# Irukandji Glitch Drum V2/3

# BUILD GUIDE

**www.beast-tek.com**


## Bill of Materials

| Irukandji Glitch Drum IO Board BOM | | | |
|---|---|---|---|
| D1, D2, D3 | 7 Segment Display 0.56" (common catchode) | | 3 |
| LED1 | 3mm LED | | 1 |
| **Version 3** - R1,R2, R3, R4 | 220 Ohm Resistor 1% | Red-Red-Black-Black-Brown | 4 |
| **Version 2** - R1,R2, R3, R4 | 180 Ohm Resistor 1% | Brown-Grey-Black-Black-Brown | 4 |
| C1 | 10uf Electrolytic Capacitor | 10uf | 1 |
| R5, R6, R7, R8, R9, R10 | 1K Ohm Resistor 1% | Brown-Brown-Black-Black-Brown | 6 |
| R11 | 10K Ohm Resistor 1% | Brown-Red-Black-Black-Brown | 1 |
| D4,D5, D6, D7, D8, D9, D10, D11 | BAT85 Diode | | 8 |
| S1, S2, S3 | SPST Switch | | 3 |
| SW1 | ALPS Rotary Encoder | | 1 |
| JP1 | 24 Way Pin Header Single Row MALE | | 1 |
| JP2 | 4 Way Pin Header Single Row MALE | | 1 |
| PA, PB, PC, PD | 5K Ohm 16mm Right Angle Potentiometer | | 4 |
| | 3.5mm Mono Socket | | 5 |

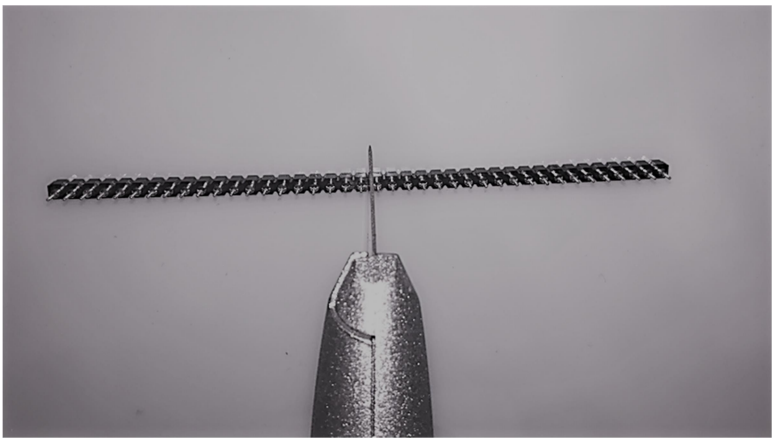
## Bill of Materials

| Irukandji Glitch Drum Board BOM | | | |
|---|---|---|---|
| IC1 | ATMEGA1284P | | 1 |
| IC2 | TL074 OPAMP | TL074 | 1 |
| IC3 | 7805 5v 1A Voltage Regulator | 7805 | 1 |
| D1, D2, D3 | IN4004 Power Diode | IN4004 | 3 |
| Q1 | 20mhz Crystal | | 1 |
| C1, C19 | 100uf Electrolytic Capacitor | 100uf | 2 |
| C4, C5 | 22pf Ceramic Capacitor | 22 | 2 |
| C11, C12 | 220nf Green Cap | 224 | 2 |
| C13, C14 | 4.7uf Electrolytic Capacitor | 4.7uf | 2 |
| C15 | 10uf Electrolytic Capacitor | 10uf | 1 |
| C17 | V2 - 10nf Ceramic Capacitor<br>V3 – Not required / Do not populate | 103 | 1 |
| C22 | 470uf Electrolytic Capacitor | 470uf | 1 |
| C2, C3, C8,C18, C20, C21, C23 | 100nf Blue Monolithic Capacitor | 104 | 7 |
| C6, C7, C9, C10, C16 | 220nf MKT Capacitor | 224 | 5 |
| R1, R8, R9, R10, R11, R13, R14 | 10K Ohm Resistor 1% | Brown-Red-Black-Black-Brown | 7 |
| **Version 3 – R2** | 39 Ohm Resistor 1% | Orange-White-Black-Gold-Brown | 1 |
| **Version 2 – R2** | 75 Ohm Resistor 1% | Violet-Green-Black-Gold-Brown | 1 |
| R3 | 75 Ohm Resistor 1% | Violet-Green-Black-Gold-Brown | 1 |
| R4, R5, R6, R7, R12 | 330 Ohm Resistor 1% | Orange-Orange-Black-Black-Brown | 5 |
| R15 | 1K Ohm Resistor 1% | Brown-Brown-Black-Black-Brown | 1 |
| JP1 | 4 Way Pin Header Single Row FEMALE | | 1 |
| JP2 | ICSP - do not populate | | 1 |
| JP3 | 24 Way Pin Header Single Row FEMALE | | 1 |
| JP4 | Shrouded 10pin (2x5) IDC Header (Eurorack Power) | | 1 |
| | 20 pin Machine pin IC socket strip | | 2 |
| | 7 pin Machine pin IC socket strip | | 2 |

# Main PCB / “Motherboard”

## Step 1

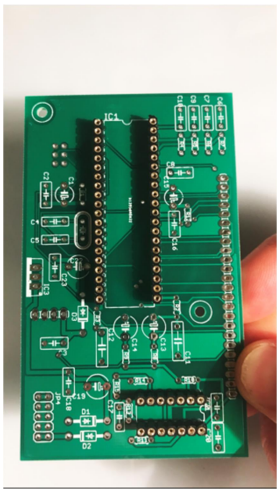
Take the 40 pin machine pin socket strips and cut one in half (2 pieces 20 pins each) using a sharp utility knife or side cutters. Also cut 2 x 7 pin pieces from the remaining piece. Tip: You can measure the cut using the PCB as a guide.

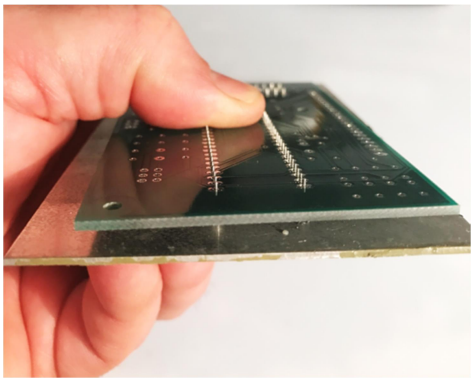


## Step 2

Solder the machine pin strips IC into place for IC1 and IC2. Tip: This step can be made a lot easier using a small piece of scrap metal or rigid cardboard slightly larger than the PCB.

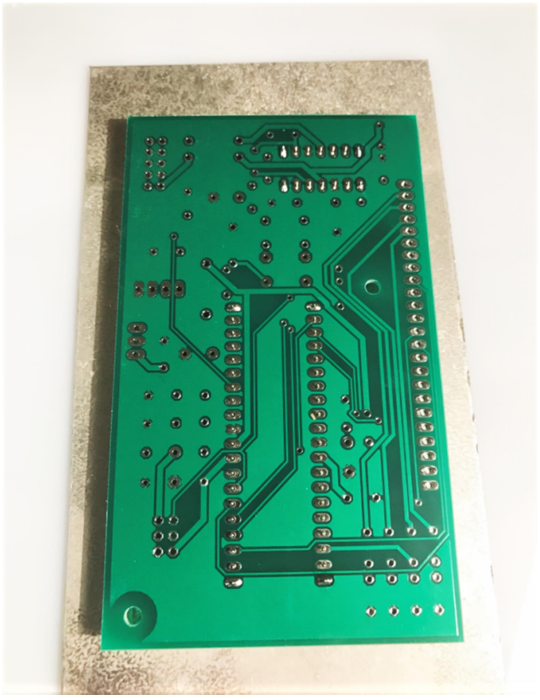
First insert the machine pin strips into place, then sitting the metal/cardboard on top and applying a small amount of pressure, flip the board over so the pins can be soldered in place.

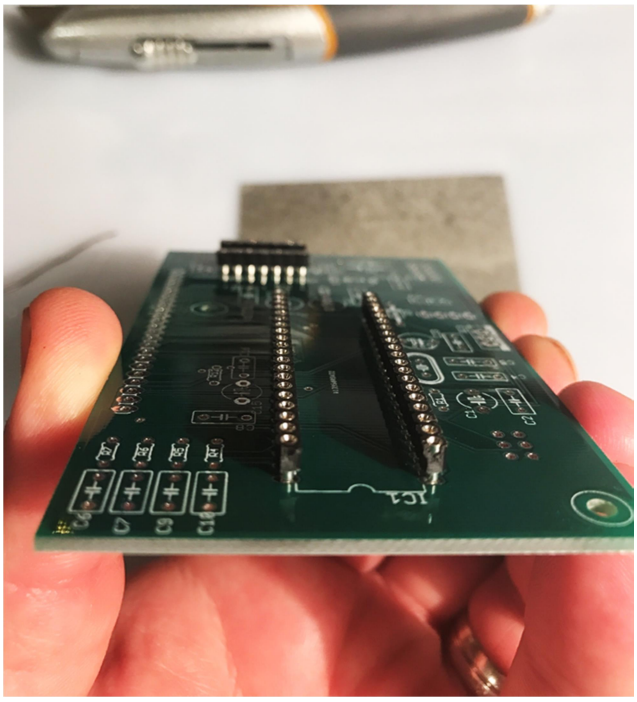




**Tip:** Solder only 2 pins of each socket at first to hold them into place. Now flip the board over to make sure the socket strips are sitting at right angles to the PCB. If required, they can be adjusted easily by heating the 2 joints one at a time and maneuvering the socket into place.

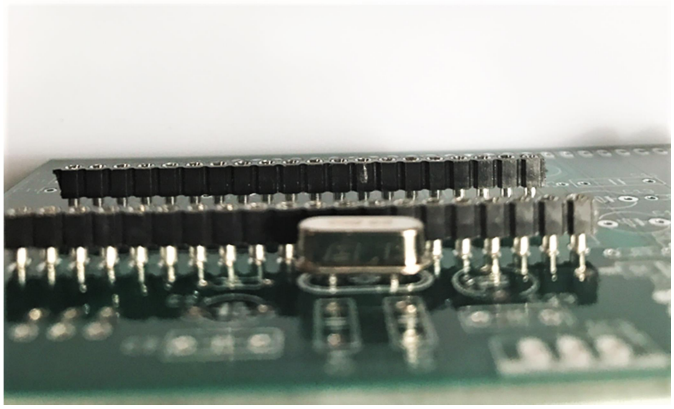
When the strip is seated correctly, continue.





## Step 3

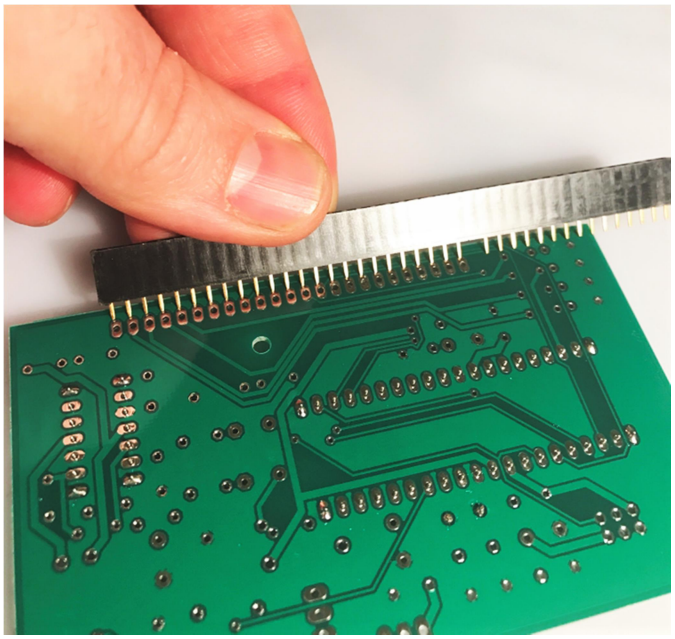
Now is the perfect time to solder the 20mhz crystal into place. The crystal should not sit in contact with the PCB. Soldering the crystal now means the machine pins can be used as a guide so that the crystal will sit approx. 1mm from the board.



Now solder the remaining IC pins and crystal.

## Step 4

Cut the female pin header down into 1 x 24 piece and 1 x 4 piece. Tip: once again the PCB can be used as a guide, use a small pair of pliers to remove the 25th pin so the blank space can be used as a cutting guide. Now cut the pin header socket using side cutters, if desired then use a file to clean up the edges.

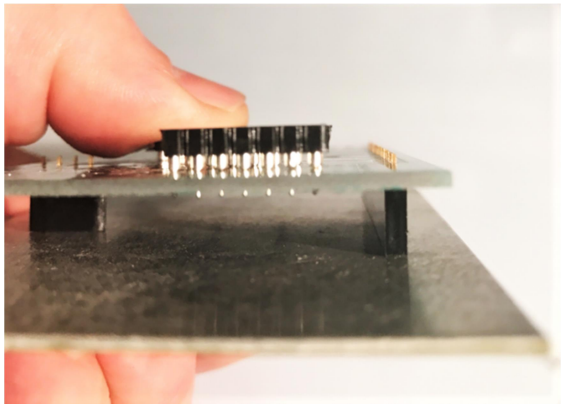


## Step 5

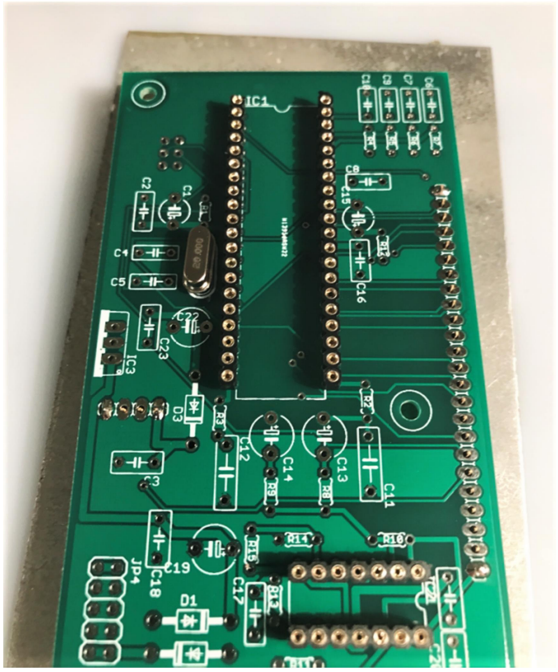
IMPORTANT!!! The female pin header strips must be installed on the REVERSE (non silk screened side).

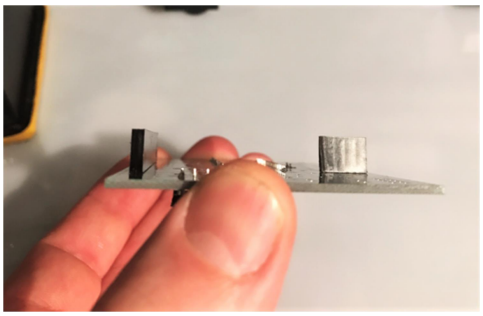
Install the pin headers into their locations (JP1, JP4) on the reverse side of the board.

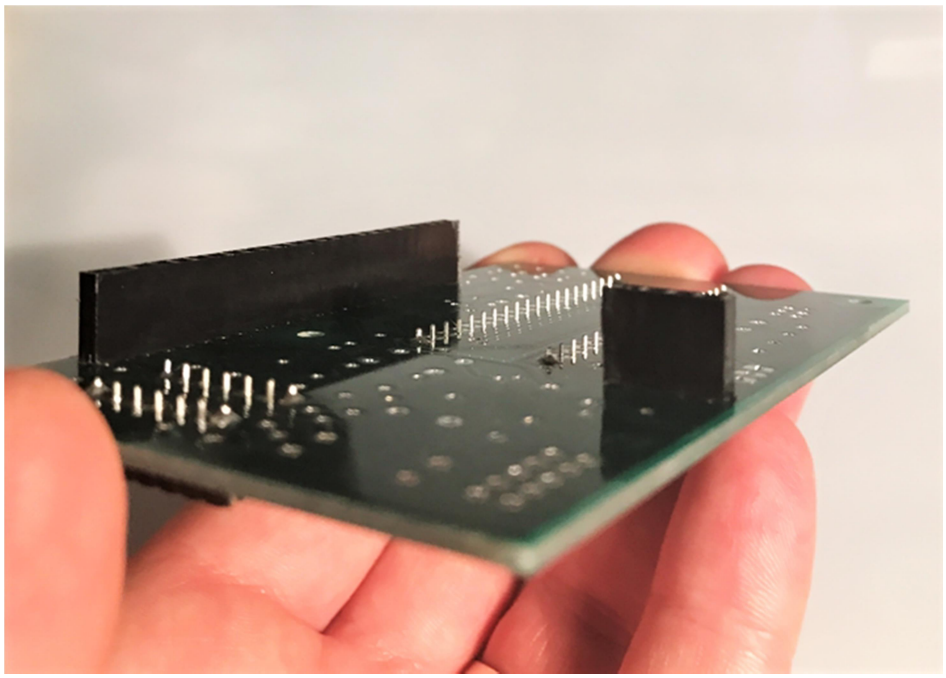
Once again use a scrap piece of metal/cardboard so that the board can be flipped over for soldering without the pieces falling out.



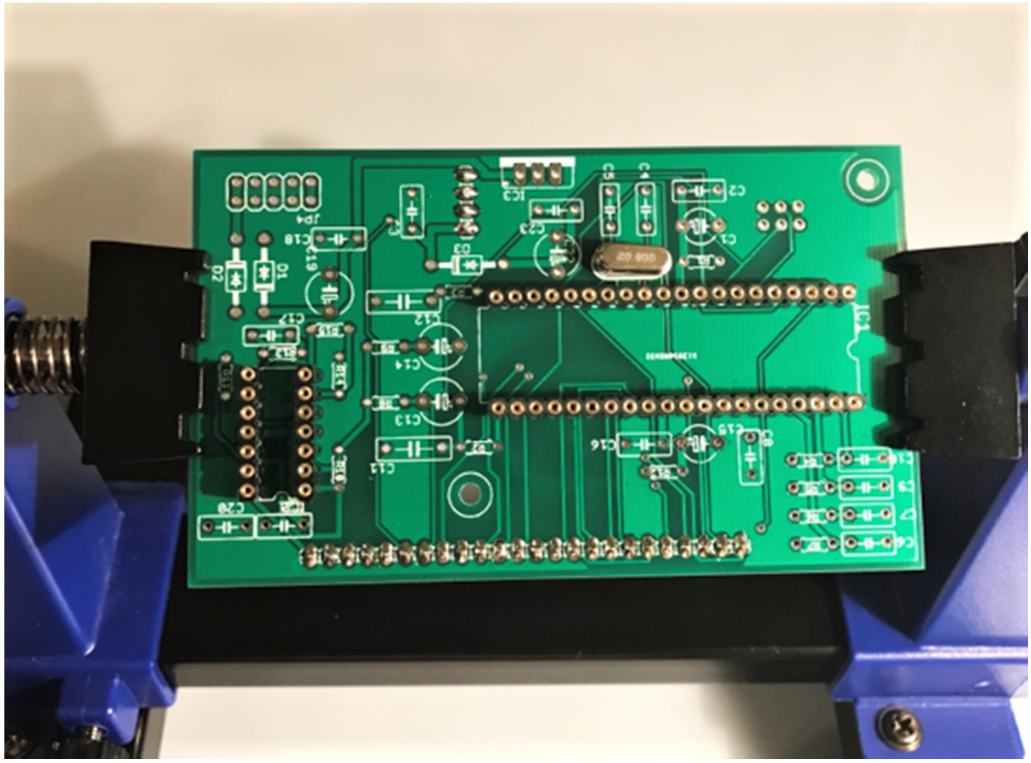
Only solder 2 pins on each strip, then turn the board over to inspect and make sure the header strips are seated at a right angle and that they are flush with the PCB board. If required, the header strip can be adjusted by heating one joint at a time and gently moving the header strip into the correct location.

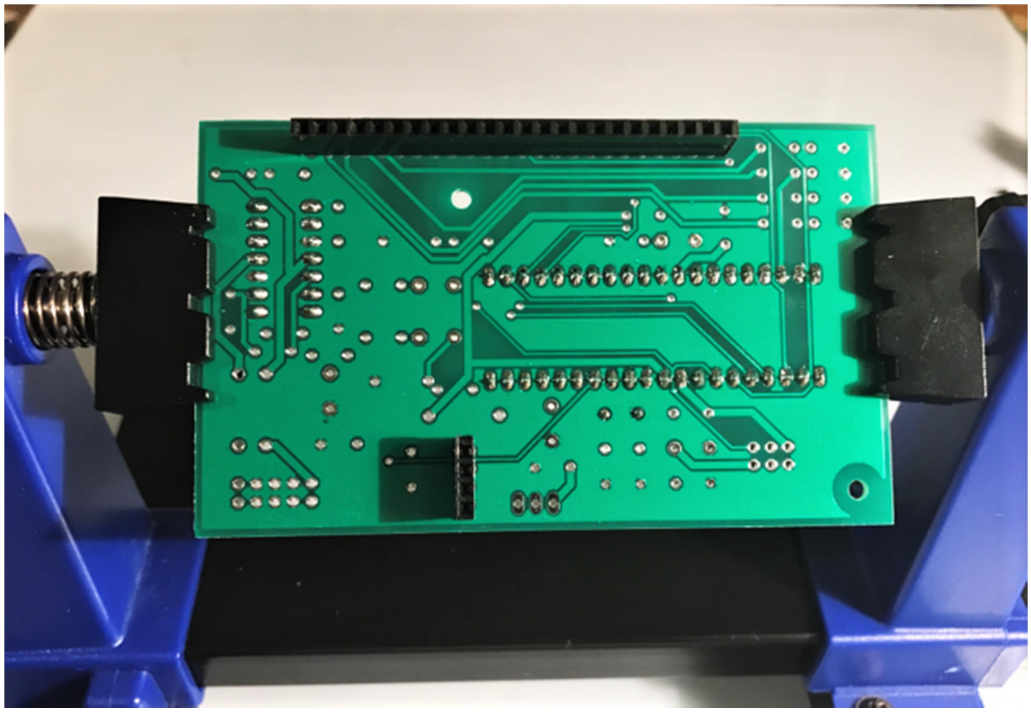




Once seated correctly, solder the remaining header strip pins.

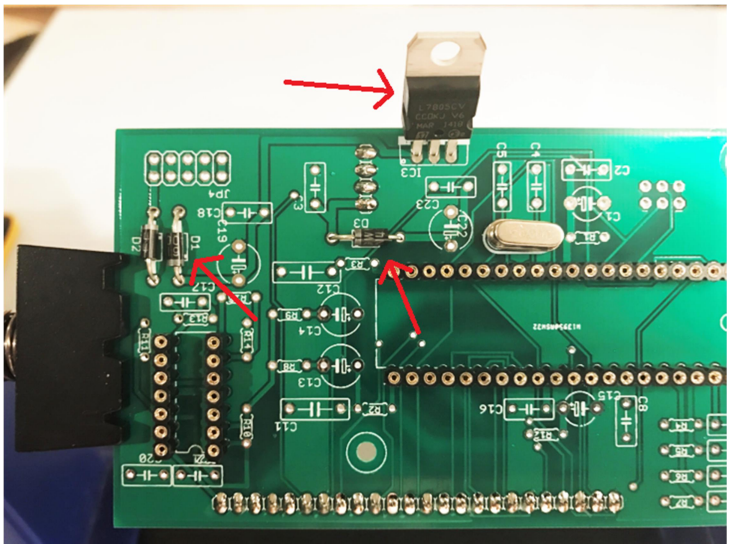




## Step 5

Install the 7805 voltage regulator (IC3) into place (refer to photos below to ensure correct orientation if unsure), take it slowly and give the component enough time to cool down between soldering each pin.

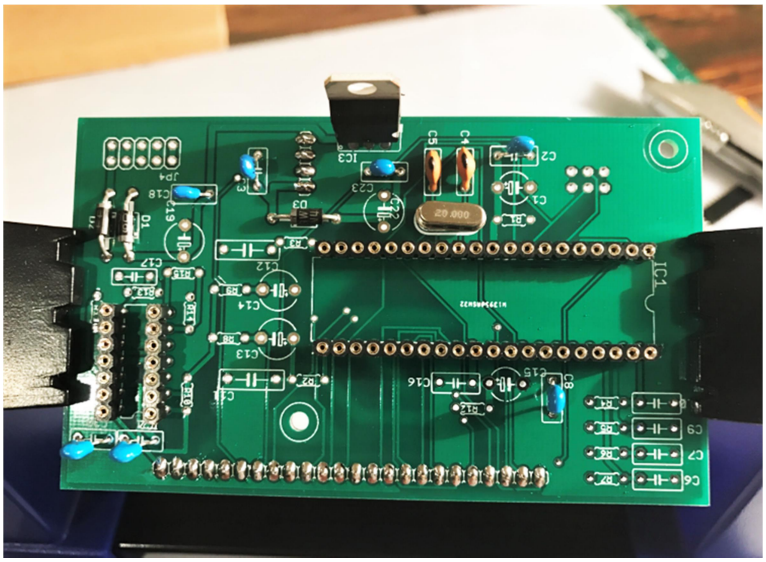
Also install the 3 x IN4004 power diodes (D1, D2, D3). Make sure the polarity of the diodes is correct - the stripes on the diode should match the stripes on the silk screen. Refer to the photos below if uncertain.



Once soldered, trim off the component legs using side cutters.

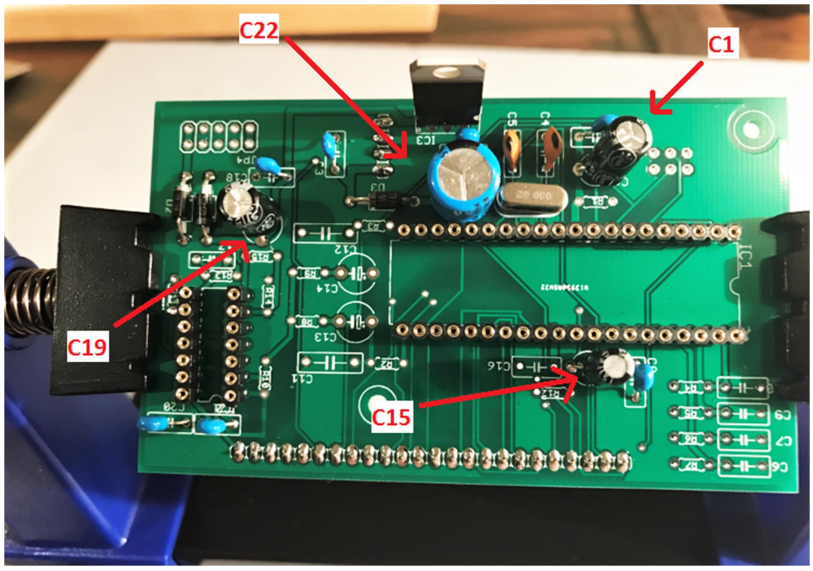
## Step 6

Install and solder the 2 x 22pf ceramic capacitors (C4, C5) and the 7 x 100nf blue monolithic capacitors (C2, C3, C8, C18, C20, C21, C23).



## Step 7

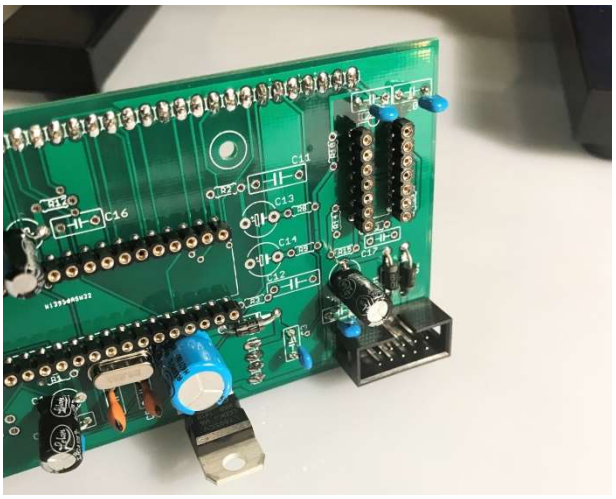
Install and solder the 2 x 100uf electrolytic capacitors (C1, C19) followed by the 10uf electrolytic capacitor (C15) and the 470uf electrolytic capacitor (C22).

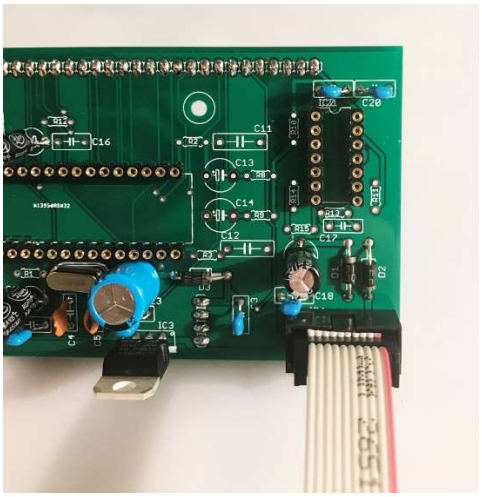



## Step 8

IMPORTANT!! The orientation of the shrouded IDC power header is critical. Use the photos and steps below to ensure it is oriented correctly.

Install the 10 pin shrouded IDC eurorack power socket (JP3). Use the photo below to ensure correct orientation. Solder one of the pins to hold in it place. Now use the supplied power cable to ensure correct orientation. If installed correctly, the red strip on the cable should be facing toward the bottom of the board.





Once the IDC socket orientation has been confirmed using the power cable, solder the remaining pins.

## Step 9

Time for some testing! Double check all solder joints. When you are happy with your work, connect the power cable to a eurorack power supply so that test measurements can be made with a multimeter.

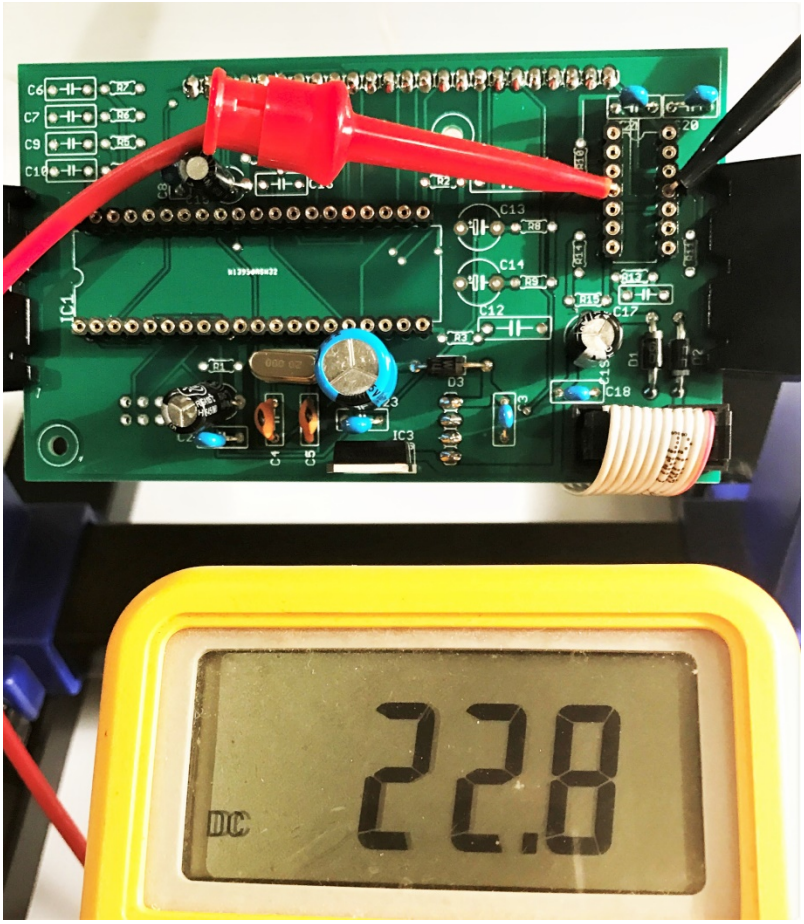
**IMPORTANT:** *Do NOT do this in a case with other modules installed*. If there is a fault with your work, it could fry/burn/destroy other modules connected to the same power system.

Using a multimeter set to DC volts with a range capable of measuring 5v – 24v, test the voltages at the following points:

IC2 – connect the positive probe to pin 4 and the negative probe to pin 11.

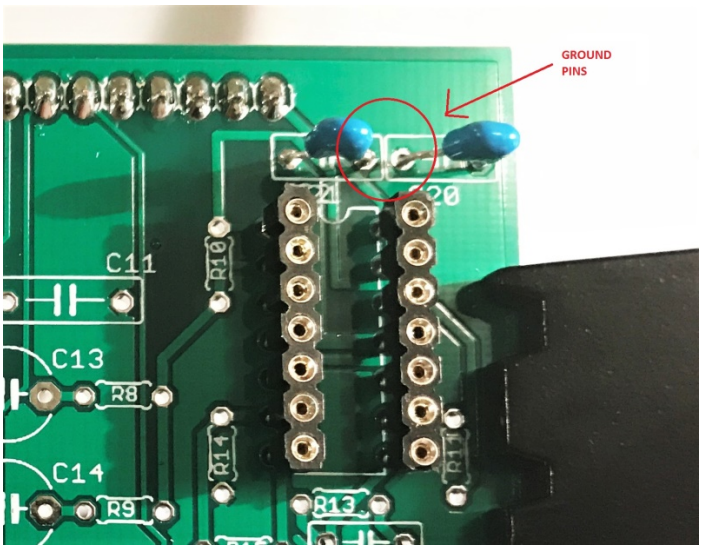
You should get a reading of between 23v to 24v .. the power diodes used to protect the module against reverse power connection do cause a small voltage drop of approx. 0.4v

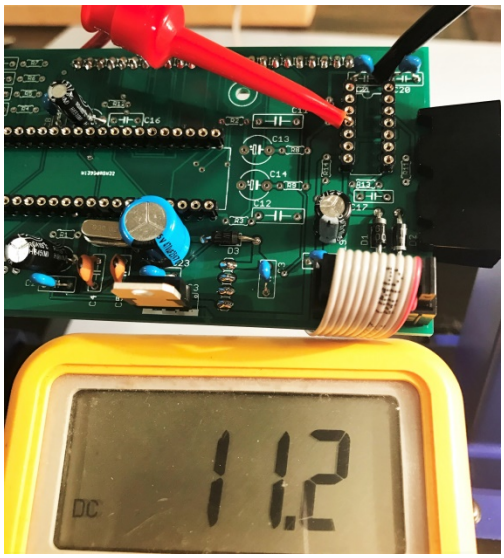
In the photo below, my reading is 22.8v because I am using a 4ft power cable (which causes significant voltage drop) so that I can assemble and take photos easily.



IC2 – connect the positive probe to pin 4 and connect the negative probe to one of the ground pins on C20 or C21 located directly above IC2.

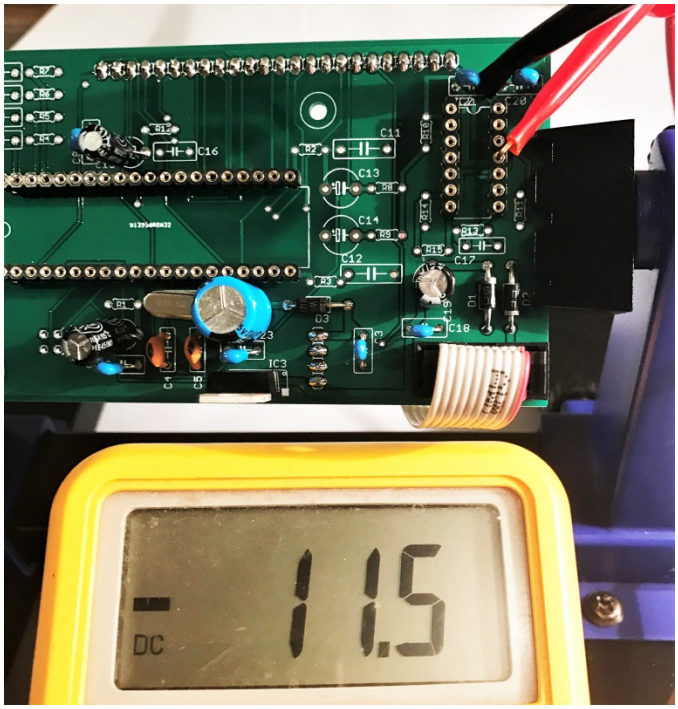
You should get a reading of around 11v to 12v .. once again the diodes will cause a voltage drop of approx. 0.2v and my measurements below are out by 0.8v because of the gigantic eurorack power cable.




IC2 – connect the positive probe to pin 11 and connect the negative probe to one of the ground pins on C20 or C21 located directly above IC2.

You should get a reading of around -11v to -12v.



If these voltages do not check out, double check all solder joints and check the orientation of the voltage regulator (IC3) and the power diodes.

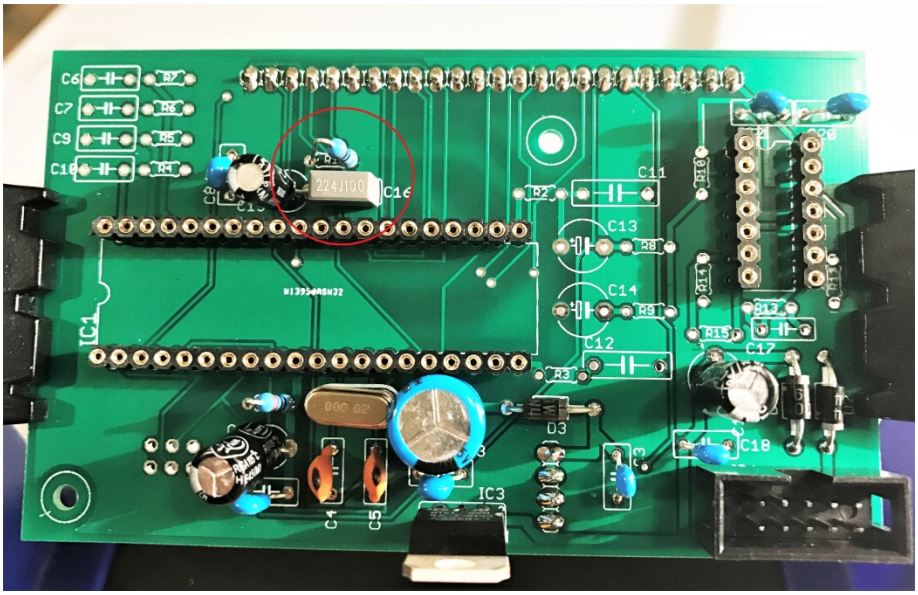
If all the voltages check out, you are fine to continue on!

***IMPORTANT:*** **Make sure you disconnect the eurorack power cable before continuing work on the PCB.**

## Step 10

***IMPORTANT:*** **Make sure you disconnect the eurorack power cable before continuing work on the PCB.**

Install the 330ohm resistor R12 and the 220nf MKT capacitor C16. Once soldered, trim the leads.




## Step 11

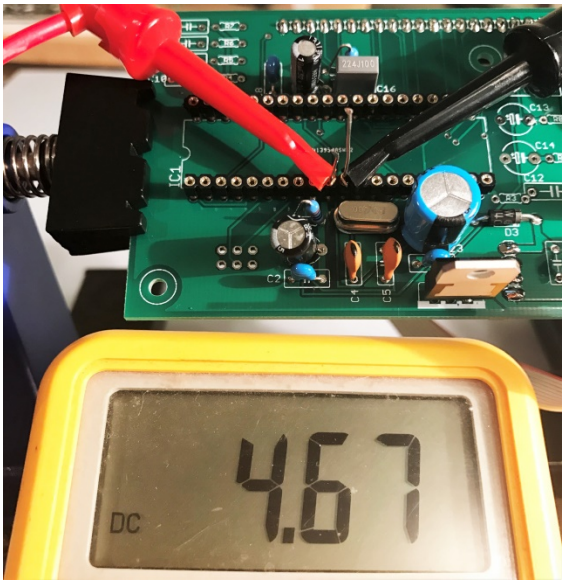
Install and solder the 10K resistor R1 located on the left hand side of the crystal in the photo above.

Now .. Time for some more tests!

This time we will be checking the 5v supply voltage for the ATMega1284p microcontroller.

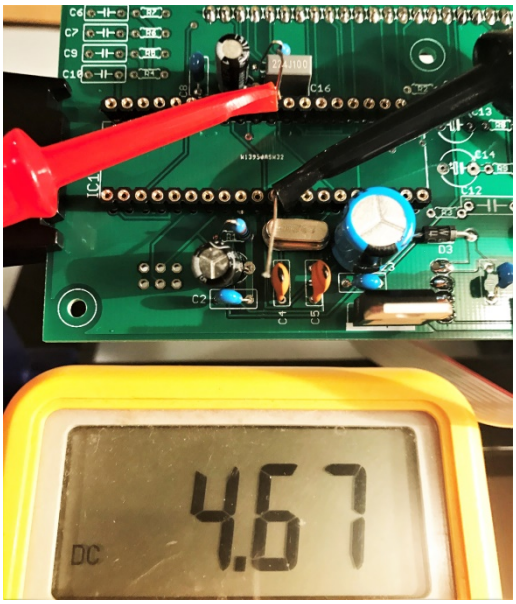
Connect the eurorack power supply cable again as per step 9.

Connect the negative probe from a multimeter to pin 11 of IC1. Connect the positive probe to pin 10. The reading should be somewhere around 4.7v – 5.0v.




Connect the negative probe from a multimeter to pin 11 of IC1. Connect the positive probe to pin 9. The reading should be somewhere around 4.75v – 5.0v.

Connect the negative probe from a multimeter to pin 11 of IC1. Connect the positive probe to pin 30. The reading should be somewhere around 4.75v – 5.0v.




If these voltages do not check out, double check all solder joints and check the orientation of the voltage regulator (IC3) and the power diodes.

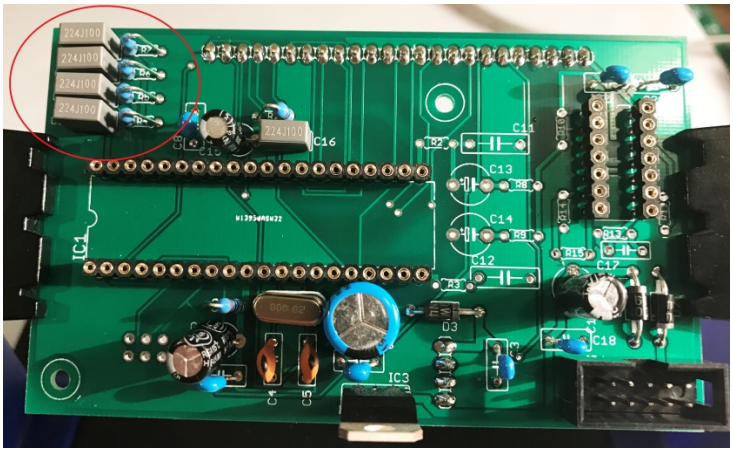
If all the voltages check out, you are fine to continue on!

**IMPORTANT: Make sure you disconnect the eurorack power cable before continuing work on the PCB.**

## Step 12

**IMPORTANT: Make sure you disconnect the eurorack power cable before continuing work on the PCB.**

Install 220nf MKT capacitors C6, C7, C9, C10. Install 330 ohm resistors R4, R5, R6, R7. Solder in place then trim leads.

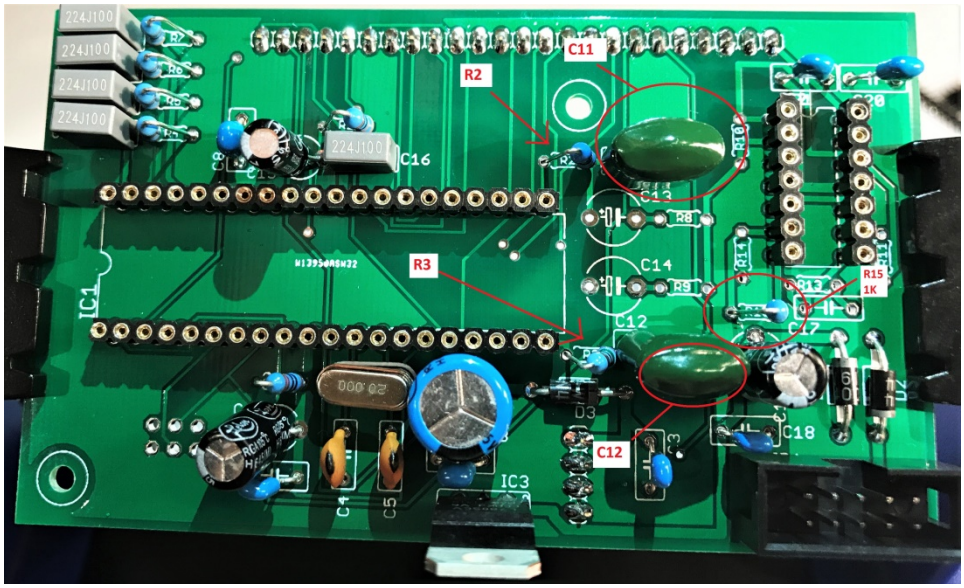



## Step 13

Next install the two 220nf green cap capacitors C11 and C12. Install 1K ohm resistor R15. Solder and trim leads.

**VERSION 2:** Install and solder 39 ohm resistor R2 and 75 ohm resistor R3.

**VERSION 3:** Install and solder the two 75 ohm resistors R2 and R3.



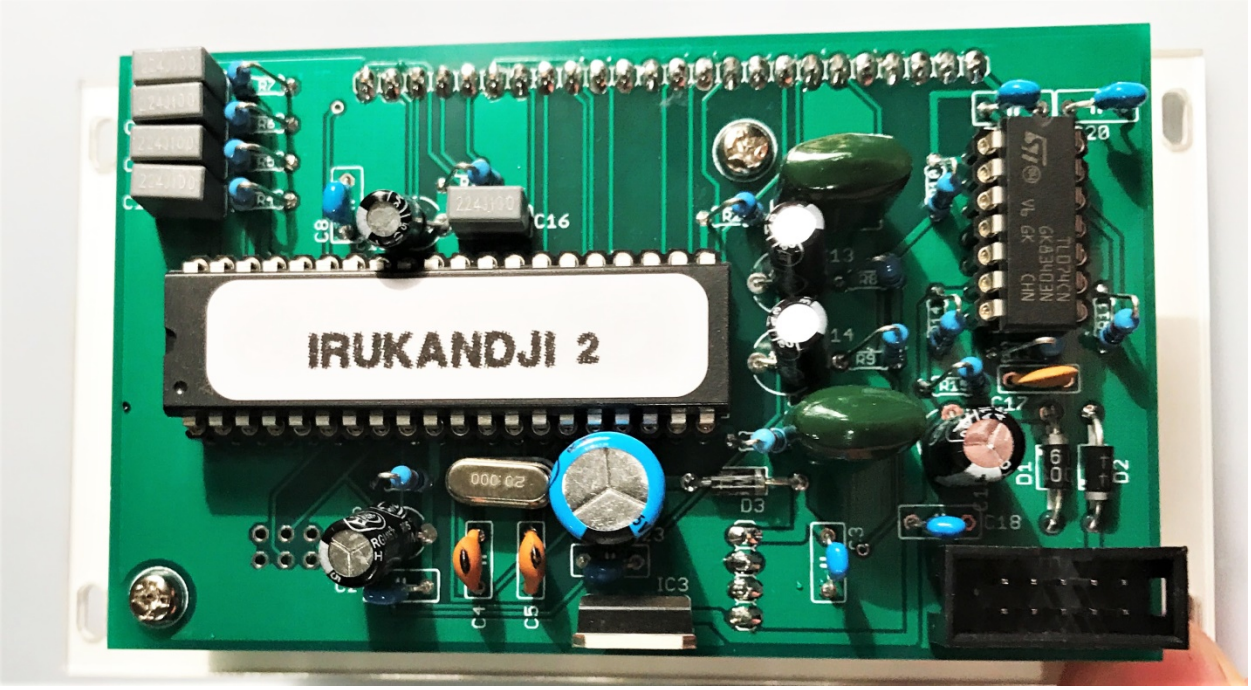
## Step 14

Install the two 4.7uf electrolytic capacitors C13 and C14. Install the six 10K ohm resistors R8, R9, R10, R11, R13 and R14. Solder and trim leads.

# == VERSION 2 ONLY ==

Next install 10nf ceramic capacitor C17 (code 103). Solder and trim leads.

**Note: C17 is NOT required for V3 firmware and should be ignored or removed if you have a V3 or above firmware.**



## Step 15

## CAUTION: IC1 and IC2 are sensitive to static discharge (ESD)

Install IC1 and IC2. Take your time to make sure you do not bend any pins. Hold the IC at a 45 degree angle to the PCB, PARTIALLY insert one row of pins and make sure they are all lined up correctly. Gently apply even force from the side that is not inserted so that the inserted pins bend slightly so that the non -inserted side lines up with its socket. Once lined up, apply a small amount of downward force so that the IC is partially inserted/seated. Double check that all pins are inside their sockets and no pins have bent outward or inward. Once you are happy everything is lined up, apply force so that the IC is fully inserted into its socket.

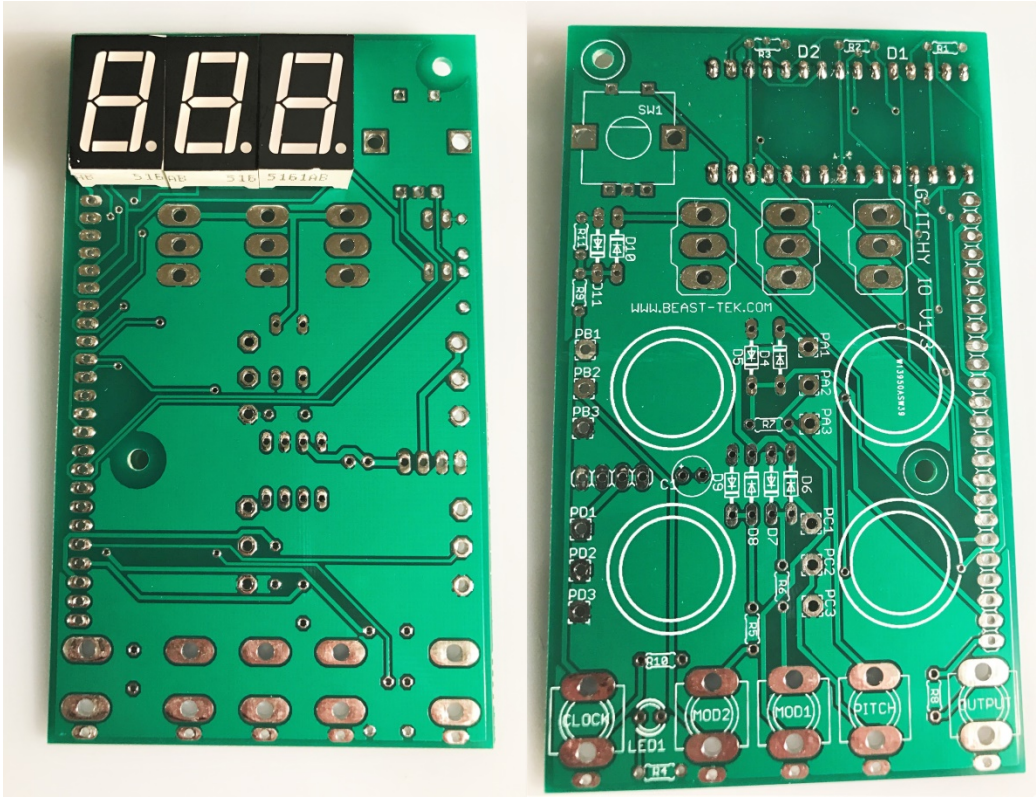
Congratulations, you have finished the main board!! Turn the board over, double check all solder joints, trim any excessive leads and inspect for missing solder joints.

# Interface Board

**IMPORTANT!!** Note: all components facing the front of the module are to be mounted on the reverse side / back / side WITHOUT the silk screen. Please refer to the photos within the build guide.

## Step 1

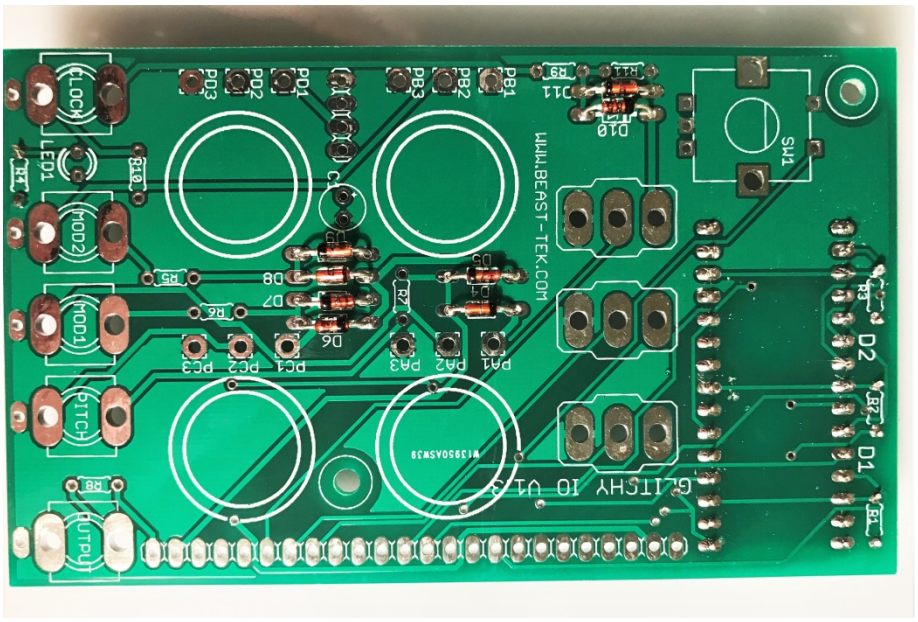
Peel the protective film from the 7 segment displays D1, D2 and D3. Insert them onto the BACK of the PCB (side without silk screen). Double check that they are oriented correctly using the decimal points on the display as a guide.




Once the orientation has been confirmed, solder one lead from each display and check that they sit flush with the board. If they are not flush, they can be adjusted easily by heating the single solder joint and maneuvering them into place. Once you are happy, solder the rest of the leads and trim.

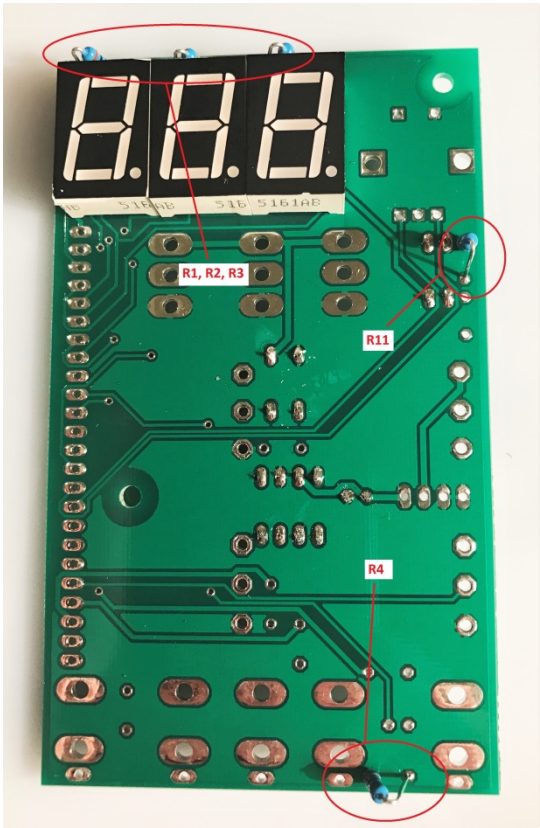
## Step 2

On the component side, insert the BAT 85 diodes D4, D5, D6, D7, D8, D9, D10, D11 being careful of the orientation. Make sure the black stripe on the diode aligns with the stripe printed on the PCB. Also insert the 10uf electrolytic capacitor C1. Double check the orientation, solder in place and trim leads.

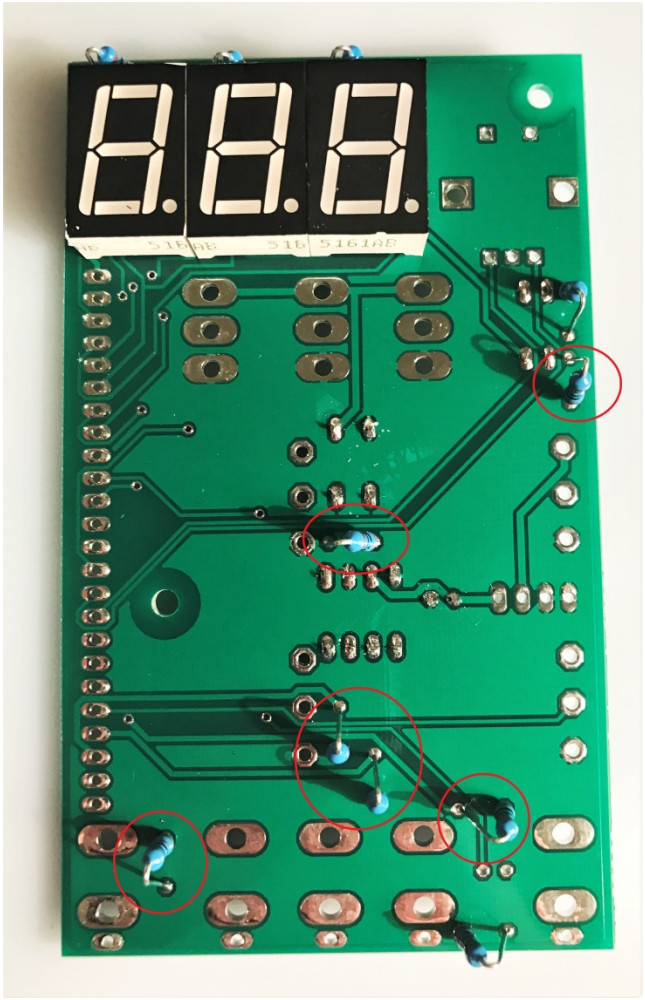



## Step 3

Install the 220 ohm (**For Version 3**) or 180 ohm (**For Version 2**) resistors R1, R2, R3 and R4 on to the BACK of the PCB (no silk screen side). Install the 10K resistor R11 on to the BACK of the PCB (no silk screen side). Solder in place, trim leads.

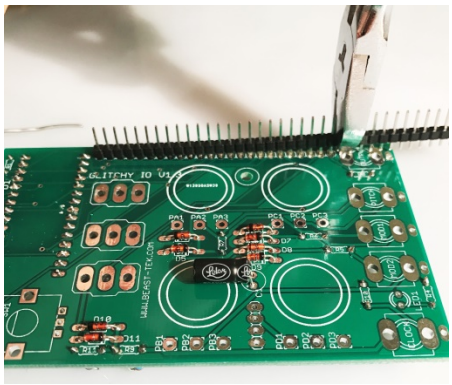


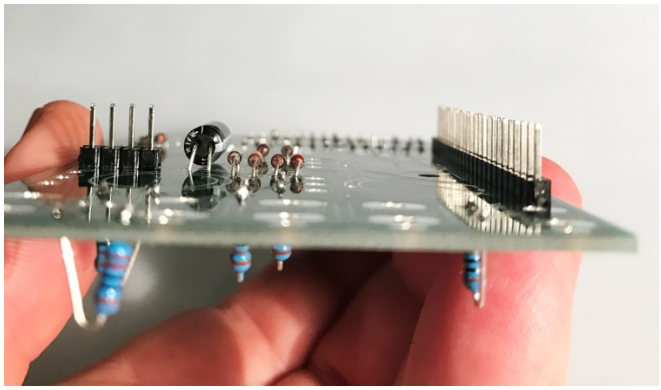
Install the six 1K ohm resistors R5, R6, R7, R8, R9 and R10 on to the BACK of the PCB (no silk screen side). Solder in place, trim leads.



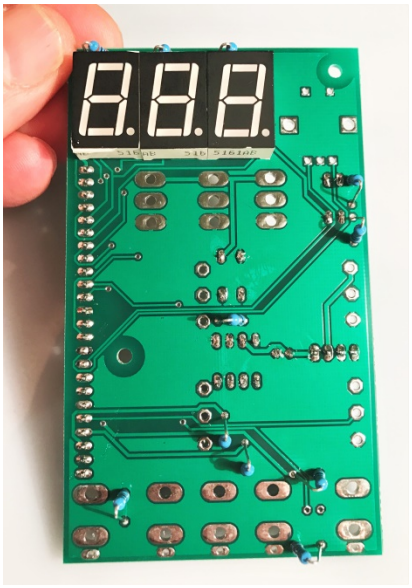
## Step 4

JP1, JP2 - Trim the male header pins into a 24 way piece and a 4 way piece. Use the PCB as a guide if required.



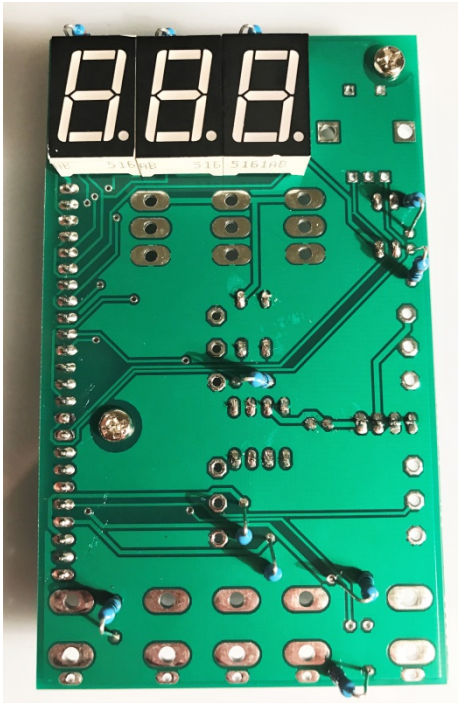


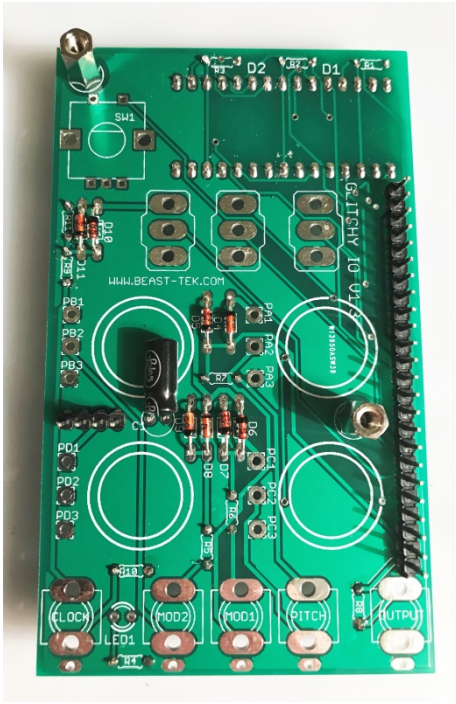
Insert JP1 and JP2 onto the front (silk screen side) of the PCB. Solder one pin at either end. Double check the header is on a right angle and is flush with the PCB. When you are happy, solder the rest of the pins.



## Step 5

Screw two silver 6mm M3 screws into the back (non silk screen) side of the board. Turn the board over and screw the two 10mm stand offs onto the screw threads on the M3 screws on the silk screen side. These will act as stabilizers when assembling the rest of the panel.





## Step 6

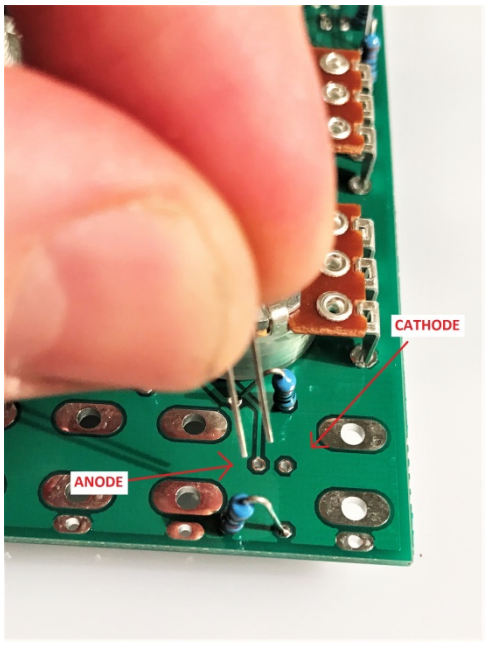
**IMPORTANT** – Do NOT solder any of the switches, pots or rotary encoder in place until the front panel is on!!! Soldering them on now will make assembling the front panel extremely difficult.

Place the four dust covers onto the back of the 16mm pots. Make sure they are lined up correctly and push them down. If they are positioned correctly you will hear a 'click' sound as they lock into place.

Take the rotary encoder and screw one nut on. Make sure it is screwed all the way down to the base of the thread. Finger tight is okay, no need for tools.

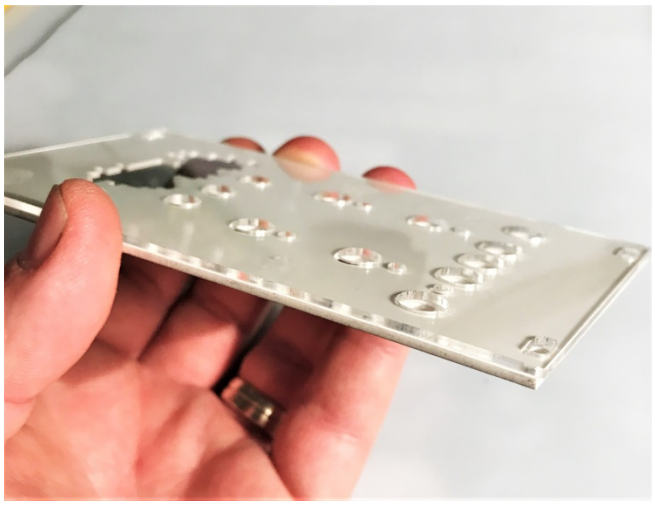
Insert the three SPST switches, press firmly so they are seated nice and tight against the PCB. Insert the rotary encoder (SW1) into place. Insert the four 5K pots into place and lastly insert the 3mm led – be careful make sure the shorter lead (cathode) is facing the edge of the board and the longer lead (anode) is facing the center of the board (refer to photos below).

DO NOT SOLDER ANYTHING JUST YET!



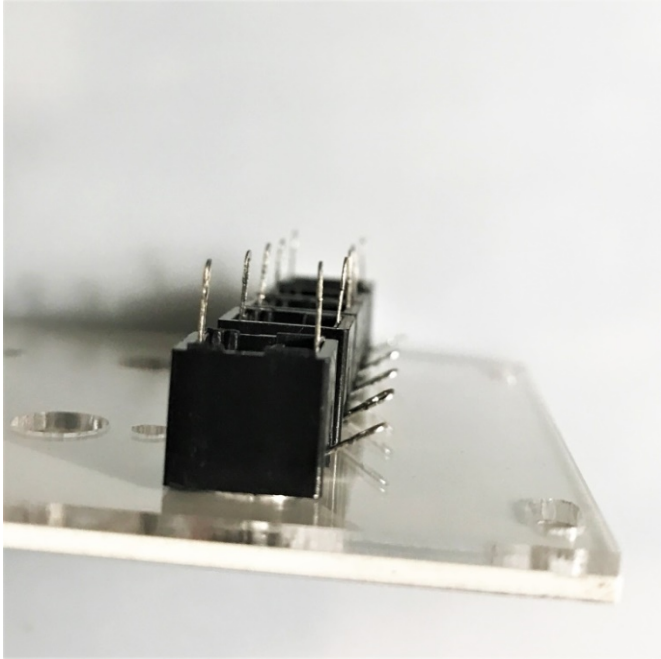
## Step 7

Take the clear acrylic panel and peel the protective coating from both sides. Place the front panel against the clear panel and make sure the holes are aligned correctly.



Now take one 3.5mm jack and insert it into the center jack hole. The hole on the clear acrylic is slightly larger so that the jack should clip into place. Now thread the nut onto the jack and finger tighten. As you tighten, hold the two panels and make sure the edges are aligned. Once you are happy with that the edges are aligned, finger tighten the nut as tightly as possible. This single jack should keep the two panels aligned whilst you insert the remaining 4 jacks and install the nuts onto them.

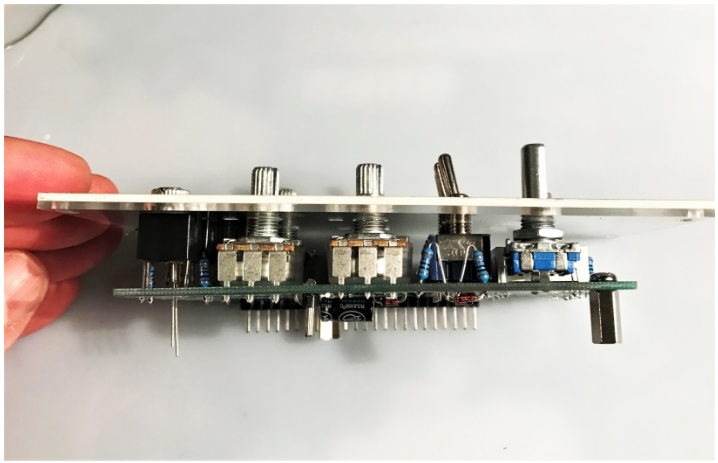

Once all five jacks are installed, make sure that the pins are all straight and make sure the jack bodies are aligned nicely. Bend the ground connector on each of the jacks at approximately 45 degrees as per the photos.

## Step 8

Now it's time to join the front panel and PCB together. If the jacks are aligned well and the pins are straight, this is a pretty easy process.

Place the panel at approximately a 45 degree angle to the PCB. First insert the SPST switches, then lower the panel over the pots and gently push the jack pins into place. Be careful not to trap R8 underneath the output jack!



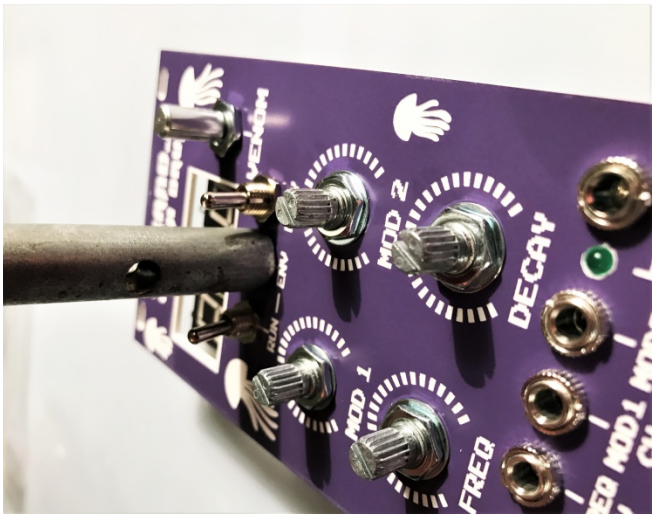
Check the alignment of everything and make sure everything is sitting flush.

Using a tube spanner or ring spanner (to avoid scratching the front panel) place and tighten a nut onto the rotary encoder.



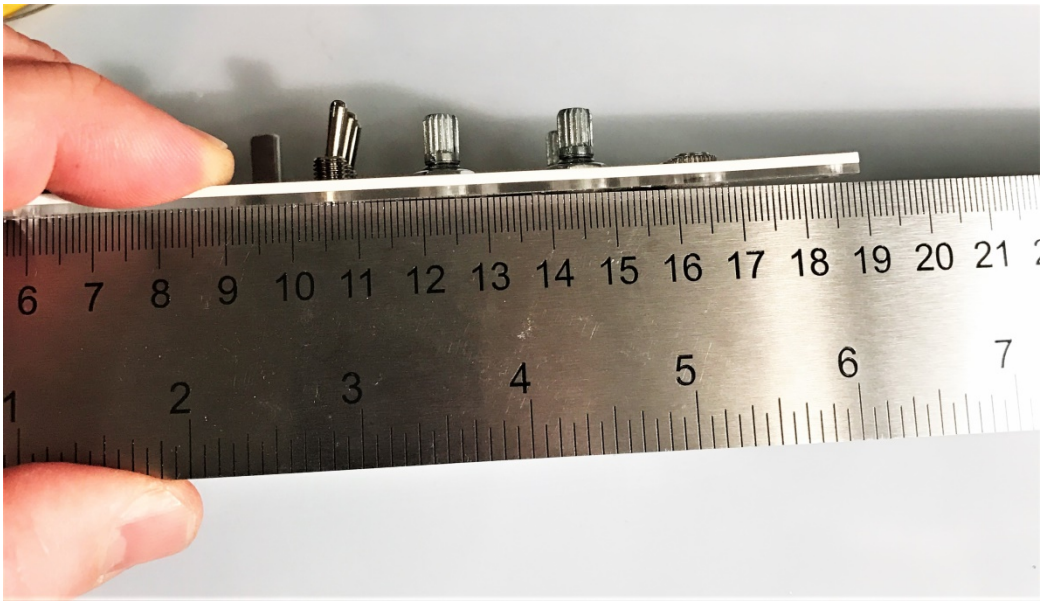
Place washers and nuts onto the 4 pots. Tighten the nuts, again using a tube spanner or ring spanner to avoid scratching the front panel.

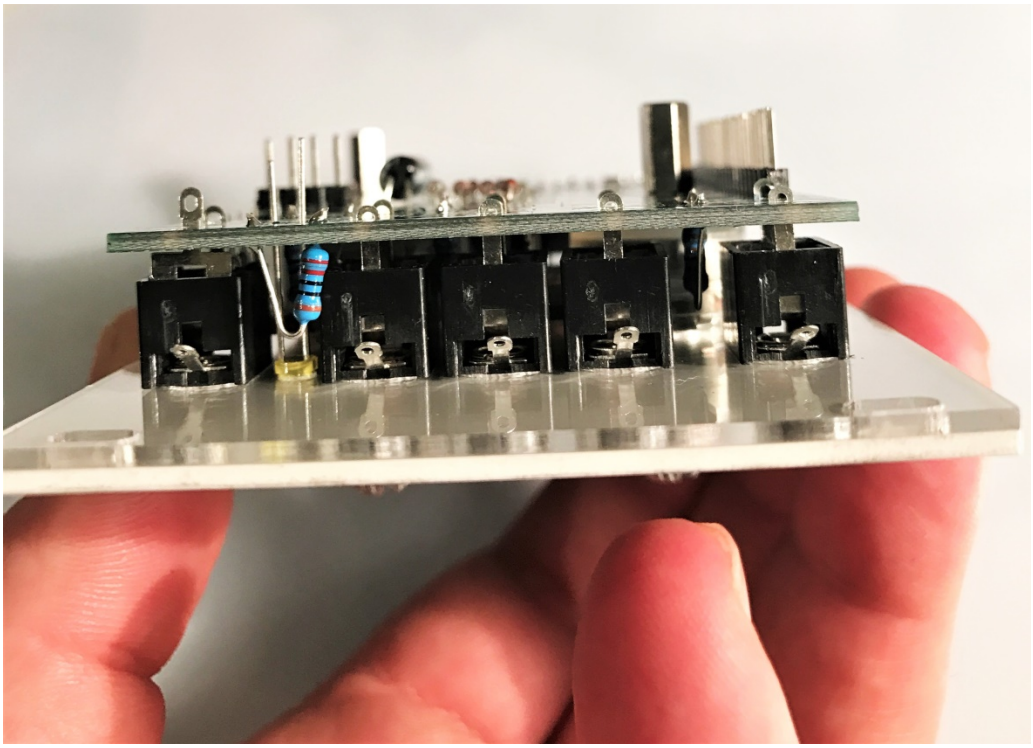
The three smaller remaining nuts are for the SPST switches. Inspect them carefully as often one side of the nut usually has a slightly raised edge from the zinc plating process, whilst the other side is smooth. Install the nuts onto the SPST switches with the SMOOTH side of the nut facing the panel. If the nuts are screwed on with the raised side of the nut facing the panel, it may scratch the panel. Tighten the nuts on the SPST switches using a tube or ring spanner.




Inspect the panel and board to make sure the panel is flat and everything is sitting flush / neatly.

Use a straight edge or square make sure the panel is straight. Because the controls and jacks have not been soldered, it is still possible to make gentle adjustments to make everything flush/square.

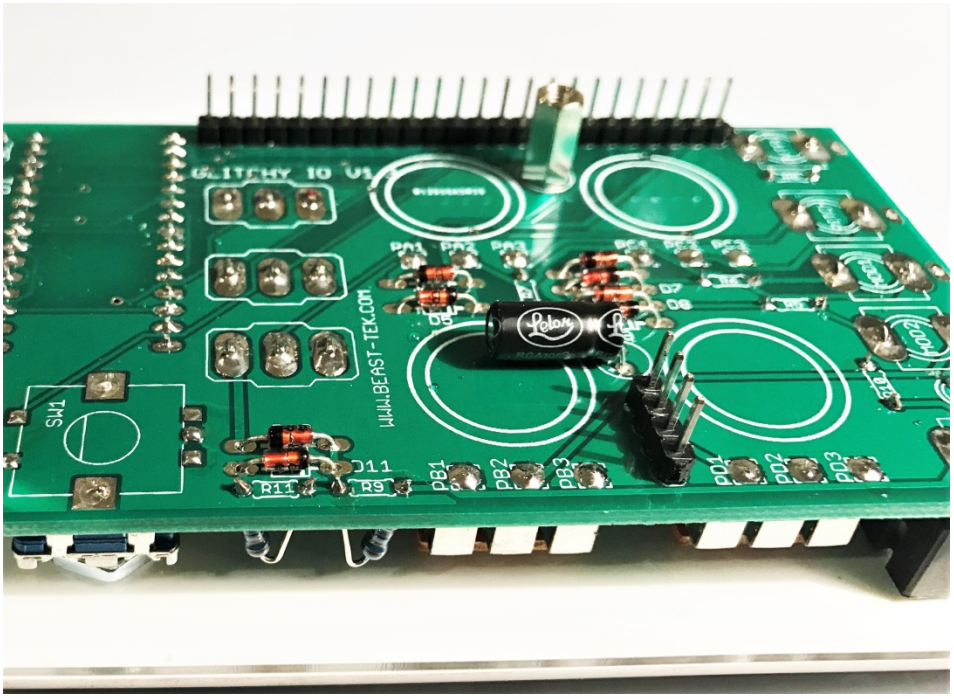


Lastly, push the LED to make sure it is inserted firmly into the hole on the front panel.

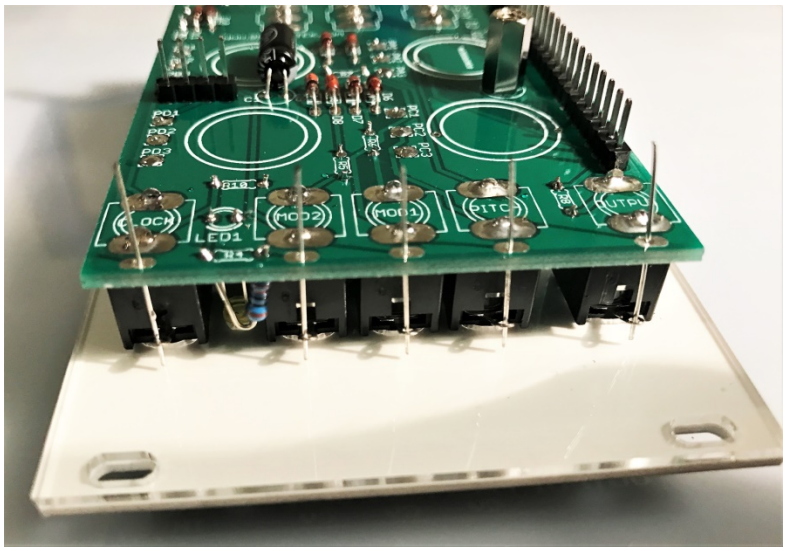
## Step 9

Carefully turn the panel over and solder all of the controls into place. Pay special attention when soldering the SPST switches as applying too much solder can cause a short circuit between the switch body and the PCB.



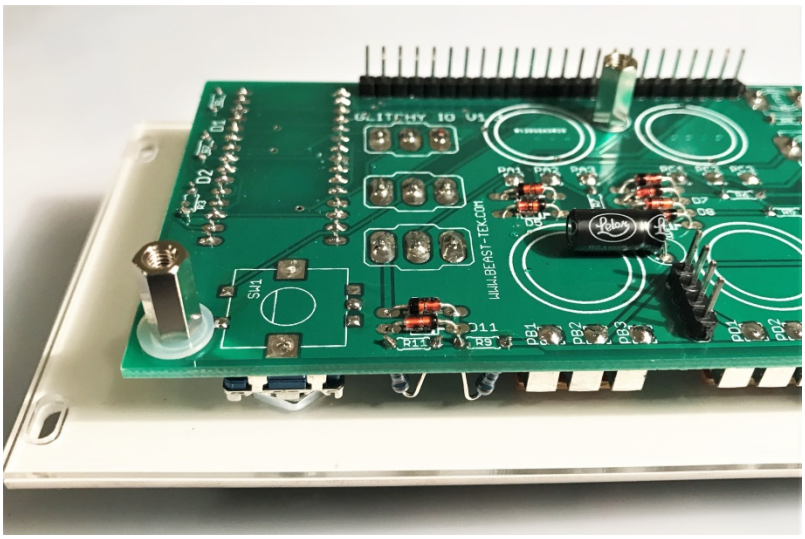

## Step 10

Take five of the cut component leads and insert them into ground pads on the PCB. Feed them into the ground pin on the jacks solder into place and trim excess.



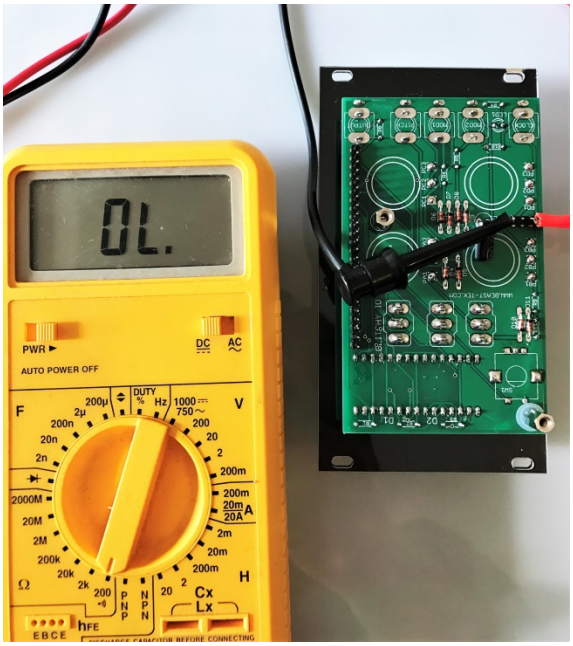
## Step 11

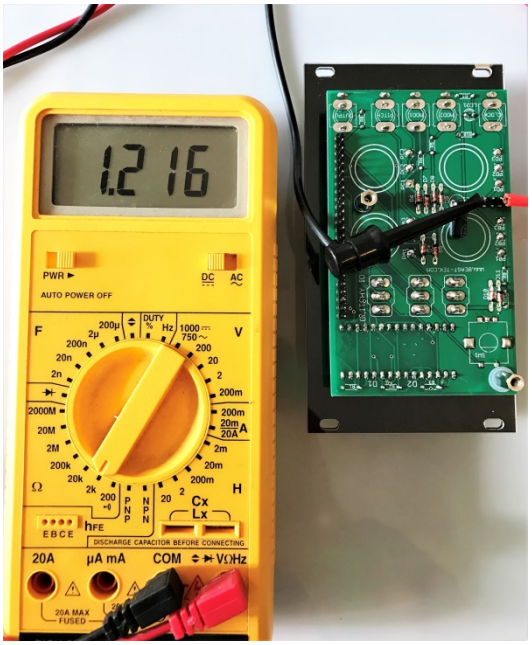
Unscrew the 10mm stand-off located in the upper corner of the PCB. Insert the 1mm nylon washer onto the screw and replace the 10mm stand-off on top of the washer.




Inspect the soldering joints and look for shorts, bad joints and unsoldered joints.

Connect a multimeter to pin 1 and pin4 of JP2 (4 way header). Set it to either continuity mode or resistance mode. In resistance mode, there should be approx 1.2K ohms of resistance between pin 1 and pin 4. In continuity mode there should be no connection detected.



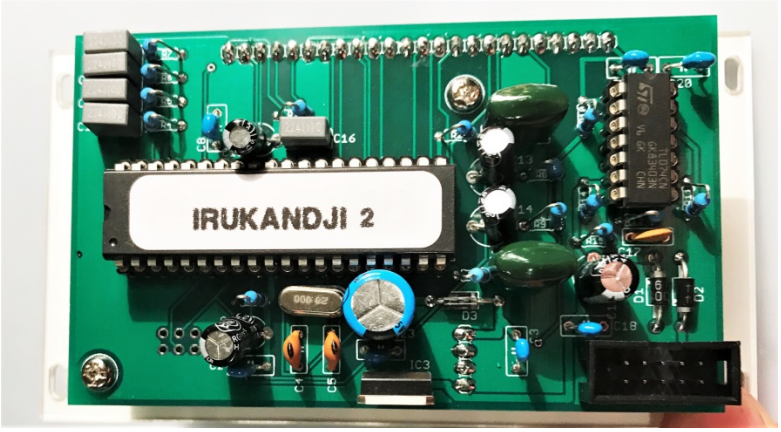
If the resistance is 0 or the continuity test registers a connection, then there is a short circuit somewhere. Inspect the board visually, if you cannot locate a short then there may well have been too much solder applied to the SPST which caused a short between a switch body and the PCB.

## Step 12

Turn the PCB/panel over and install the knobs.

## Step 13

Join the main board and the control panel together. Use the two remaining silver 6mm M3 screws to attach the main board to the stand-offs on the control board.




## Step 14

Power on/test!


# Troubleshooting

## I can't get good solder flow on a few of the pads!

The most likely cause of this is your iron is not hot enough. I love my 13w pencil iron but it doesn't get hot enough to get good solder flow on ground pads. I grabbed a cheap 25w iron and all pads flowed like magic. 25 watt should be plenty warm enough, don't go higher or you will start cooking components.

## No audio coming out of the output jack

See step 8 of the Interface board build. The only time I had audio issues was once when I trapped R8 under the output jack, which snapped the lead on R8.

## One or more knobs don't work

If everything works fine with the exception of one or more knobs, the most likely cause is that some of the BAT85 diodes are installed in reverse.

## I think the microcontroller is dead

Every single MCU has been programmed and fully tested, then packaged into antistatic foam under ESD safe conditions. Unless you did something crazy like rubbing a balloon on your jumper and discharging it onto the MCU pins then probably not .. check your solder joints!

## I think the 7 segment display is dead

Every 7-segment display was tested during packaging. Double check the solder joints on the display pins, there could be a bridge/short or dry joint somewhere.

## The knob on the rotary encoder keeps sliding down

The rotary encoder has a D-shaft and although it seems it seems logical to place the grub screw on the knob onto the flat part of the shaft, the grub screw must be in contact with the rounded part of the shaft so it can grip tightly .. also make sure you leave enough of a gap between the knob and the nut so that the button can be pressed.

## When I connect the module to the PSU, the power trips

This will happen when there is a short circuit somewhere. Refer to step 11 of the Interface board build. If it's not a short under one of the SPST switches, then it must be a sort in a solder joint or maybe a component lead is touching something it shouldn't.

# Troubleshooting (continued)

## General trouble shooting

The most likely cause of failures is bad solder joints and incorrect component orientation. Inspecting the board in daylight can really help identify bad solder joints.

Email info@beast-tek.com for help if you get stuck :-)