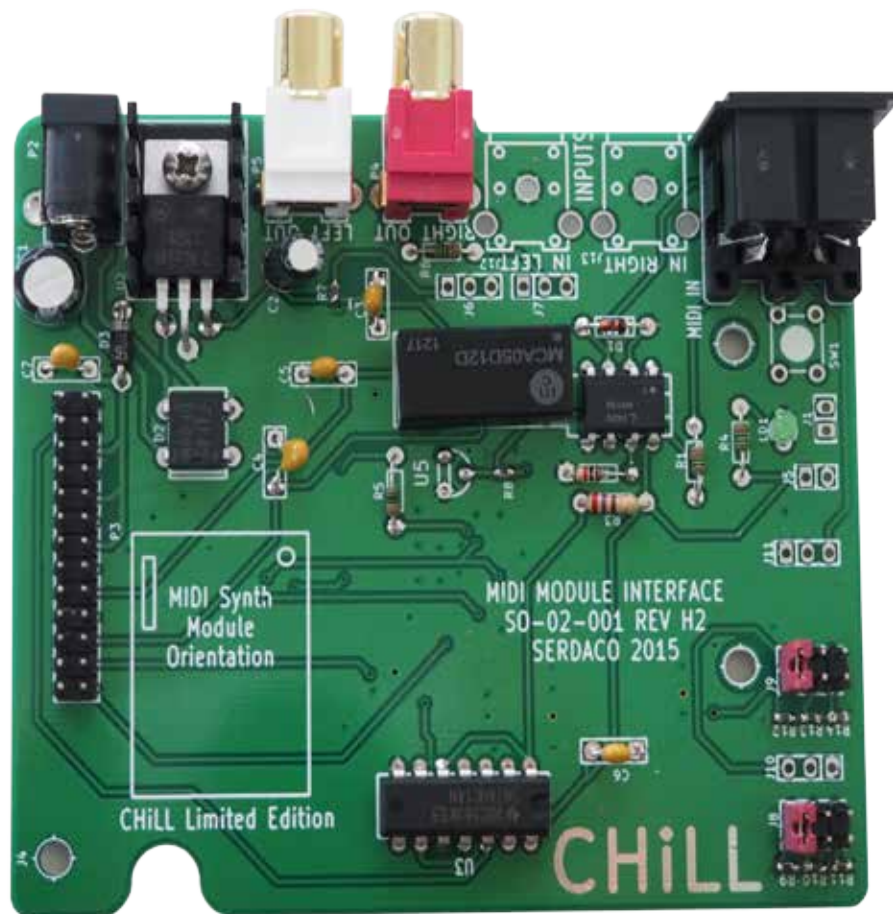


Chill Interface PCB

Assembly Instructions



Waveblaster Module MIDI Interface Board 'Chill Limited Edition' V2 – Assembly Kit

Standalone midi interface board for Waveblaster synthesizer modules.

Suitable for most Waveblaster cards.



Contents

Introduction	3
Resources	3
Requirements	4
Board	4
Schematic	5
Assembly	6
SMD Resistors	6
Header Connectors	7
Ceramic Capacitors	9
Electrolytic Capacitors	10
Diodes	10
Linear Resistors	11
Regulator - 5V	13
Bridge Rectifier	14
Miscellaneous	16
Reset Chip	18
Assembled board	18

Introduction

Thank you for purchasing the Waveblaster Module MIDI Interface Board 'Chill Limited Edition' V2 – Assembly Kit.



This board is a standalone MIDI interface board for Waveblaster synthesizer modules. Suitable for most Waveblaster cards. This board is currently only available as soldering kit.

We hope you are satisfied with this product and enjoy it!

Resources

Resources	URL
Product Information	http://www.excelvalley.com/product/waveblaster-wb02/
Headers Pinout	http://www.dream.fr/pdf/Serie5000/Soundbanks/GMBK5X64.pdf

Requirements

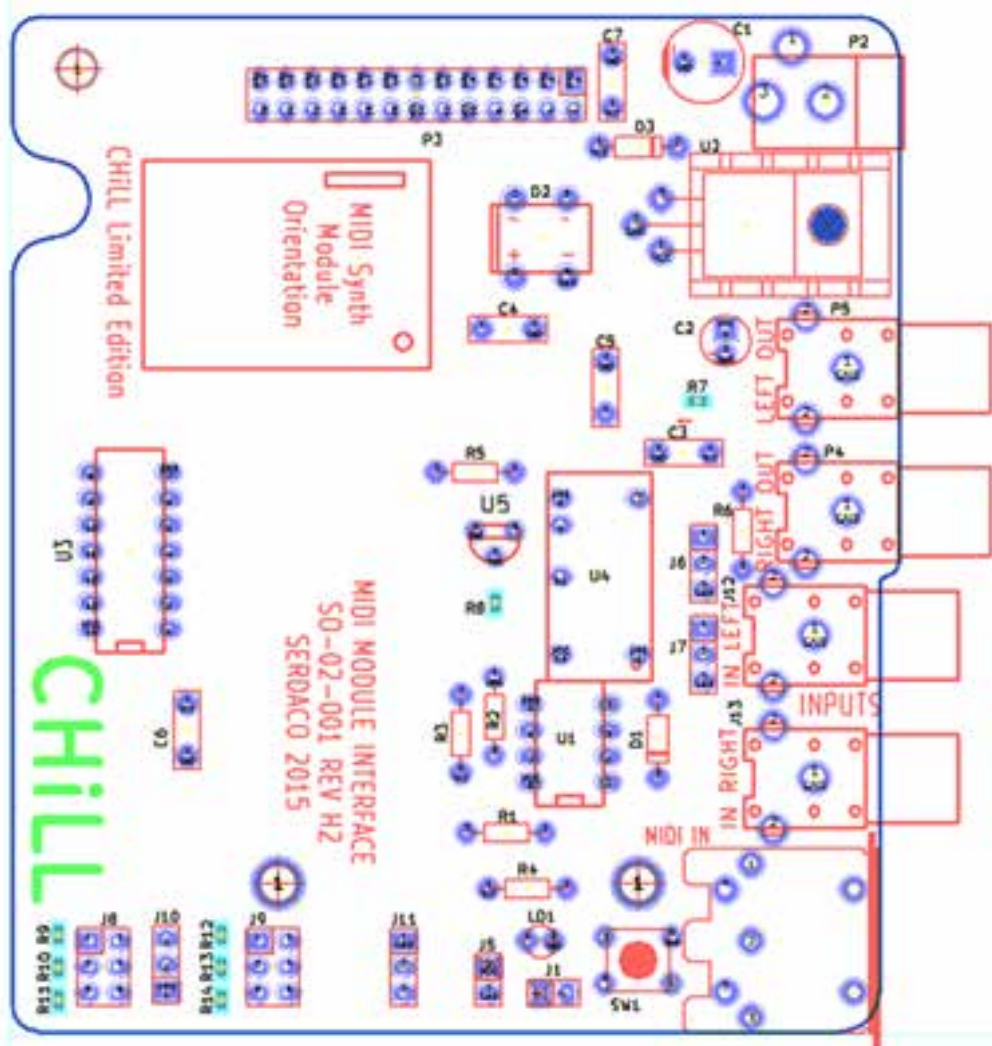
Tools:

- Precise tweezers
- Fine soldering iron
- Screwdriver
- Wire cutter

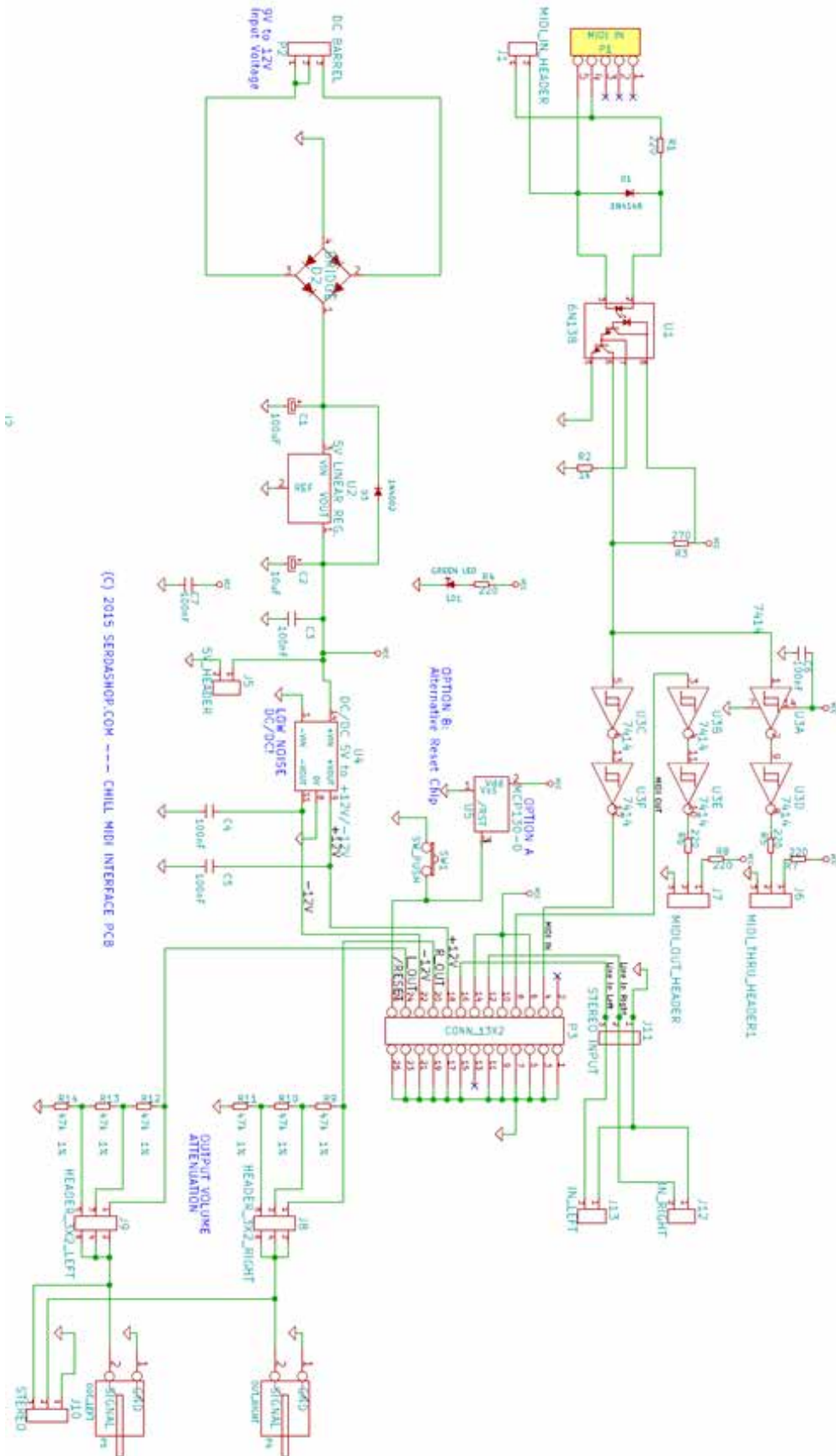
Assembly time: approximately 30 minutes

Board

Component positions



Schematic



Assembly

Identifying and placing the components on the PCB.

We will start with the smallest SMD components:

SMD Resistors

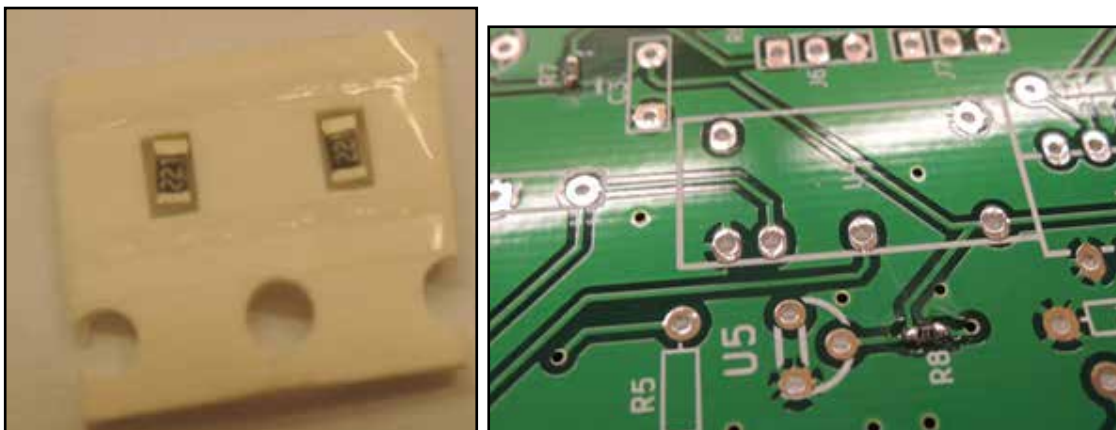
R9, R10, R11, R12, R13, R14: 0603 format SMD 47K 1% precision resistors



These resistors are used for the volume attenuation jumpers. If you do not solder these resistors, you will only be able to use the highest volume.

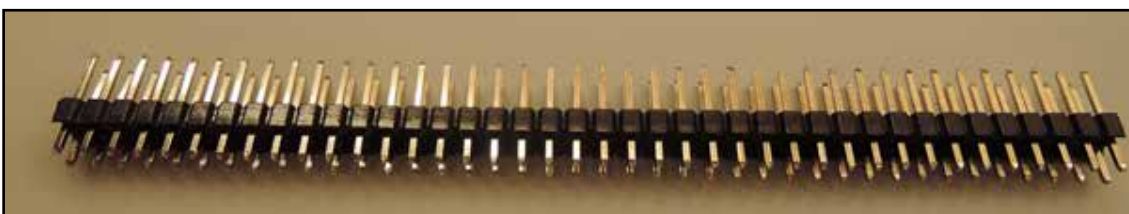


R7, R8: 0603 format SMD 220Ohm precision resistors



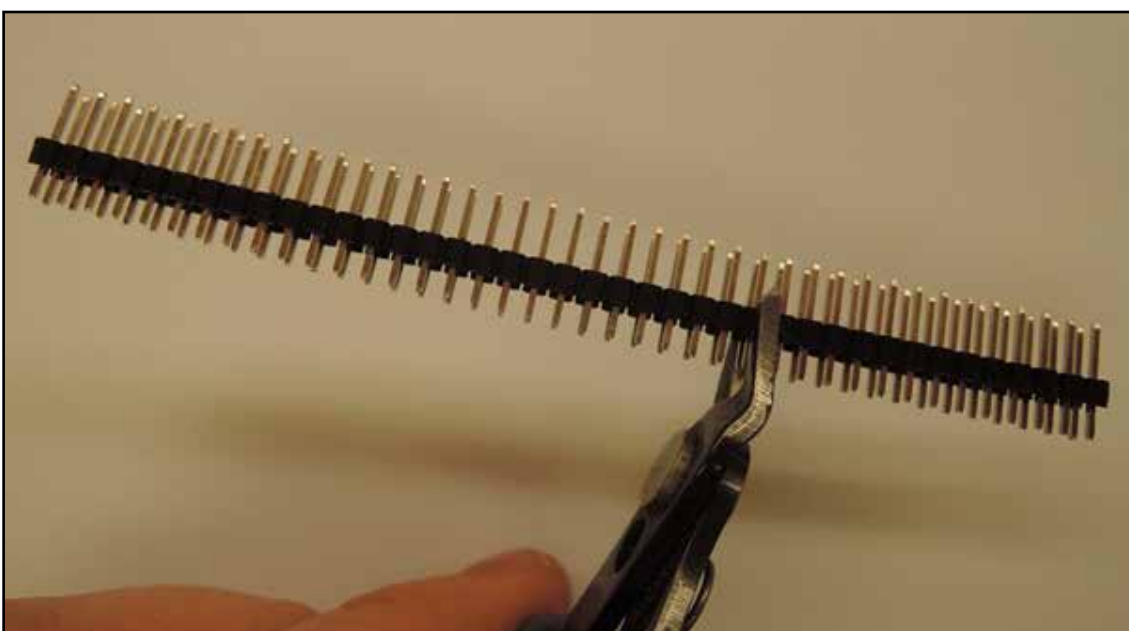
Header Connectors

The kit includes a long strip of 40 double row male headers:

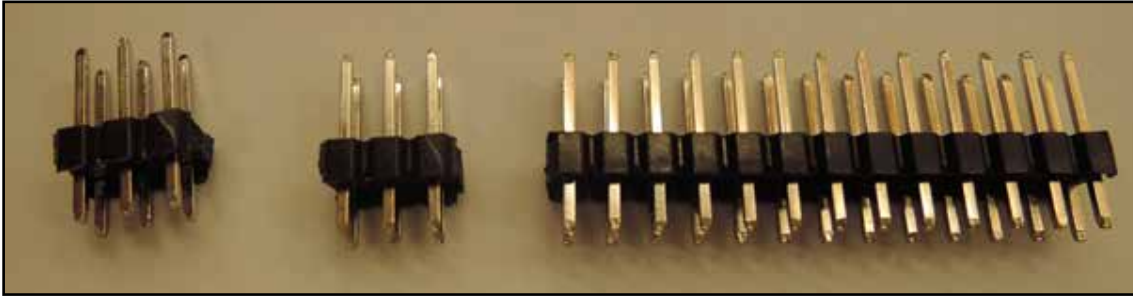


This header connector has to be cut up in 3 pieces.

We need a 13x2, and two 3x2 headers. Use your wire cutter to cut them to length.

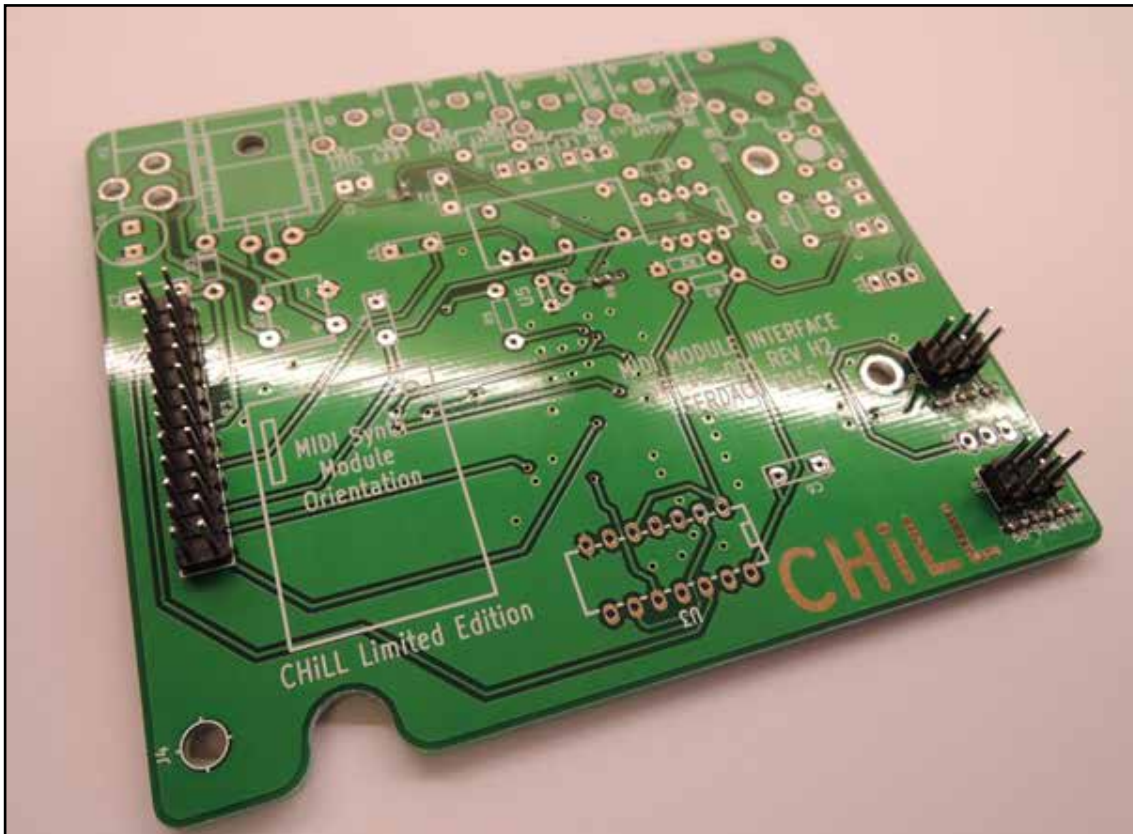


After cutting, you end up with these 3 connectors:



Solder these on the board :

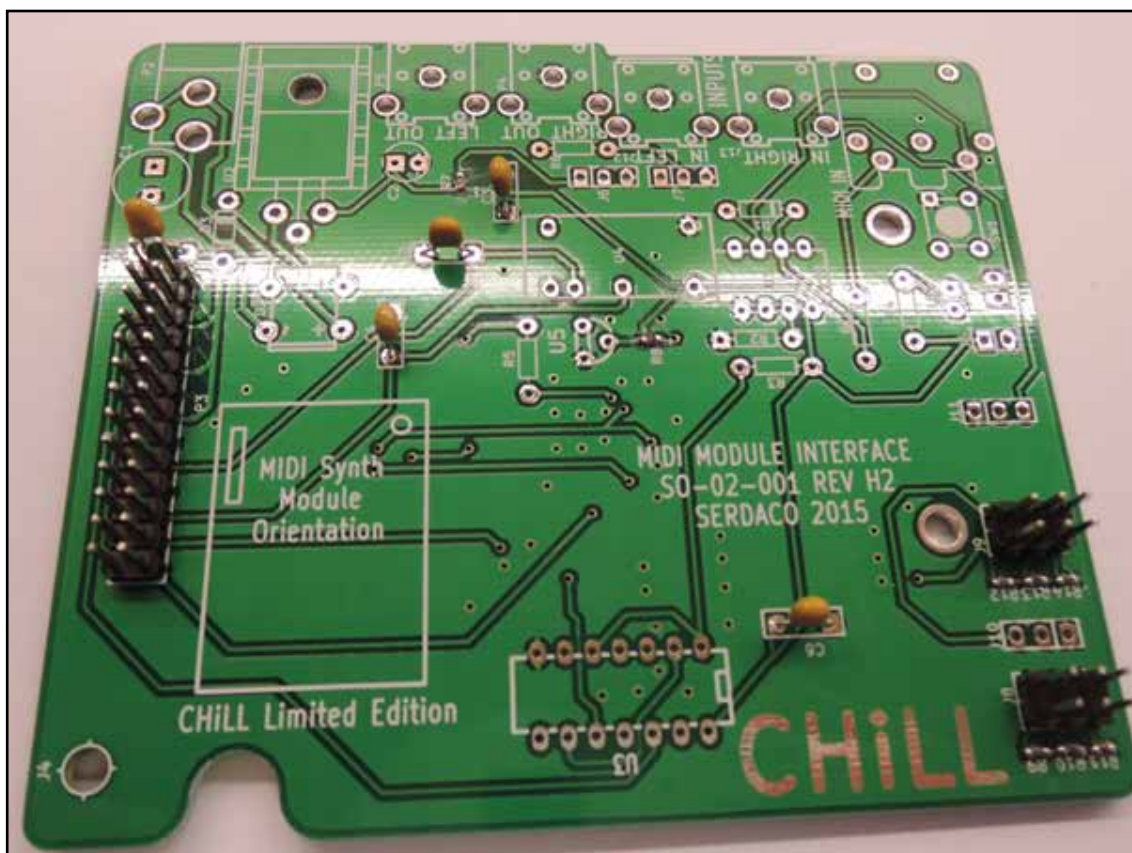
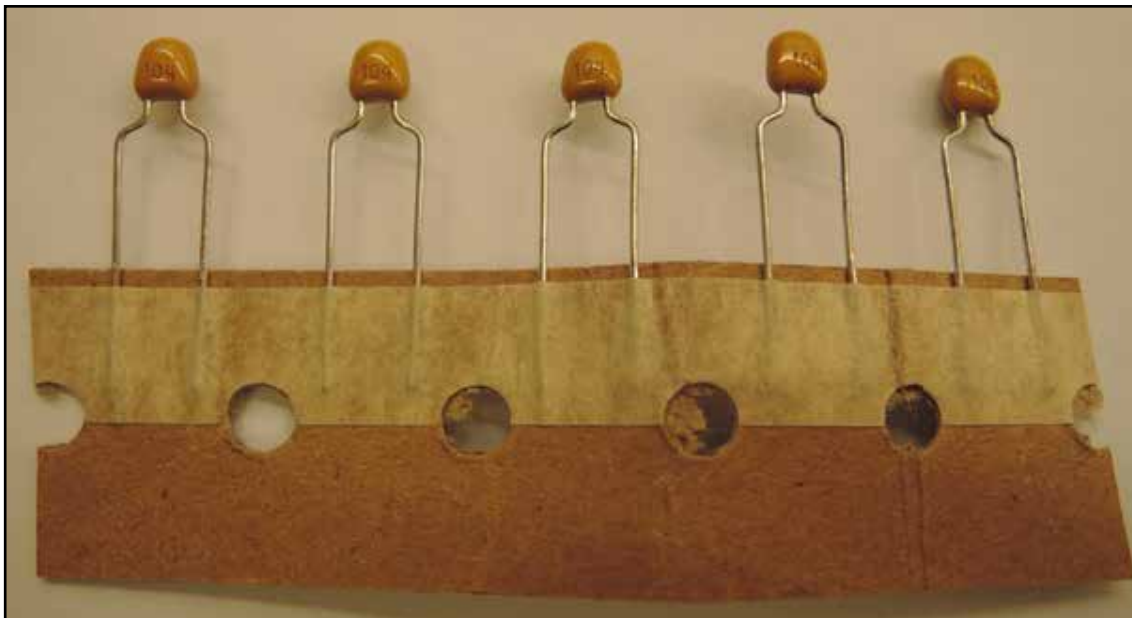
- P3: Header 13x2
- J8, J9 : Header 3x2



Ceramic Capacitors

Now we will solder some capacitors

C3,C4,C5,C6,C7 : 100nF ceramic capacitors :



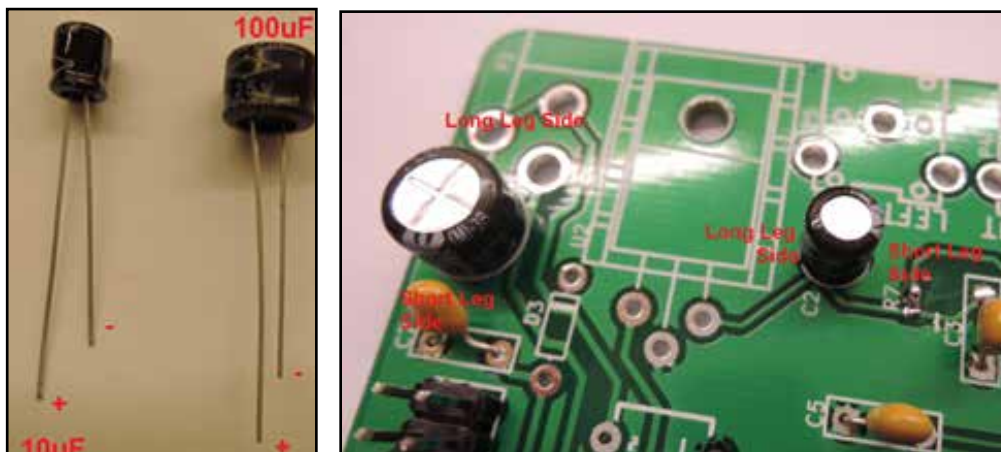
Electrolytic Capacitors

C1: 100uF electrolytic capacitor

C2: 10uF electrolytic capacitor

Make sure you respect the polarity!!

For these capacitors, the shortest leg is the Negative side.



Diodes

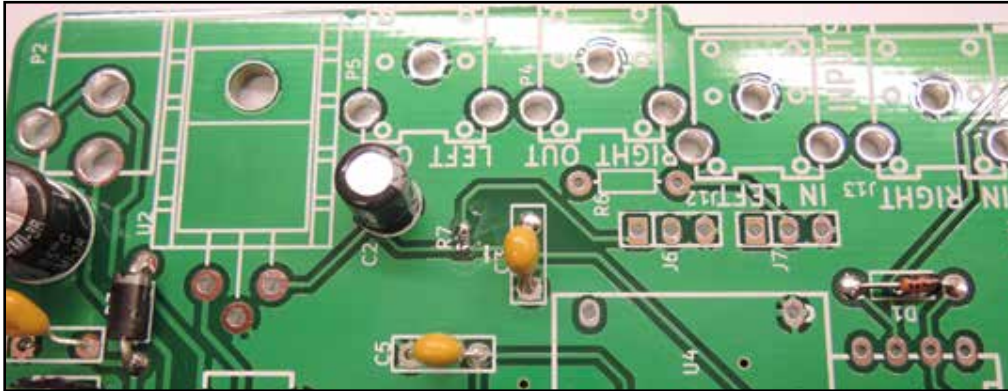
D1: signal diode



D2: protection diode



Diodes D1 and D2 on the PCB.



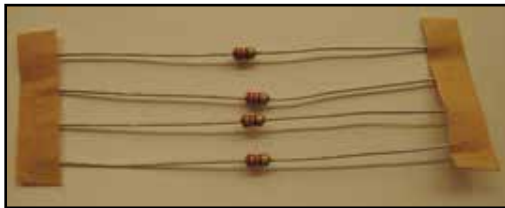
Double check the polarity of the diodes, this is important.

Linear Resistors

R1,R4, R5,R6 : a pack of four 220 Ohm Resistors:

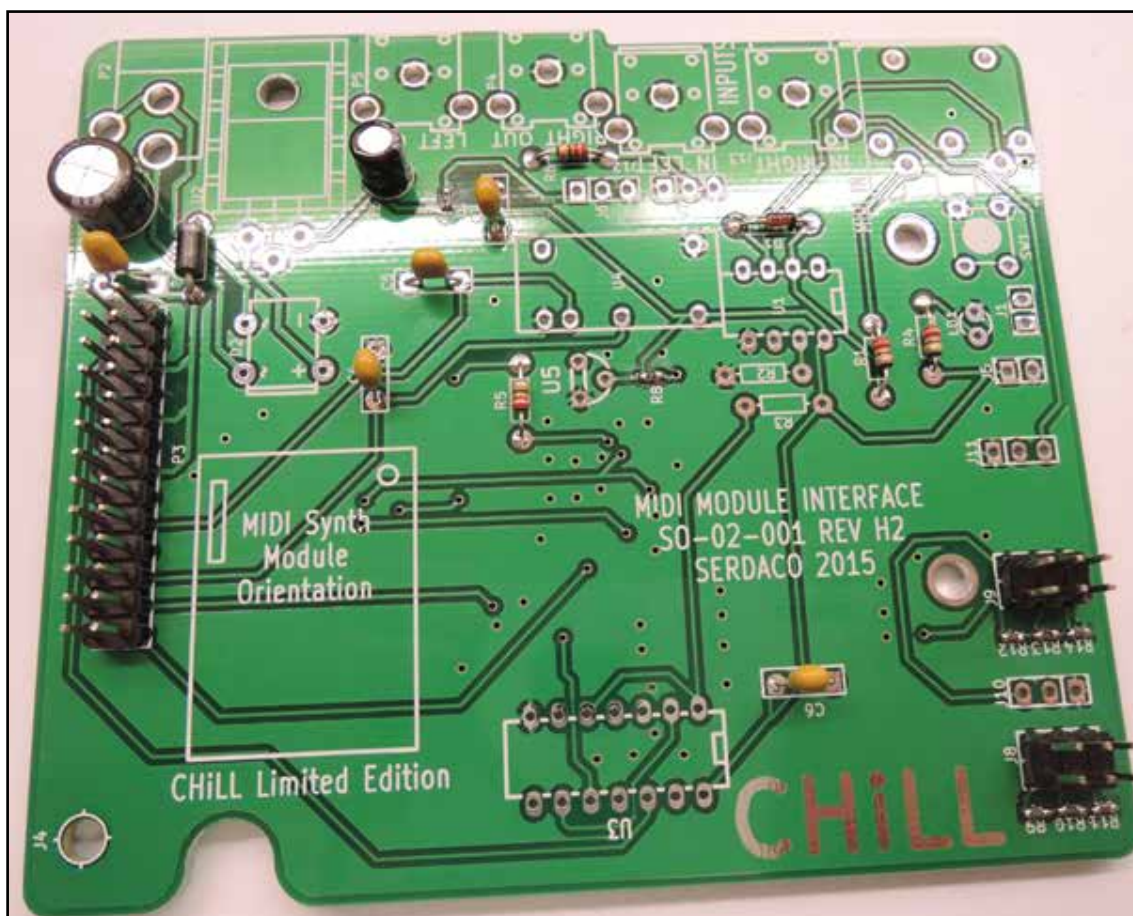
Colour code:

- in case of 3 band resistors : RED RED BROWN GOLD



- in case of 4 band resistors : RED RED BLACK BLACK BROWN





Now solder the remaining 2 resistors :

R3: 270 Ohm Resistor :

- Colour code: First 3 bands are RED VIOLET BROWN

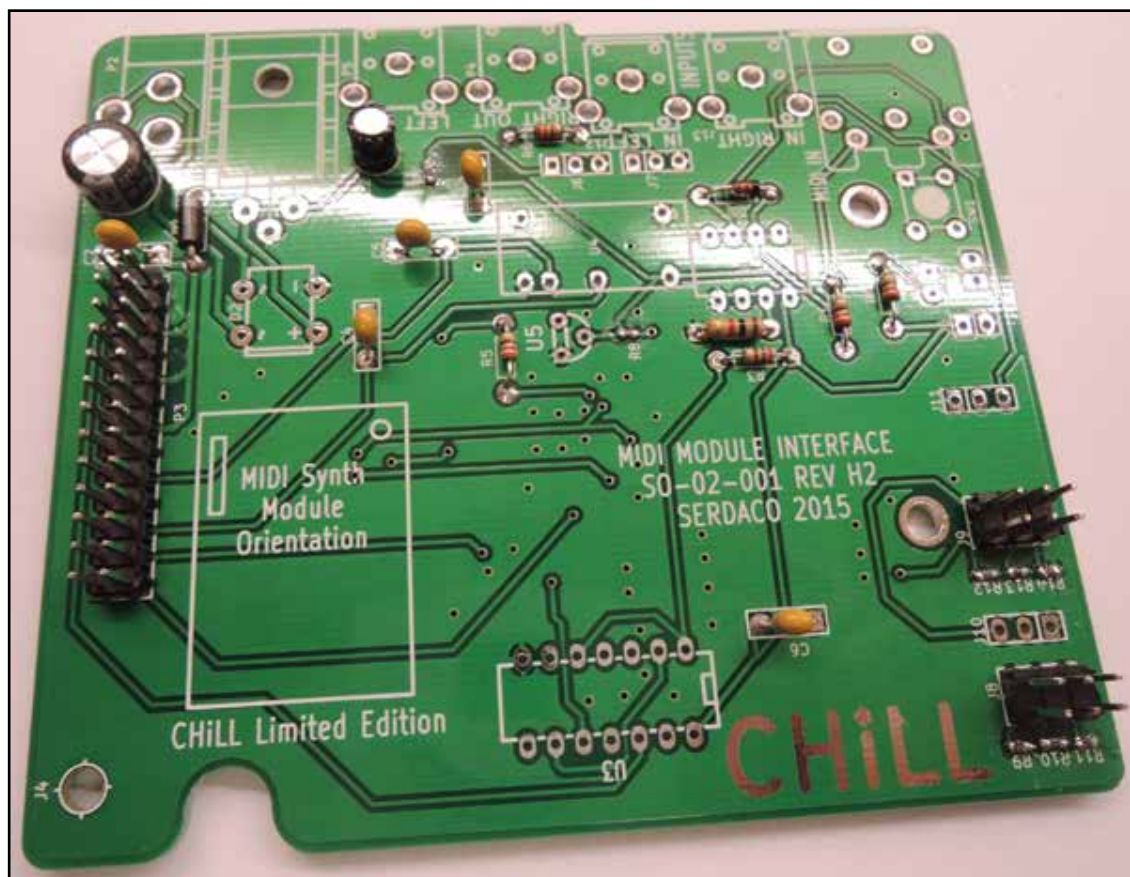


R2: 1KOhm Resistor

- Colour code : First 3 bands are BROWN BLACK RED



All resistors are now placed:



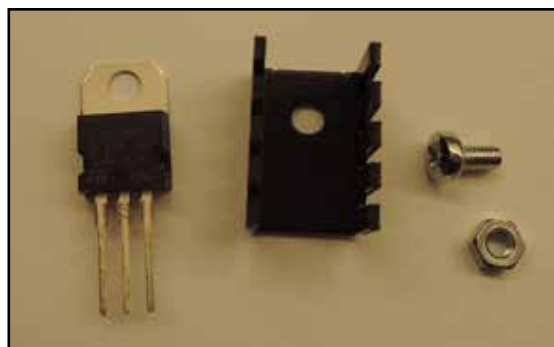
Regulator - 5V

Now we add the parts for the 5V regulator :

P2: power connector:

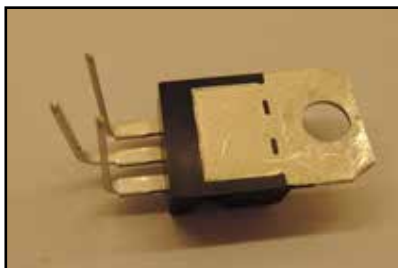


U2 : 7805 regulator + heat sink + nut and bolt:

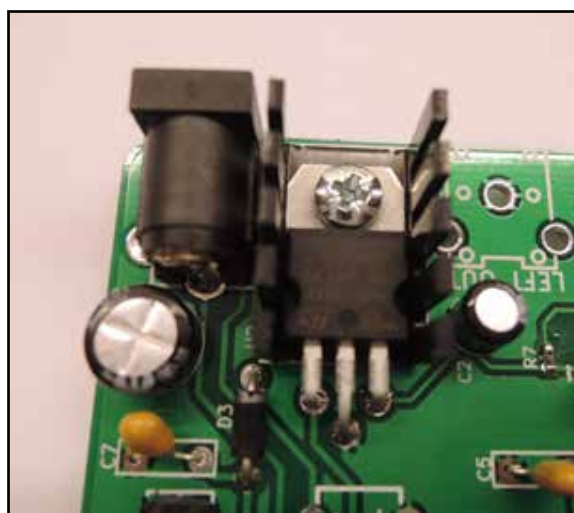


Bend the legs of the regulator as in the photo.

If you happen to have thermal paste, you can add some thermal paste between the heatsink and the regulator.



And then mount + solder everything on the pcb:

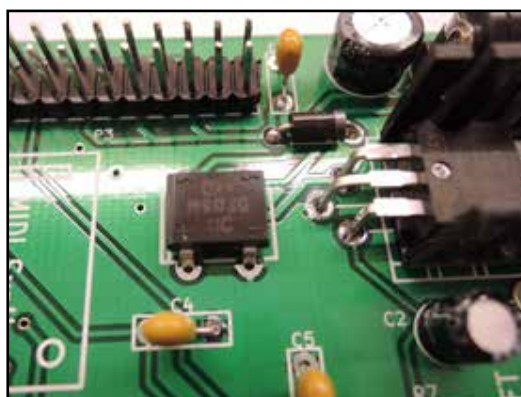


Bridge Rectifier

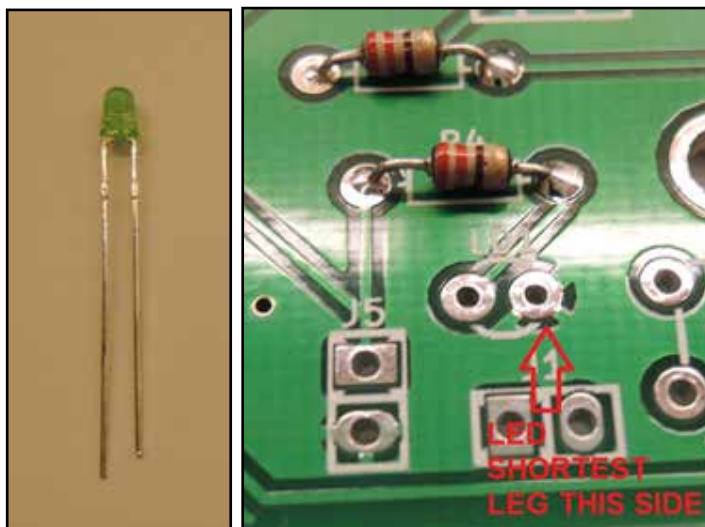
Now add the bridge rectifier.

D2: bridge rectifier

Make sure the orientation is correct :



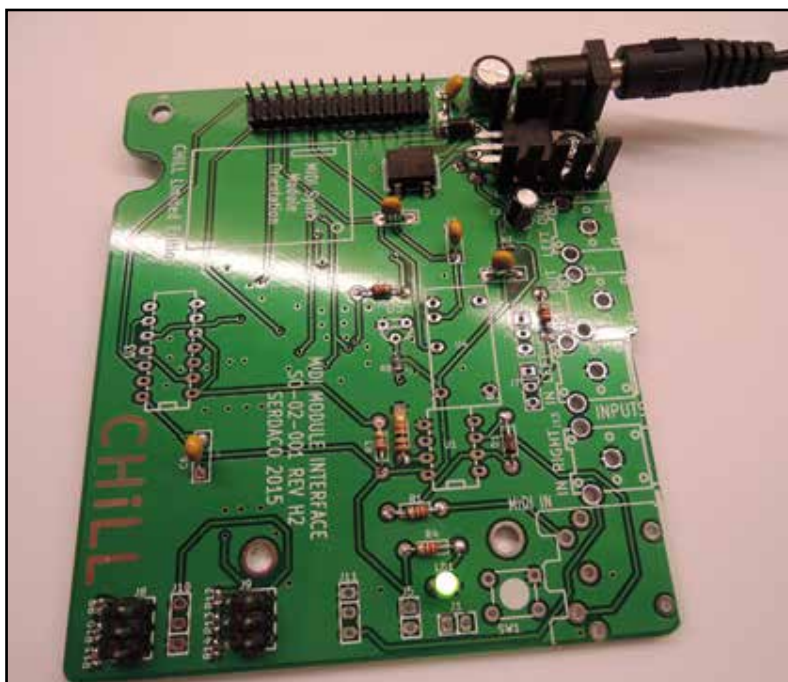
LD1 : green led : The short leg needs to be connected to ground.



At this point you can verify that the power part is correct:

- If you insert a 9 Volt DC plug in the power inlet, you will see the green led lit up.
- Over the pins of J5 you can measure a regulated 5 Volt.

Now we know the voltage supply is OK.



Miscellaneous

We can now add the TTL logic chips, DC/DC regulator, and the Audio + MIDI connectors :

Make sure you get the orientation of the chips right (see overview at the end).

U3: Hex Schmitt Inverted Buffer Chip 7414



U4: DC/DC regulator : 5V to +12V and -12V



U1: 6N138 Optocoupler



Audio Line Out Connectors

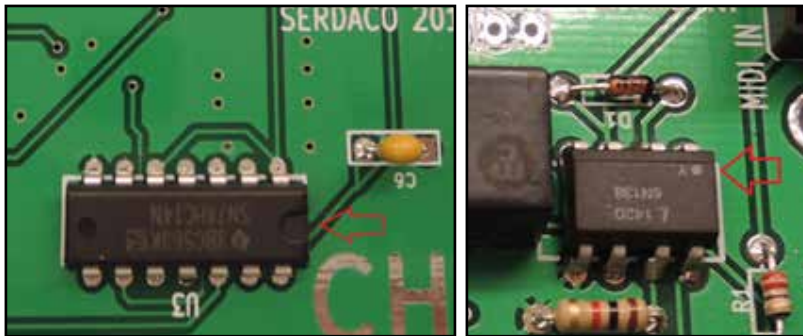
- P4 : Right Line Out = red RCA connector
 - P5 : Left Line Out = white or black RCA connector
- Option : J12 and J13 : RCA connectors for XR385 input



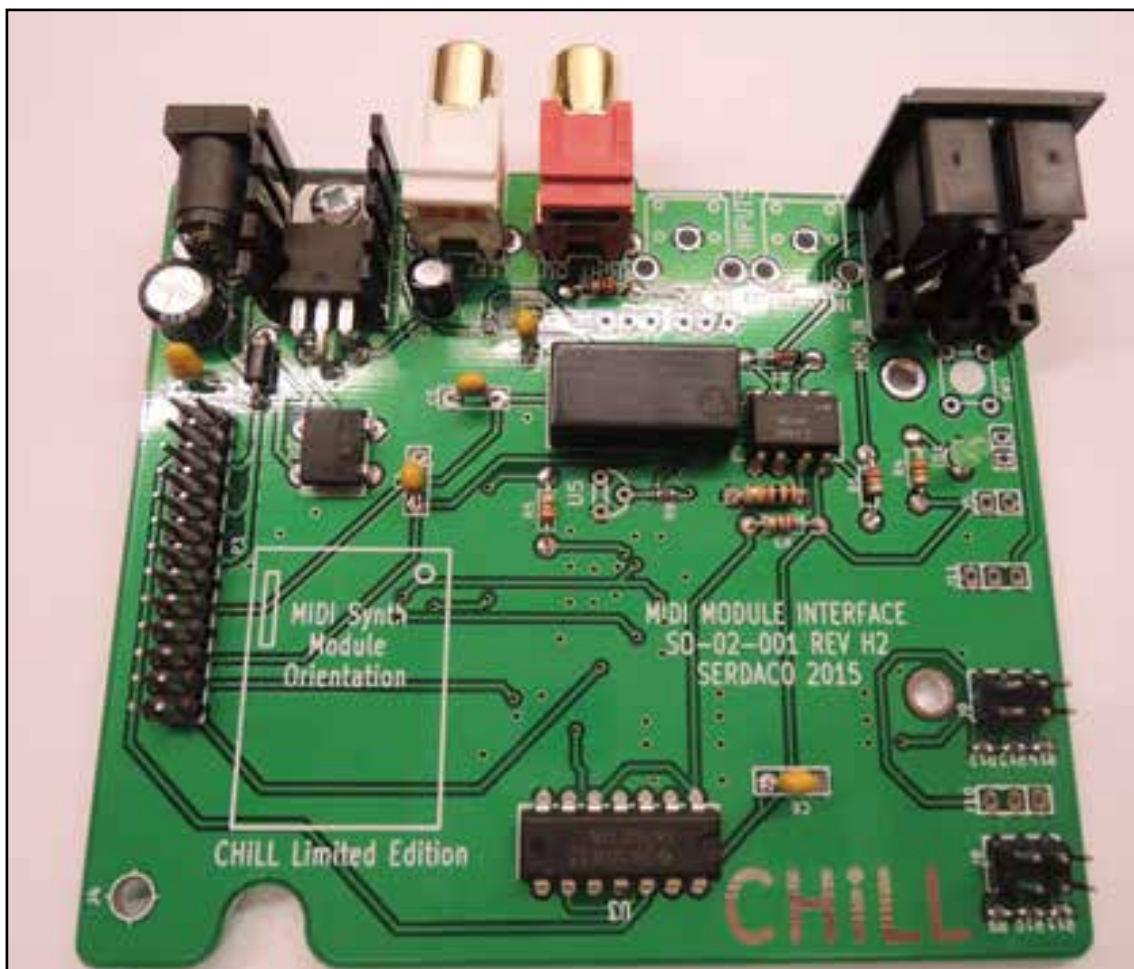
MIDI IN Connector



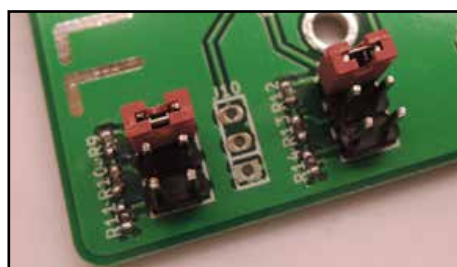
Please note of the chip orientations before soldering!



Overview of all parts mounted.



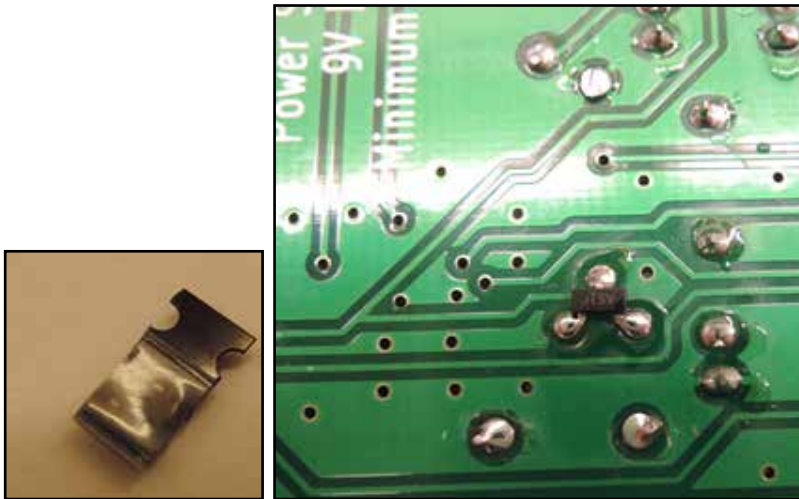
Now place the two jumpers on the volume settings J8 and J9.



Reset Chip

The last part is the reset chip.

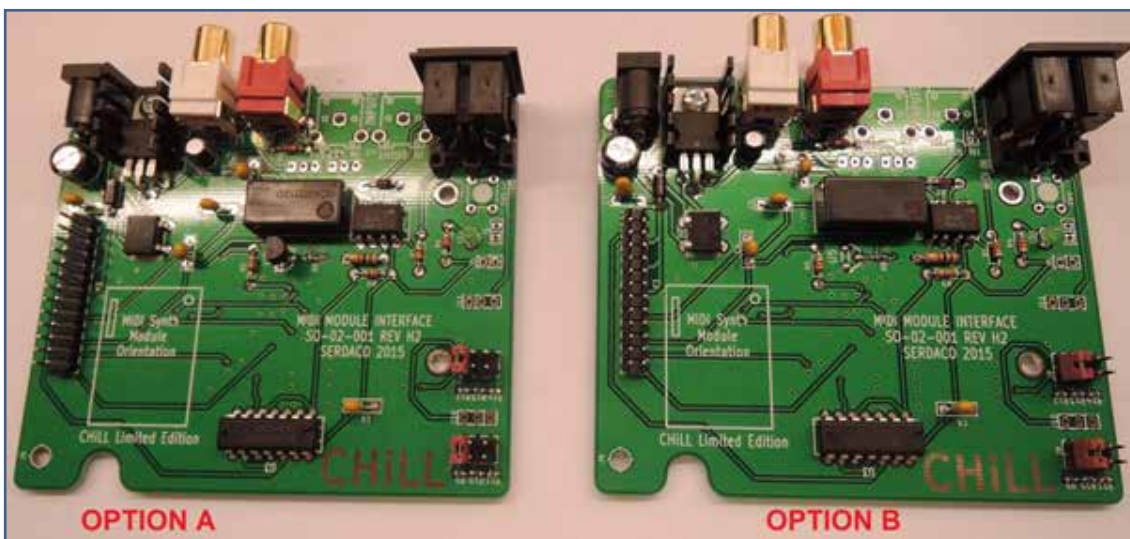
Add the SMD Reset chip to the back side of the board : solder it to the pins of U5 at the BACK side of the pcb, exactly as in the photo :



This ensures compatibility with the Roland SCB-7. Although, it is not compatible with the Avance PRO32AW.

Assembled board

The final result is a completed CHILL MIDI interface PCB :

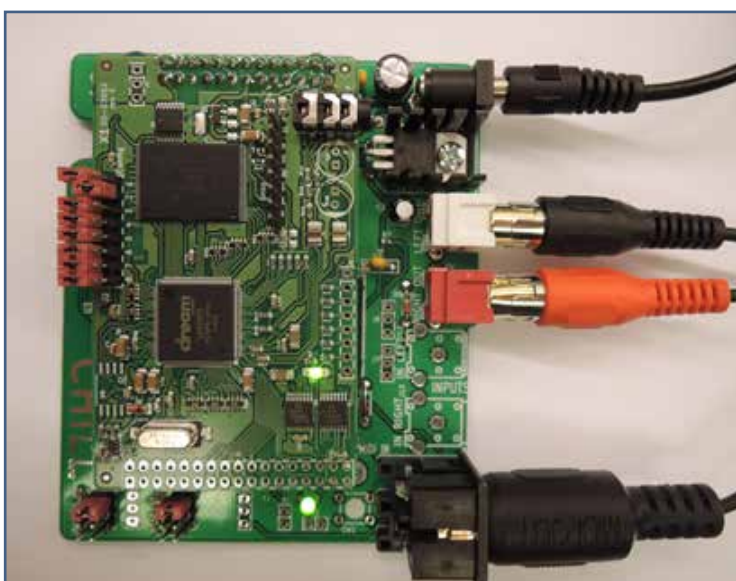
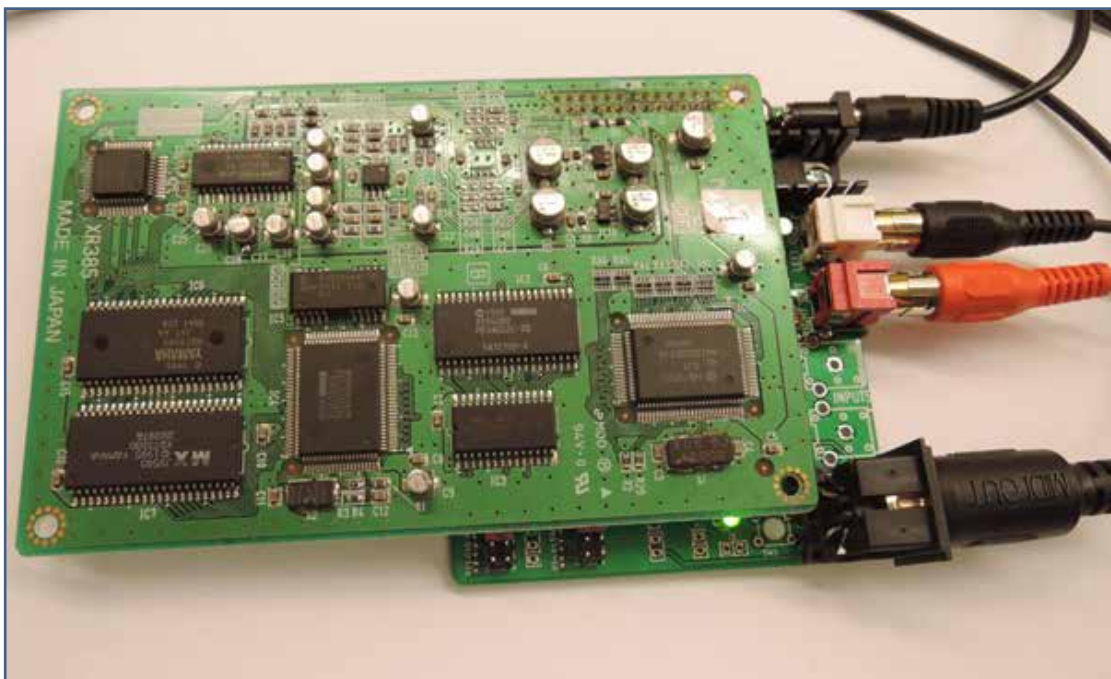


Now you can start using your CHiLL board. Plug on a waveblaster:
Plug on a compatible board such as the:

- NEC XR385
- Dreamblaster X2 - [<http://www.excelvalley.com/product/dreamblaster-x21/>]

Plug in

- the MIDI cable
- the left and right line level outputs
- the DC 9V jack.



Enjoy the music!