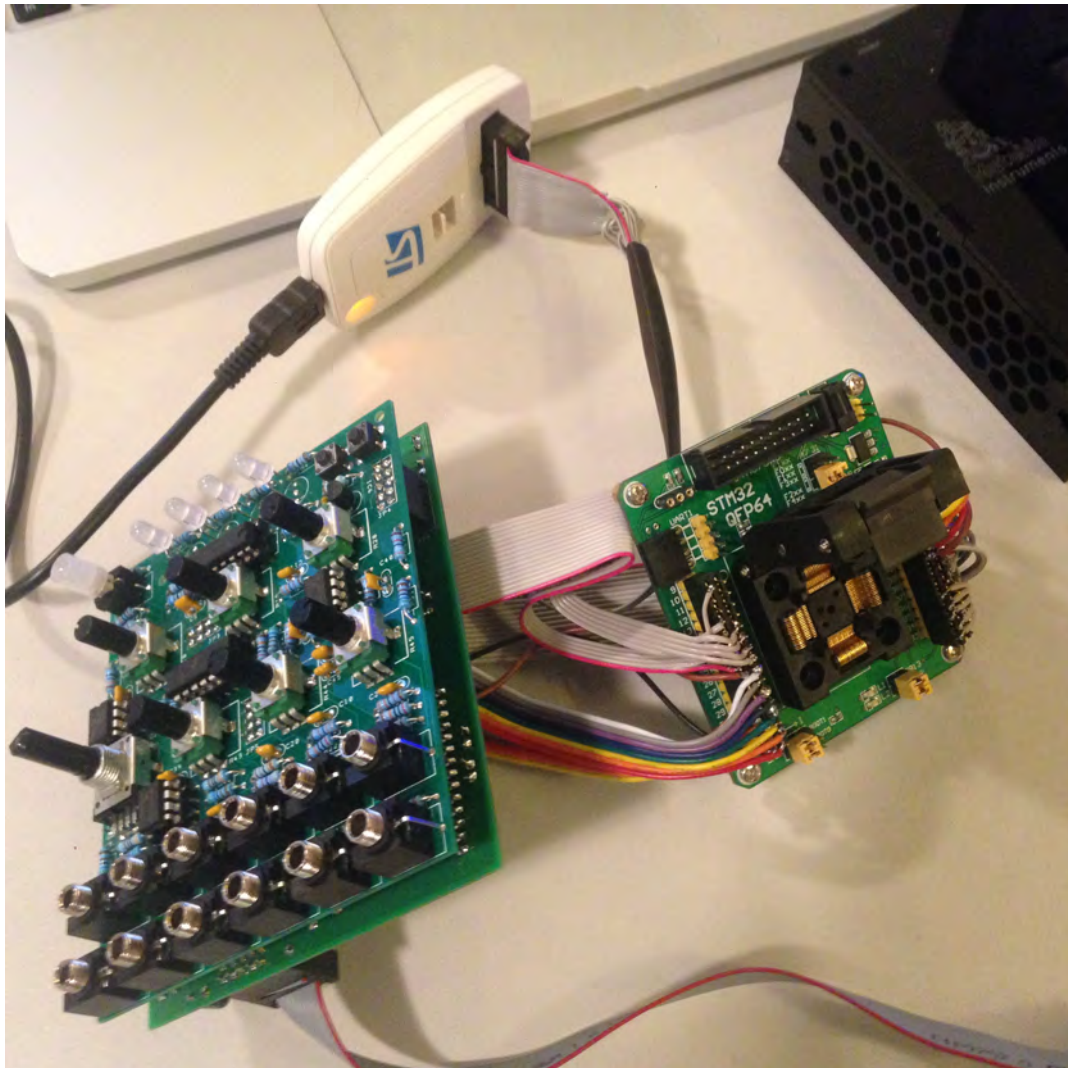
## ***Components in the wrong places or the wrong way round***

Triple check all components are in the right place. The photos above will show you where all the components should be. Also check the ICs are in the correct way round

If you're still having problems, check out the [Muffwiggler THC Build Thread](#)

***How do I know the STM32 has been flashed correctly?***

Every chip was flashed and tested on the FrankenClouds...





## **Finally...**

If you are still struggling, please send an email explaining everything you've tried with photos of both sides of the board, stating where you got the components and anything else that might be important to [GMSN! Support](#). (Don't change the subject line as there's some automated stuff going on in the background)

I hope you've enjoyed building the GMSN! THC. We can't wait to see what you're going to do with it.

More kits are available at [gmsn.co.uk/shop](https://gmsn.co.uk/shop)