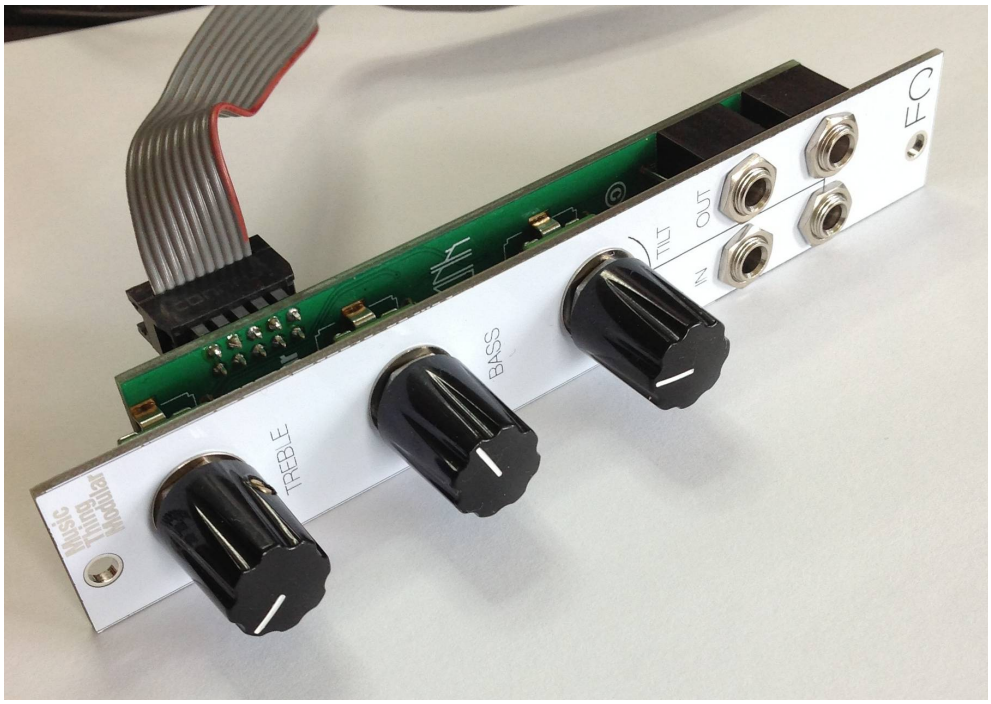
Music Thing Modular
SimpleEQ
Build Doc

(0805 version)



Music Thing Modular

Useful Links

The latest version of this doc and BOM can always be found at http://thonk.co.uk/documents/eq/

A build thread on the Muffwiggler DIY forum can be found at this URL

Quickstart

If you're new to building with SMD components, skip to the detailed notes on the next page.

If you're an experienced SMD builder you'll probably want to skip the lengthy notes below, so here's some things that will probably save you time when you're building this project.

- A detailed BOM can be found on the last page of this document.
- The silk on the PCB for diodes D1 & D2 does not show their polarity. The cathode for both components is aligned towards the nearest edge of the PCB, away from the power header. There are pictures on page 16 that show this.
- The op-amps might not have pin 1 indicated by an adjacent dimple/dot in the package, or by a semi-circular indentation at the end nearest pin 1/pin 8, as is common with through hole ICs. SMD devices often have one of the longer sides with a sloped face, the other with a vertical face. If you have such a device, pin 1 is the left-most pin of the side with the sloped face when the device is pointing upwards. The silk screen on the PCB shows both the traditional circle next to pin 1 and also a line to indicate the orientation for devices with a sloped face. Tthere are pictures on page 18 that show this.
- The suggested build order is to start with the components that have the lowest profile & work upwards, so thats
  - resistors
  - non-polarised capacitors
  - diodes & PTC fuses
  - ICs
  - electrolytic capacitors
  - through hole components and panel
- It's suggested that you regularly stop to clean and inspect your work. It's much easier to re-solder a joint when the board doesn't have the jacks and knobs fitted.

Recommended Tools

To work with surface mount devices, such as in this kit, it's a good idea to have the following tools to hand:

- Soldering iron with a small tip. A 1-2mm chisel or point tip is preferable.
- Fine gauge solder. Ideally 0.7mm or less.
- Solder wick. This is almost essential for removing excess solder, especially when soldering the ICs.
- Flux pen. This isn't essential for a kit like this with only 2 x 8-pin IC's, but it does make soldering some components much easier.
- Fine point tweezers. These are essential for handling surface mount components.
- Magnifying glass or jewellers Loupe. You'll need some method of inspecting solder joints. Magnification of 10x is recommended.
- Soft haired brush and PCB cleaner for removing flux. The type of cleaner will depend the on the solder you use. Isopropyl alcohol usually works well.

Introduction to working with Surface Mount Components

This section isn't meant to be a definitive guide to working with surface mount technologies. You'll find plenty of tutorials online for that. This is intended to provide some tips as to how to approach building this kit.

Many surface mount components are supplied in tape packaging. For passive components such as resistors and capacitors, the tape is usually made of cardboard, whilst semiconductors, such as ICs, transistors, diodes etc will be in a anti-ESD plastic. Most tapes will have a layer of clear plastic film over one side. To remove the device, the easiest way is to carefully peel away the plastic film with a pair of tweezers or fine nosed pliers. Take care when you remove this film. These components are small, and it's easy to lose them. It's a good idea to remove them from the packaging & place them into a bowl or other similarly shaped container, then pick them from there as you place them onto the PCB.

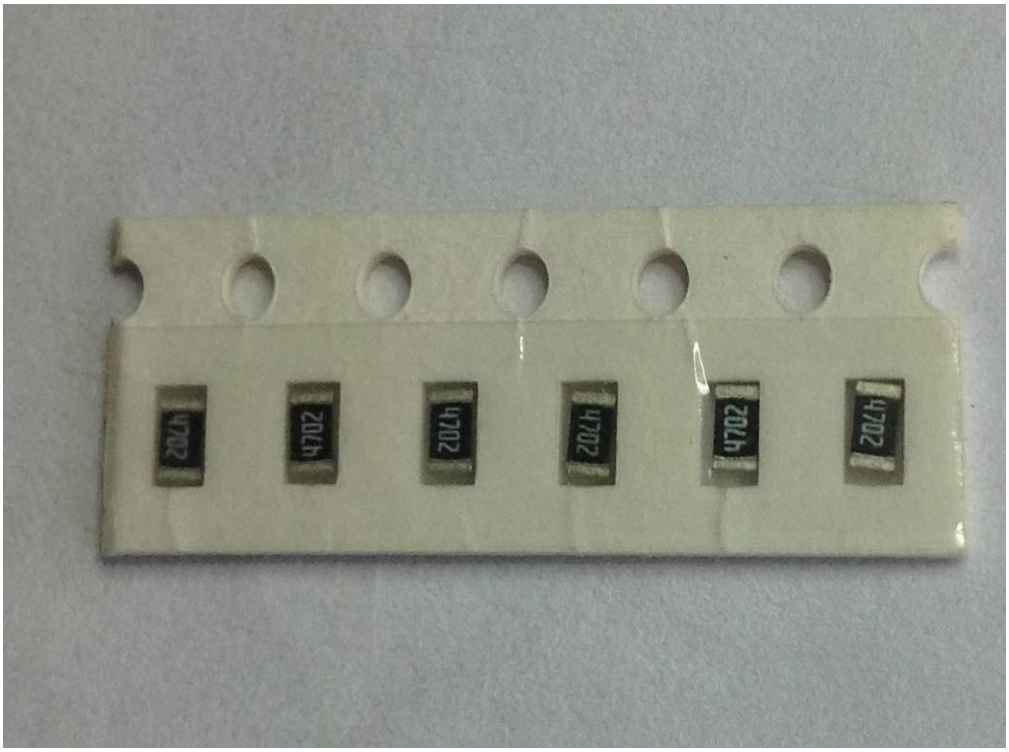


Illustration 1: SMD resistors supplied in tape

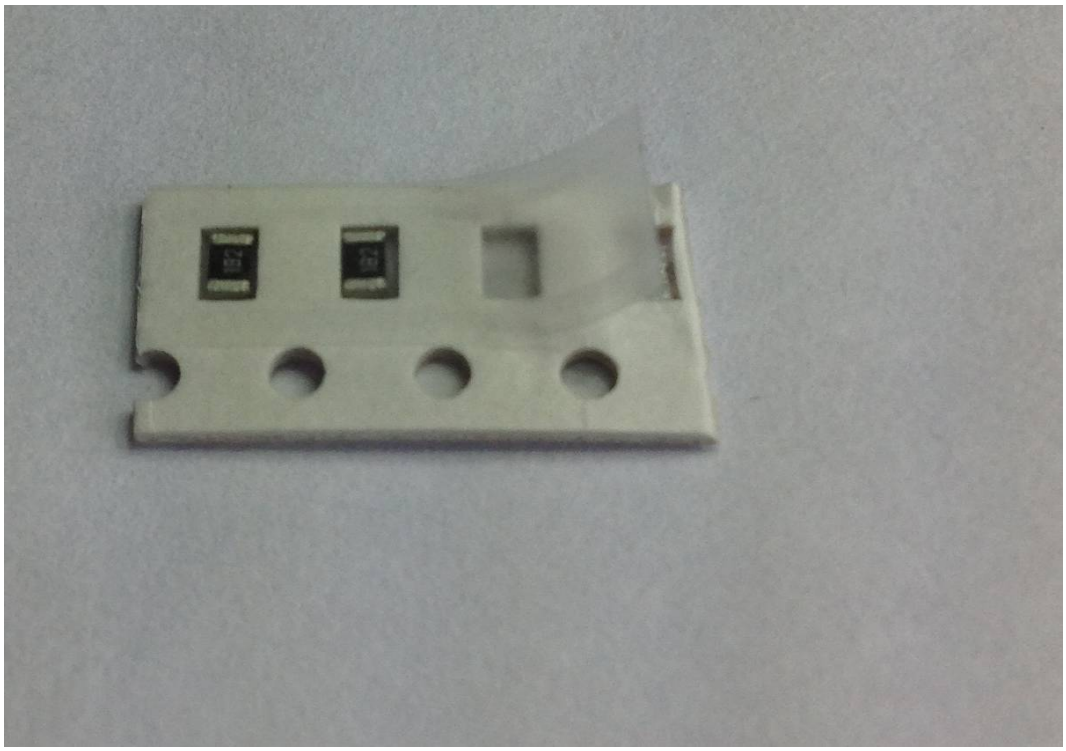


Illustration 2: SMD resistors in tape showing plastic cover partially peeled away

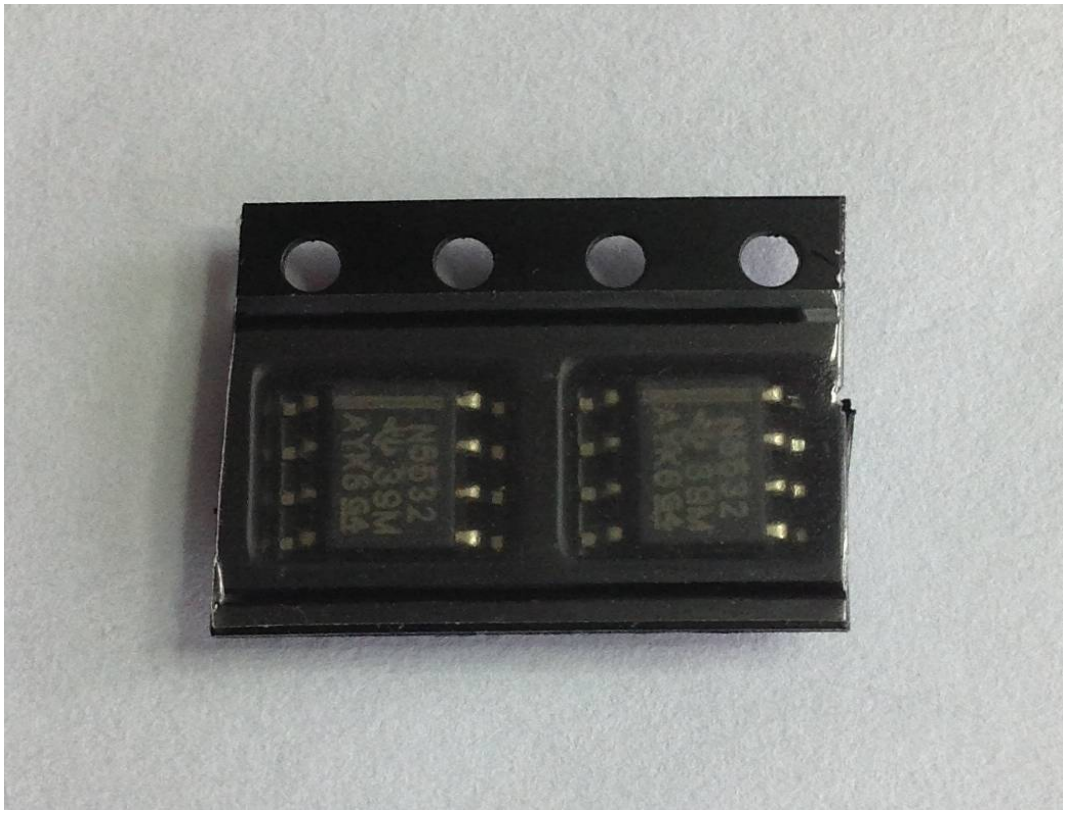


Illustration 3: SMD IC's in tape packaging

Install one component type & value at a time. Many of these components are not labelled with their values, so don't mix them up. Finish one component type & value before proceeding to the next. Don't rush. Regularly inspect your work with the magnifying glass or loupe. Clean your work as

you go. You'll find it much easier to inspect your work with the solder flux residue removed.

There's many useful resources online for tutorials on working with surface mount devices. The techniques here suggested here are one way to solder surface mount devices. As you become more familiar with working with this technology, you may find other ways of working that you prefer.

So, to the kit ...

The Build

Lets start with the resistors.

Find the component tape strip that's marked “10K”. As described above, open up the tape strip by removing the clear film cover, tipping it's contents into the bowl. There will be 4 resistors. One side will probably be a darker colour than the other. The darker side may be marked with it's value “103” (10×10^3 or 10,000 or 10K). Whilst it doesn't matter which side of the component you have facing upwards, it's usual to have the side with the value pointing upwards. As resistors are not polarised it doesn't matter which end of the resistor is soldered to which pad on the PCB.

Illustration 5: 10K SMD resistor top view showing the value “103”.

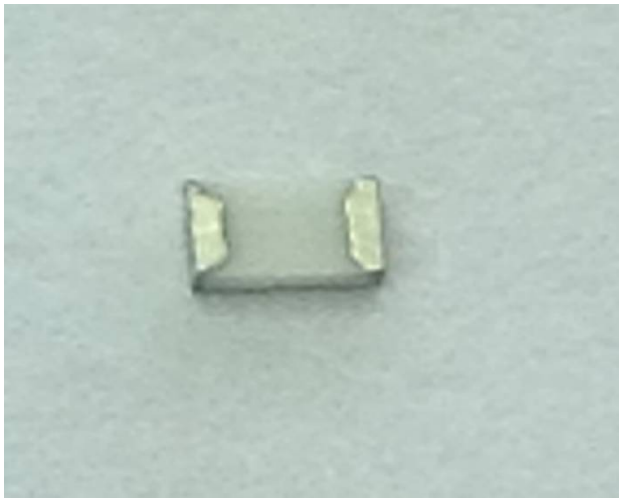


Illustration 4: SMD resistor bottom view

Before we actually solder the first of the resistors, lets take some time to actually see how the resistor will be located on the PCB. Place the PCB on a flat surface. Find the location of the first of the 10K resistors, R1. It's over towards one side of the board. Notice that the PCB has 2 pads for the component, and white silk screening around the outside of the pads.

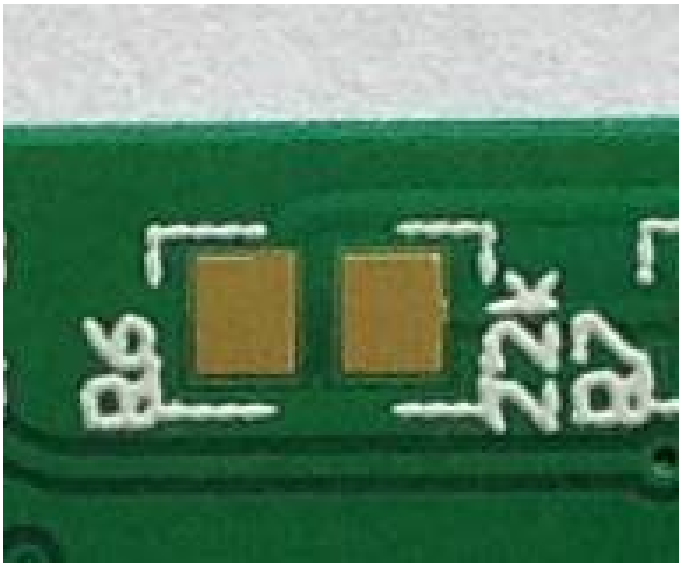


Illustration 6: SMD resistor pads and silk screen on PCB (showing R6)

Pick up one of the resistors using the tweezers, & place the ends of the resistor centrally over the two pads. Notice how much clearance you have around the outside of the resistor on the pads. When you solder the resistor in place, you'll want to get similar clearance.

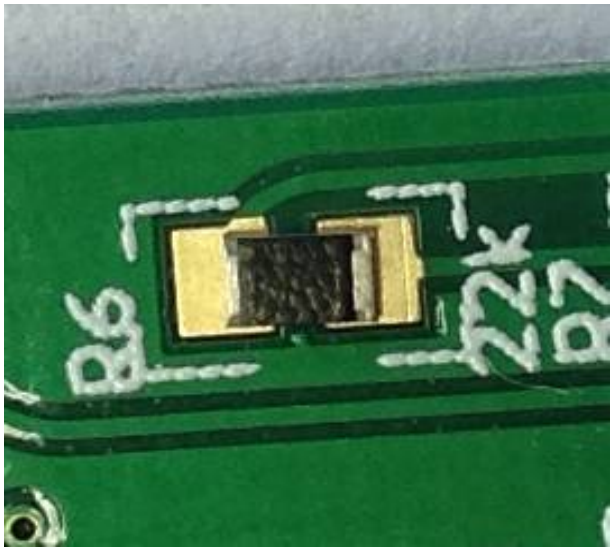
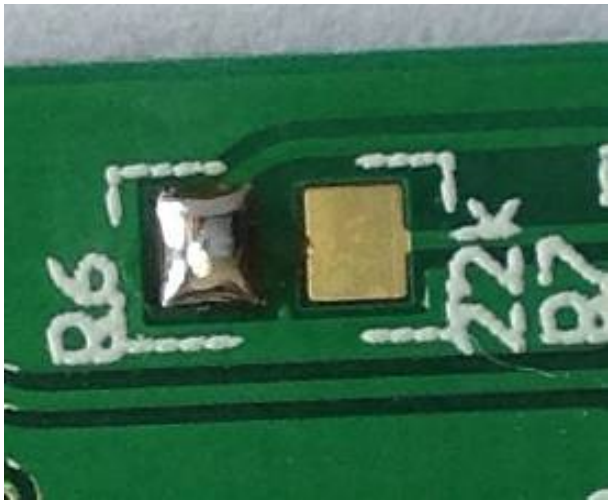


Illustration 7: SMD resistor located centrally on pads

Using the tweezers carefully place the resistor back in the bowl.

Place the tip of your soldering iron on one of the PCB pads for R1, & immediately apply a small amount of solder to the pad. Remove the iron tip. You should have a small rounded blob of solder on the pad.



If you think you've applied too much solder to the pad, you can remove this with solder wick. Place the wick on top of the blob of solder, apply the tip of the soldering iron briefly. The wick will soak up the solder. The wick will probably remove almost all the solder so you'll need to re-apply a small amount.

Take one of the resistors in the tweezers, ensuring the side of the resistor with the value marking is pointing upwards. Re-apply the soldering iron tip to the pad with the solder on it, & quickly slide one side of the resistor onto the pad, so that the edge of the resistor is lined up on the middle of the pad, and the resistor is approximately covering equal parts of both pads, as you practiced previously. Quickly remove the soldering iron tip.

If the resistor is not placed correctly, re-apply the tip of the soldering iron to the pad & the edge of the resistor & use the tweezers to align the resistor. Remove the iron tip. Try to do this as quickly as possible. It's much easier to move the resistor with just one pad soldered, so now is the time to get it's positioning right.

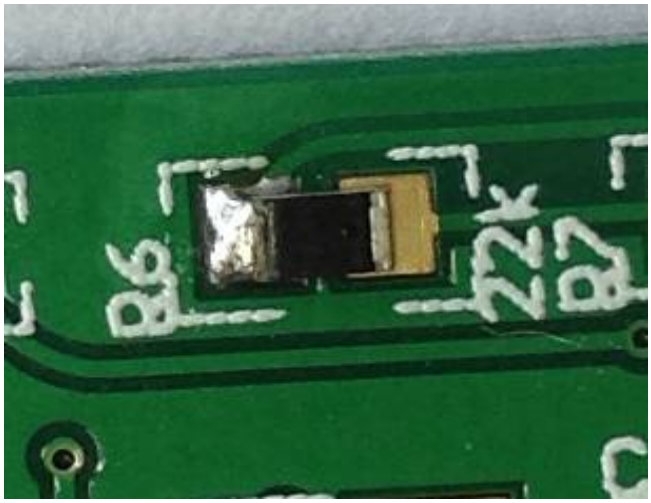


Illustration 8: resistor soldered to one pad

Now solder the other pad. Place the soldering iron tip on the R1 pad that you didn't solder, resting the end of the tip up against the side of the resistor, then immediately apply a small amount of solder to the contact point of the soldering iron tip, the pad & the resistor. The solder will flow over the pad & into the gap between the resistor & the pad. Remove the soldering iron tip. This action should take no more than a couple of seconds. Don't worry if there's a large blob of solder over the end of the resistor. Provided it's not touching the other pad, it should be fine.

The resistor should now be located on top of the pads in a similar position to how you practised earlier.

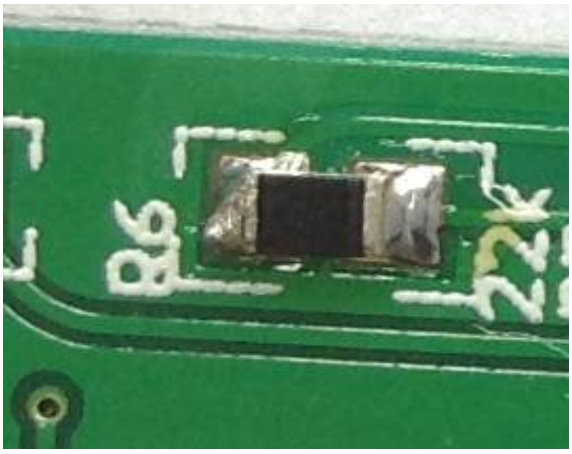
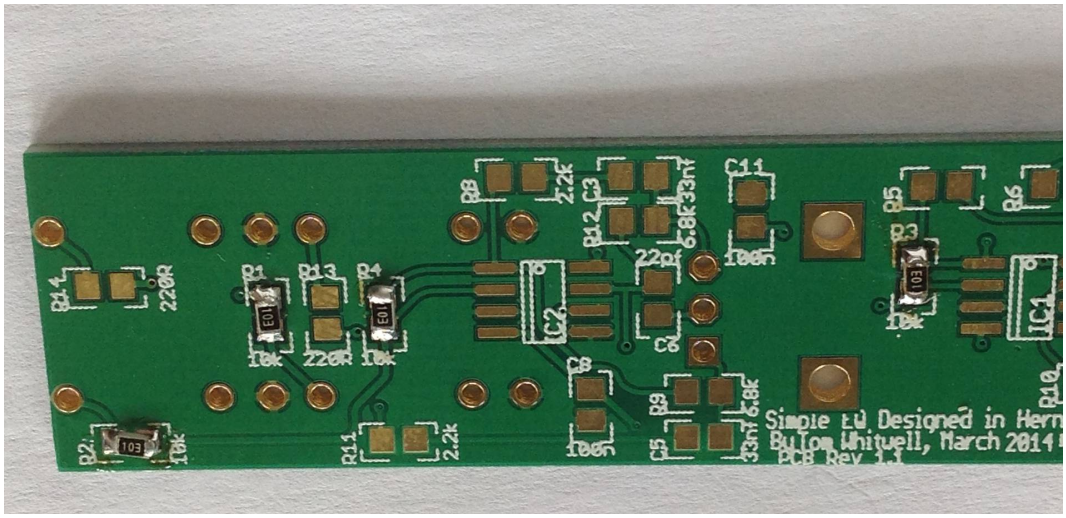


Illustration 9: resistor soldered to both pads

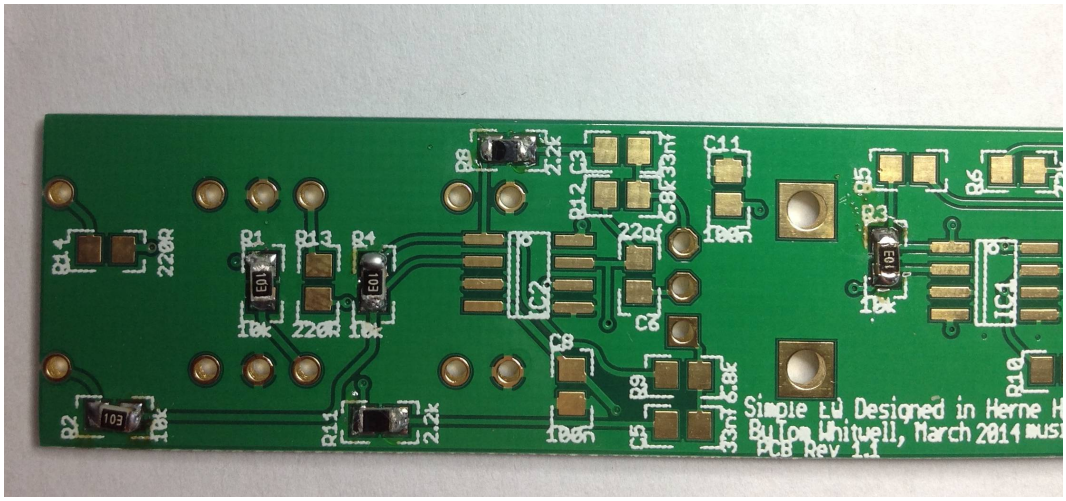
If you put too much solder on either pad, you can remove this using solder braid. Carefully place the braid on top of the solder blob, & briefly apply the soldering iron so the braid soaks up the solder. If the braid soaks up too much solder, re-apply a small amount of fresh solder.

Now install resistor R2 using the same procedure as you've just done for R1 - Blob of solder on one pad ; Install the resistor ; Solder the other pad.

Then repeat this for R3 & R4. After which the board should look something like this:

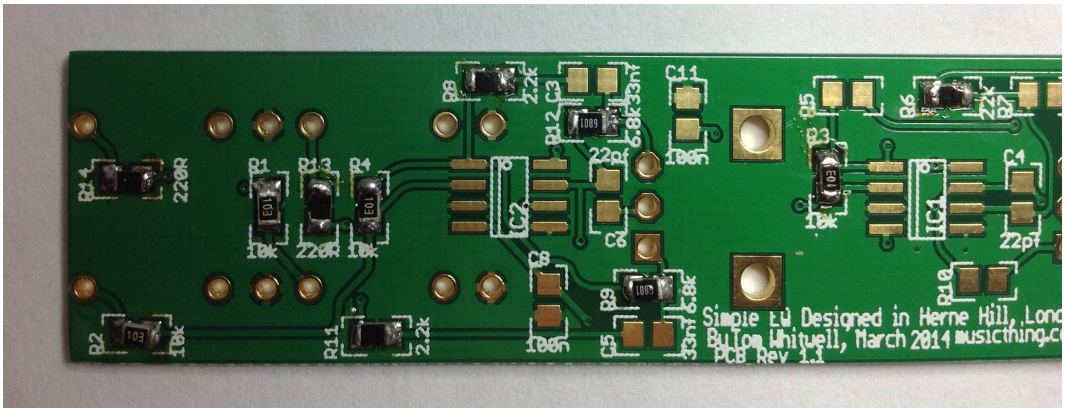


Now install the 2 x 2K2 resistors R8 & R11. Note the one used in the picture here doesn't have the value written on the black side. Thats not a problem.



Now install the 2 x 220R resistors R13 & R14:
Now install the 22K resistor R6:
Now install the 2 x 6K8 resistors R9 & R12:
Now install the 2 x 1K8 resistors R5 & R10:
And finally the 9K1 resistor R7.

Now that all the resistors are installed, the board should look something like this:



It's a good idea now to clean up the board & then give it an inspection to check everything you've installed looks OK.

To clean the board, use a soft haired brush soaked in a cleaner appropriate for your solder, as you would do for through hole boards. Gently brush the components, to remove the flux. You may need to apply more than one brush-full of cleaner.

Now use a magnifier glass or jewellers loupe to inspect your solder joints. Check that the contact points on the resistors & the pads on the PCB are covered in solder. Check that the solder doesn't short on to other tracks or pads. Excess solder can be removed using solder wick as described previously.

Once you're happy with the placement of the resistors, it's time to progress on to the ceramic capacitors. As with the resistors, these are non-polarised, meaning they can be oriented in either direction on the PCB. Unlike the resistors, you'll notice that these are not flat, but have a square profile. With these it doesn't matter which face is pointing upwards, as you'll see they don't have a value printed on any face.

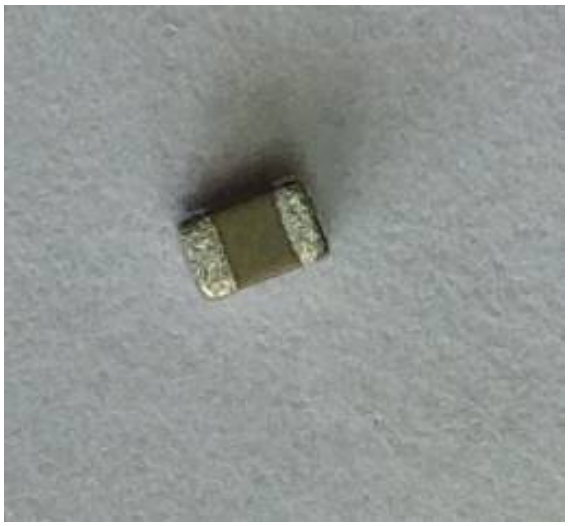


Illustration 10: surface mount ceramic capacitor

Start with the 5 x 100nF capacitors designated C1, C7, C8, C11, C12. Use exactly the same procedure to solder these capacitors as you used for the resistors. Again, you might want to do a dry run & offer one of the capacitors up to it's pads before you start to solder, so you get an idea of how the capacitor should be located on the pads.

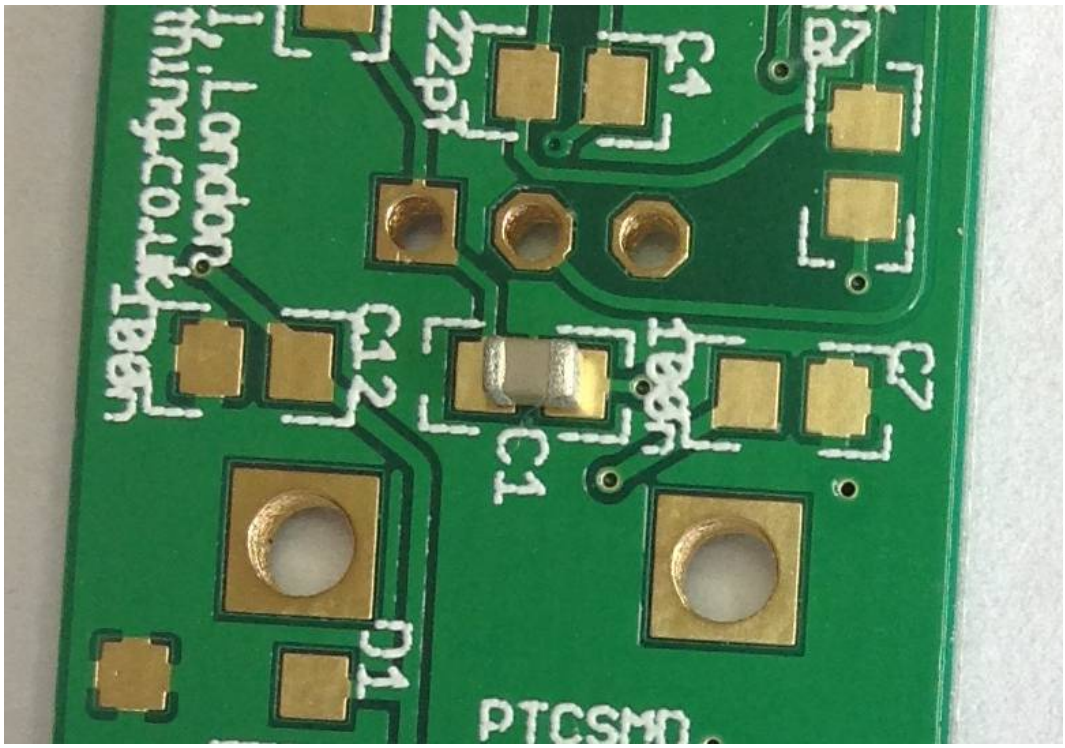


Illustration 11: ceramic capacitor aligned on PCB pads

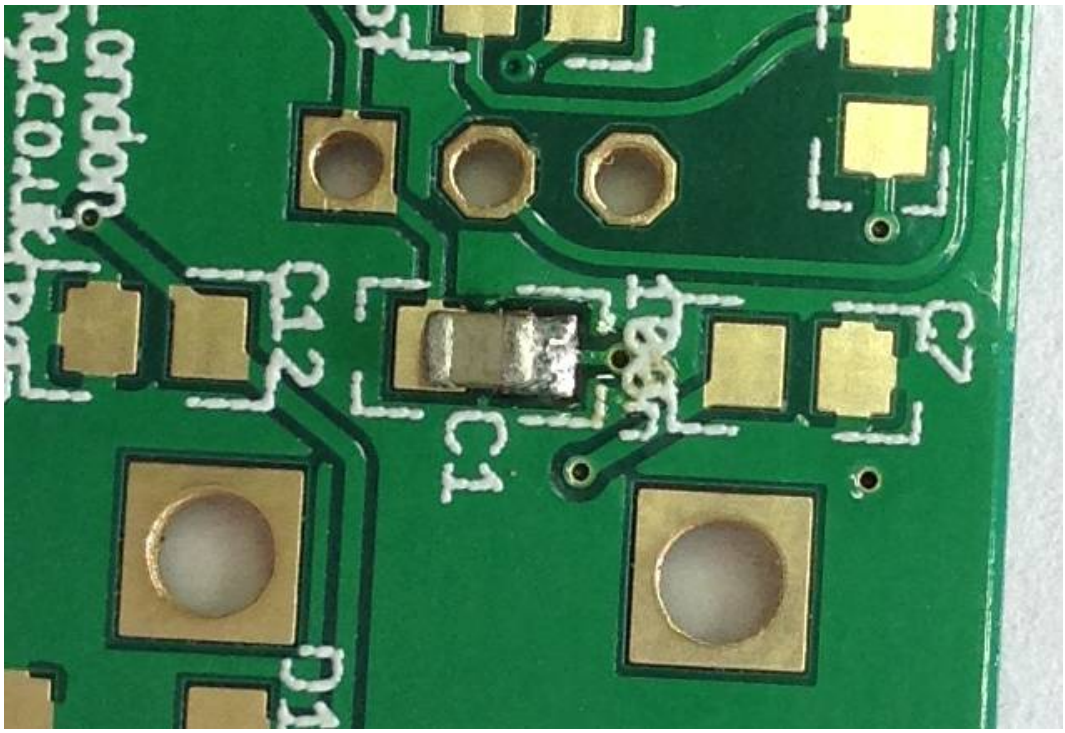


Illustration 12: ceramic capacitor soldered on one side

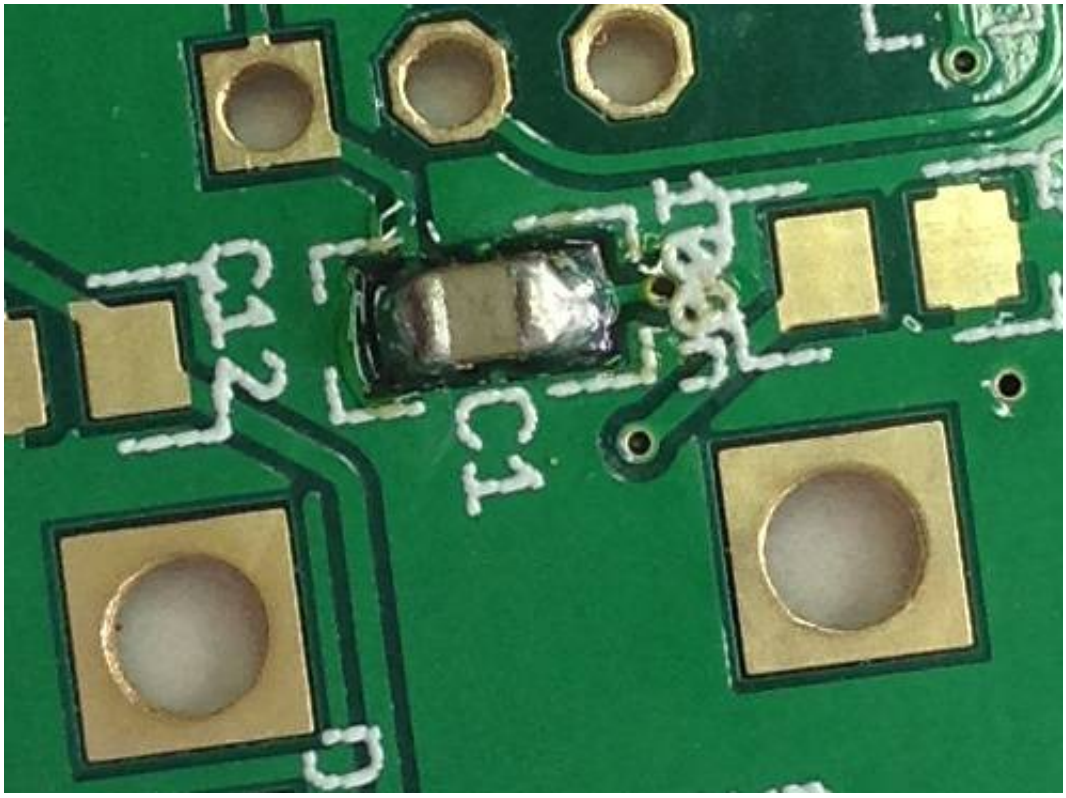


Illustration 13: ceramic capacitor soldered on both sides

Now install the 2 x 22pF capacitors C4 & C6
Now install the 2 x 33nF capacitors C3 & C5
Now install the 4.7nF capacitor C2

Next install the PTC fuses, F1 & F2, which are installed by the power header. Use the same technique for these as for the resistors and the non-polarised capacitors. These devices are not polarised, but they do need to be installed the right way up, as one side has much larger pads than the other. The side with the larger pads is the one that needs to be placed on the PCB. Try to avoid overheating these devices as they can be damaged by excess heat.

Illustration 14: PTC fuse top side. Install with this face upwards.

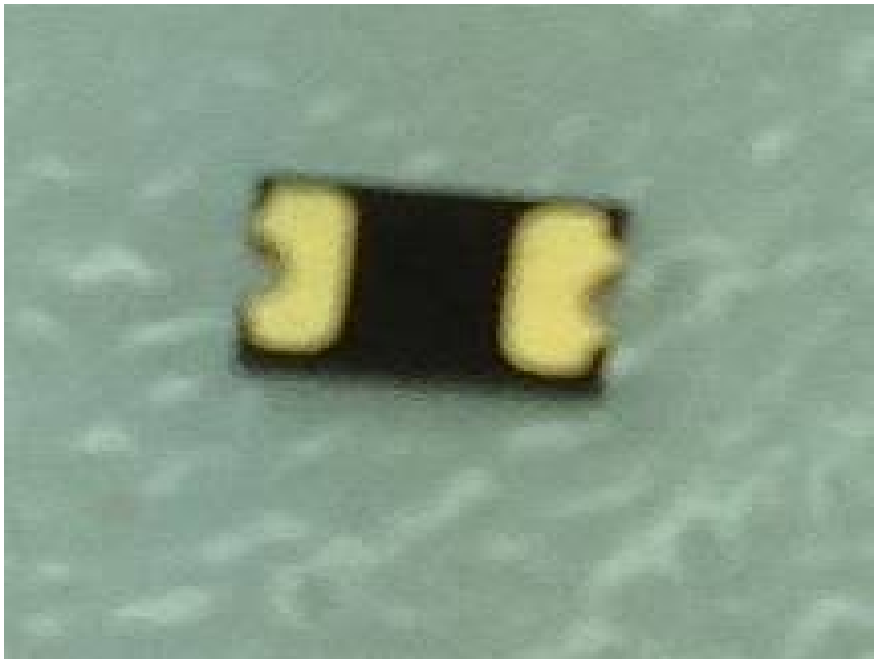


Illustration 15: PTC fuse bottom side. Install with this side on the PCB.

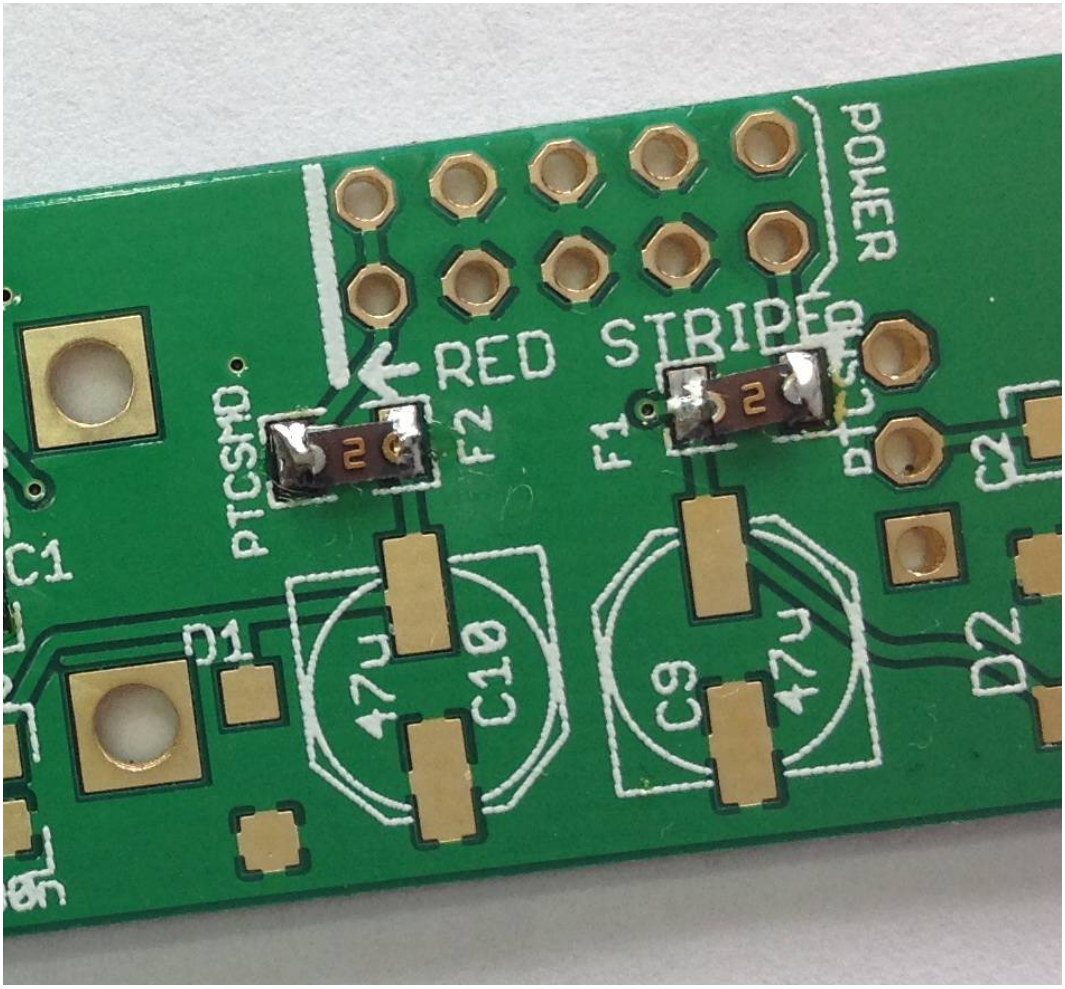


Illustration 16: PTC fuses F1 & F2 installed on the PCB

Next, install the two 1N5819 diodes, D1 & D2. Remove the diodes from the tape, & place them in the bowl. You'll notice that unlike the resistors and capacitors you've installed already, these diodes have a metal tab protruding outwards at an angle from the device. When you solder the diode onto the PCB, you want these metal tabs pointing downwards so that they're in contact with the pads on the PCB.

In addition, unlike the components you've installed up to now, these are polarised devices, meaning they must be installed on the PCB with the correct orientation. If you look at one closely, with the tabs pointing downwards, you'll notice a line marked on the top of the package, with the line closer to one of the tabs than the other. (The line may be faint, so the use of a magnifying glass or loupe will help). The end of the diode marked with this line is known as the cathode. The PCB silk screen around the pads for each of the diodes should have a marking identifying which pad the cathode end should be installed to. If this identification isn't present, the cathode end of both diodes should be installed on the pad nearest the edge of the PCB (see the picture below).

Illustration 17: Surface mount diode, showing line identifying cathode as the left hand pin

So, now you know which way the diode needs to be installed on the PCB, solder them using the same technique as the previous components. It's recommended that once you solder one end of the diode, double check you have the cathode end by the edge of the PCB, before committing to soldering the other end.

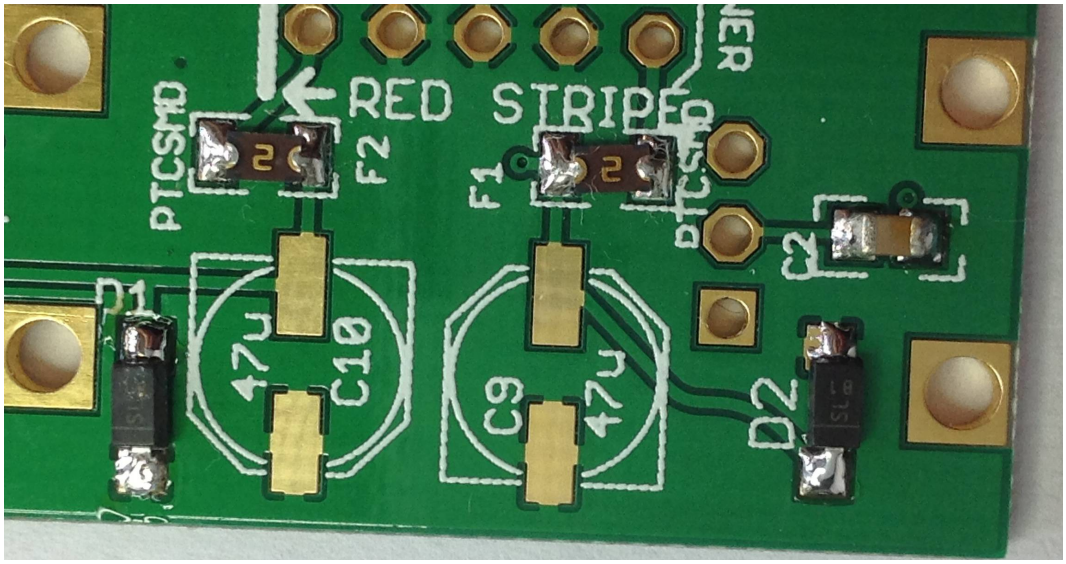


Illustration 18: Diodes D1 & D2 installed, with line identifying cathode towards the PCB edge

Now take some time to clean the board as before & inspect your soldering on the diodes, capacitors & fuses.

Next we'll do the two IC's, NE5532 dual op-amps. Remove one from the tape. Take some time to inspect the device.

Similar to the diodes that you just installed, you'll notice that the IC legs are bent outwards and downwards. You need to install the device with these legs in contact with the pads on the PCB.

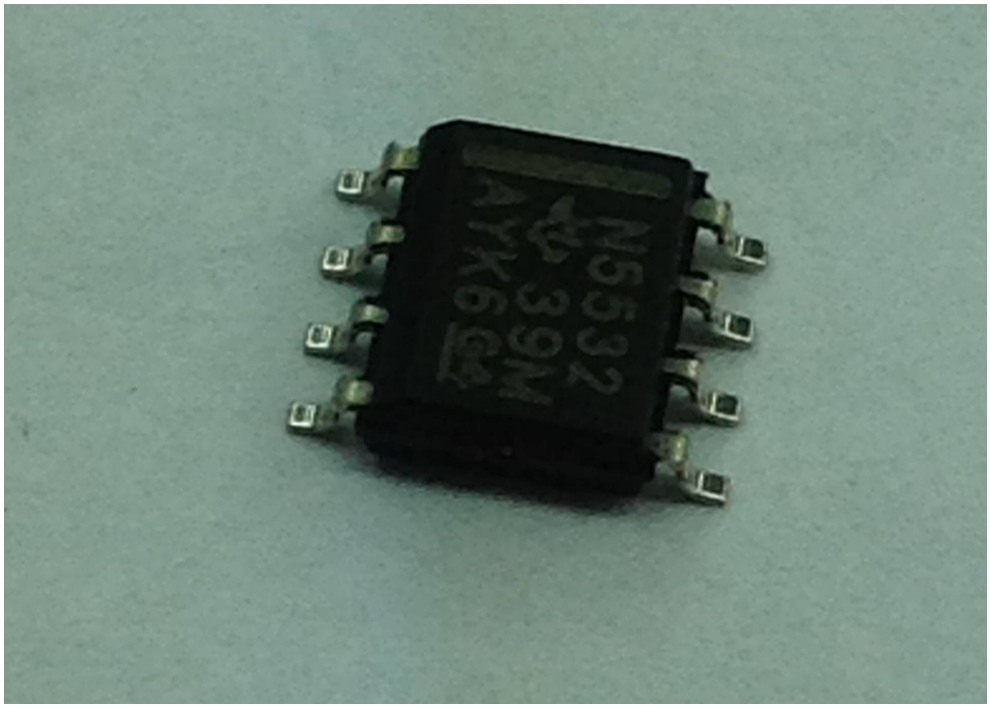


Illustration 19: Surface mount op-amp, NE5532

Unlike through hole IC's, many surface mount IC's don't have a dimple or crescent embossed into the casing to identify the location of pin 1. Instead, they have a sloped face on the side with pin 1. The other side has a vertical face. If you look at the silk screen for the two ICs on your PCB, you'll notice that it has both the usual dot adjacent to pin 1, but also a second line along one side. This second line represents the sloped face side that the slope should be placed against.

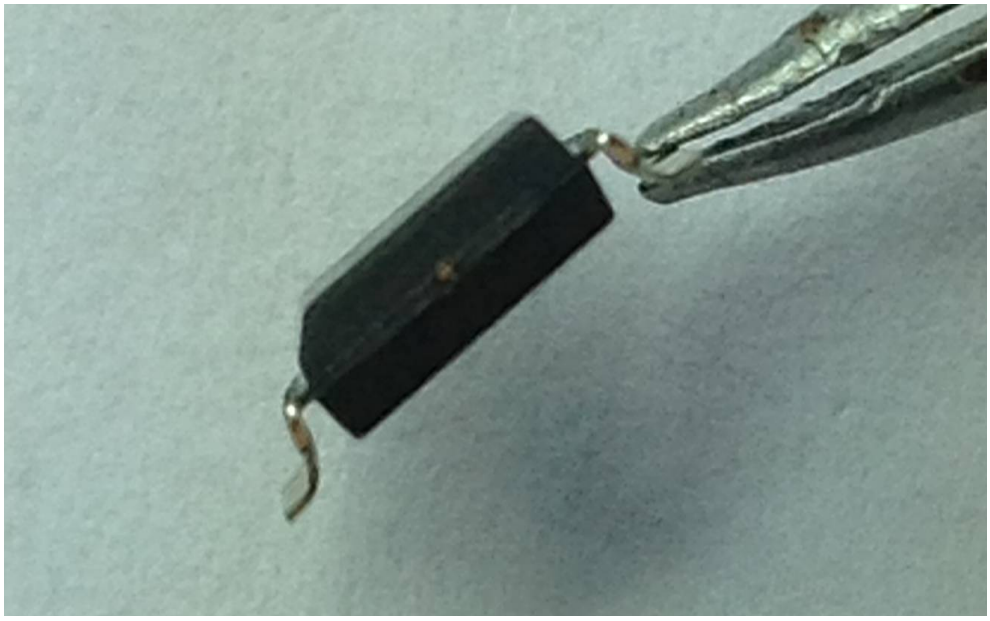


Illustration 20: Surface mount op-amp showing sloped face on packaging.

Again, before you start soldering these devices, offer one up to it's destination on the PCB. Familiarise yourself with how the device should be orientated, also how the device's pins should sit on the pads.

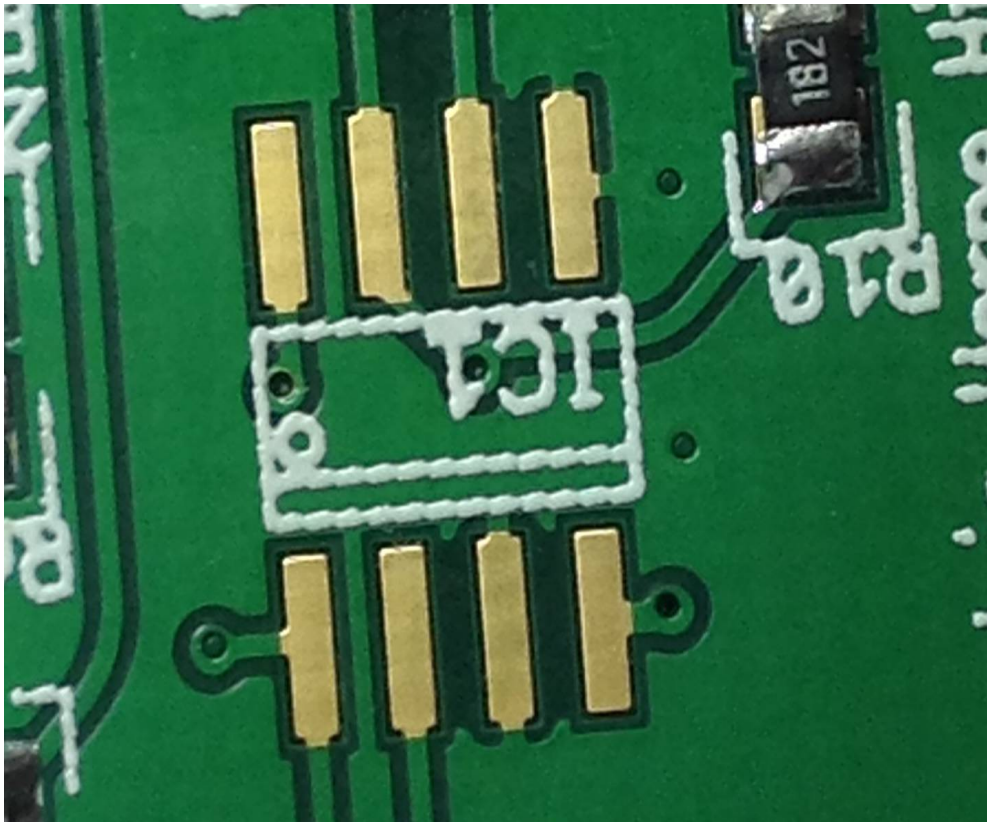


Illustration 21: silk screen for IC1 showing the dot marking the location of pin 1, and the second line for devices that have a sloping face on the package

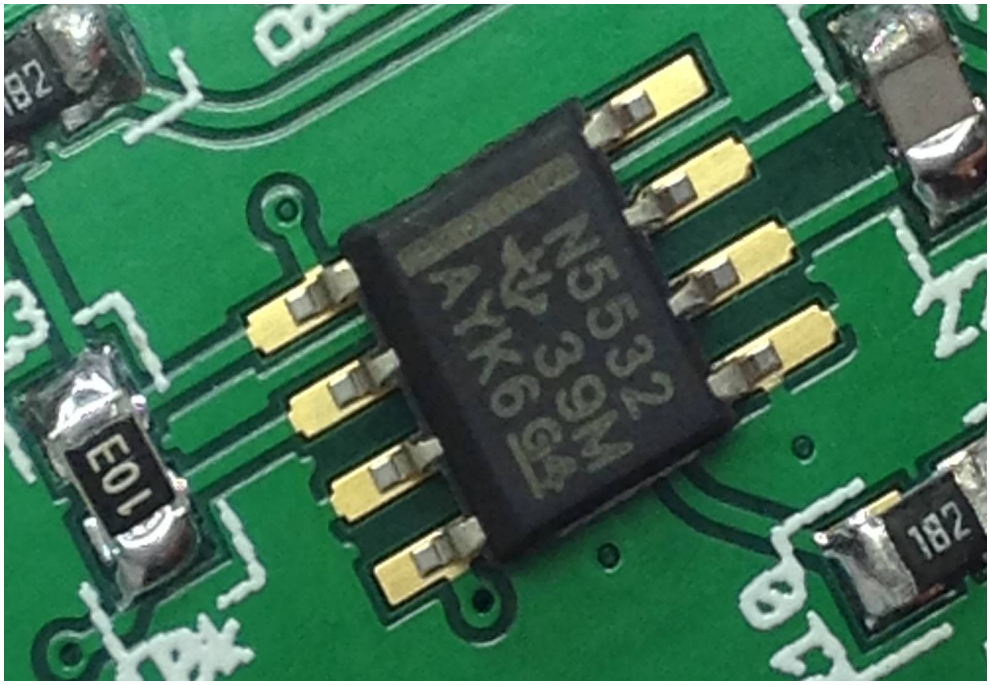
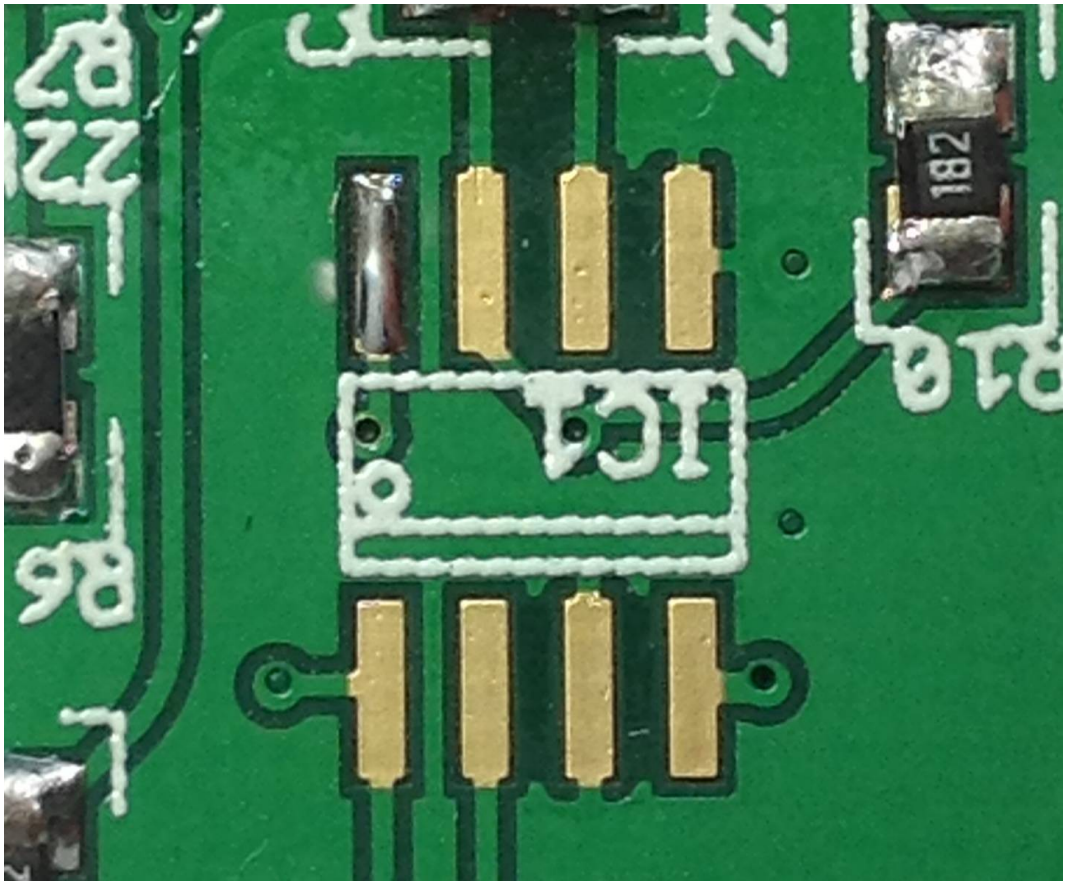


Illustration 22: NE5532 op-amp located on the pads at IC1 prior to soldering

Now it's time to solder the first IC, IC1.

If you have a flux pen, the application of flux to the pads will help soldering the ICs. If you're going to use one, apply some flux to the pads of both IC1 & IC2.

In a similar way to how you soldered the previous components, place a small blob of solder to one of the outer pins of IC1.



Gently take one of the NE5532 ICs in the tweezers. Place it on the PCB close to IC1 location, ensuring that you know which side has pin 1. Orient the IC so that it's ready to slide into place. Place the tip of the soldering iron on the outside edge of the soldered pad, so that the solder melts. Quickly slide the IC into place, trying to ensure that the IC's other legs are sat squarely on the other pads. Remove the soldering iron as quickly as you can to avoid overheating the IC. If the IC isn't quite sat squarely on the pads, re-apply the soldering iron to the soldered pad, & gently nudge the IC into place with the tweezers. Again, try to minimise the time that the IC is being heated up. If you've used it, the stickiness of the flux will help hold the IC in place. Don't worry too much about the soldering itself on this joint right now, we're really aiming to hold the device in place with this joint right now. We can redo this later once the other pins are soldered.

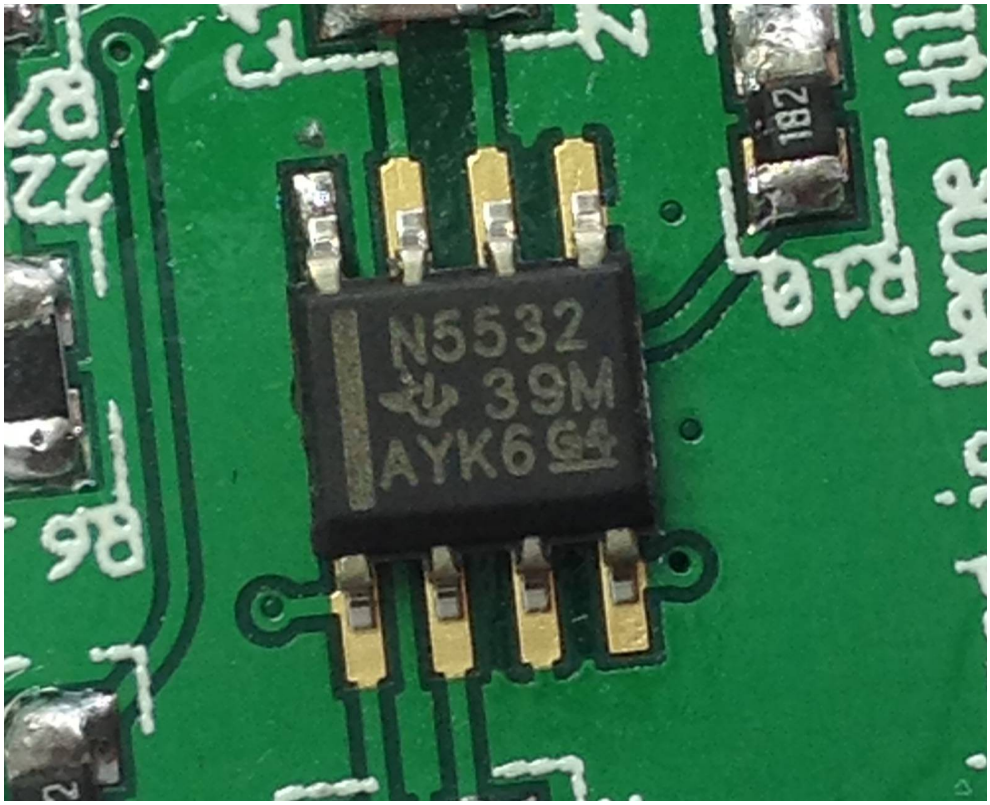


Illustration 23: NE5532 with one pin soldered

Once you're happy with the orientation of the device, it's time to solder the rest of the legs. If you're using a flux pen, apply some flux to the legs of the IC. This will help the solder flow into the joints. Start on the legs on the other side from the one you've already soldered. Put the tip of the soldering iron at the end of the pad & apply a small amount of solder. The solder should flow down the pad & over the IC leg. Repeat this for the other pins on that side of the IC, then rotate the board & do the other pins.

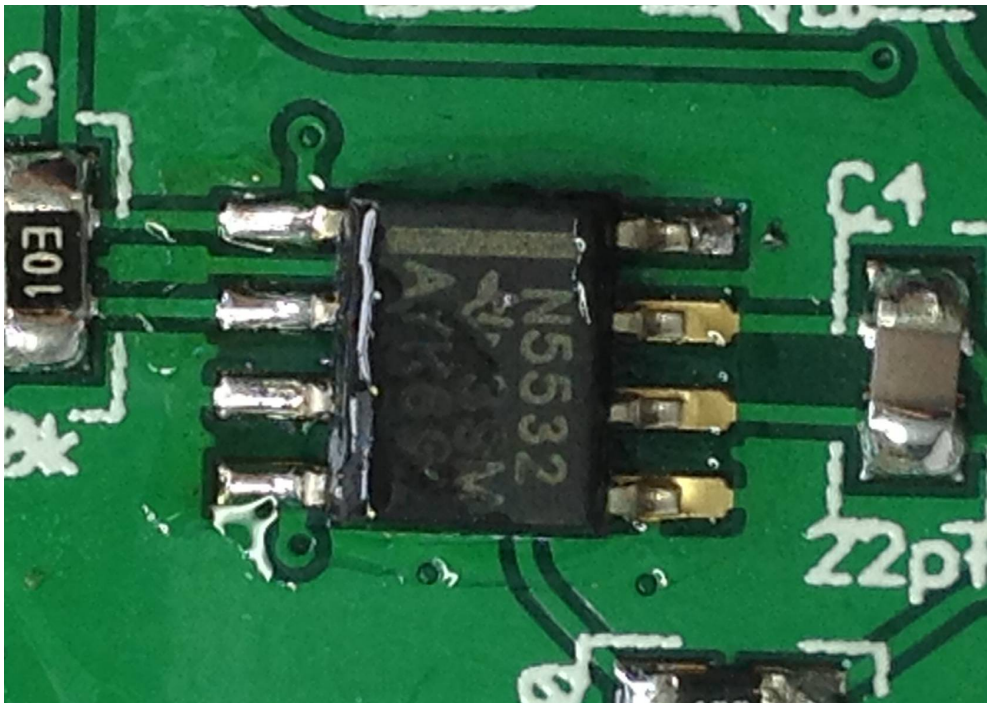


Illustration 24: NE5532 with 5 pins soldered.

Don't worry if you solder two pins together. Use the solder wick to remove the excess solder. Place the wick on top of the joined pins, & apply the tip of the soldering iron on top of the wick. Almost immediately you'll see the solder flow into the wick. Remove the soldering iron and the wick. Usually you'll find that there is still sufficient solder on the pads & the pins that you won't have to re-apply any solder.

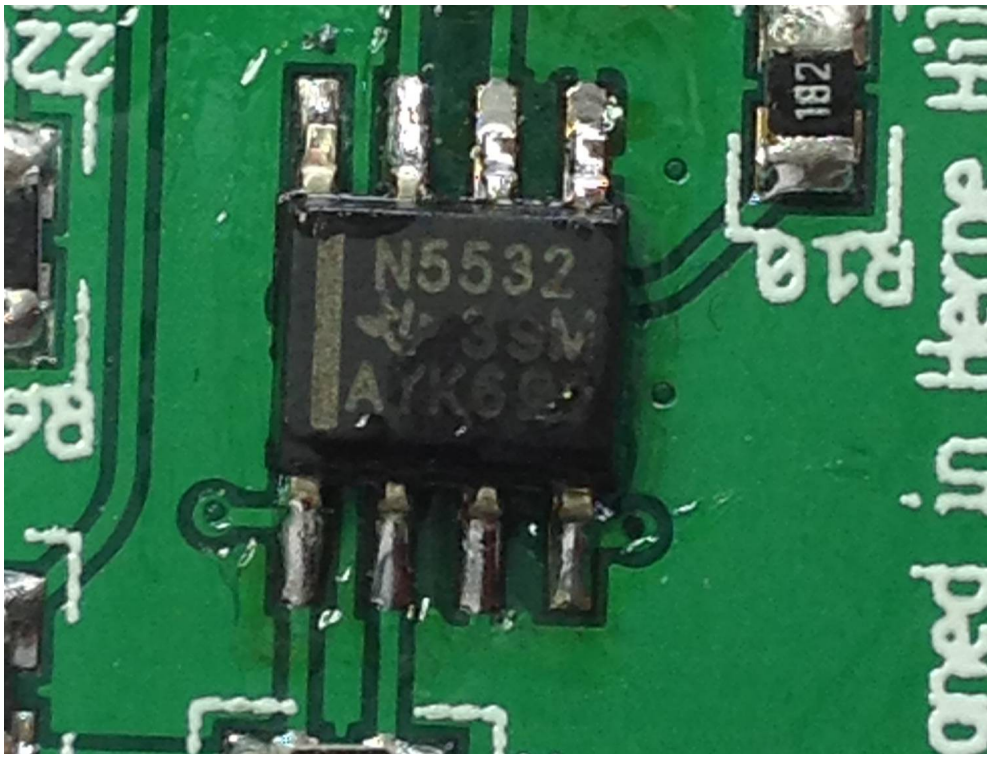
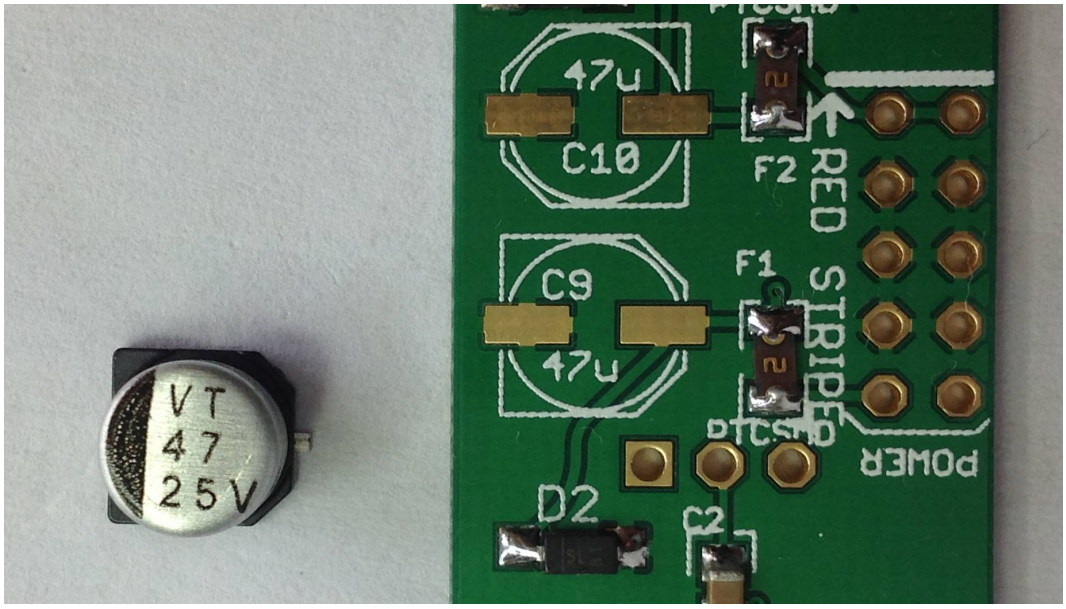


Illustration 25: NE5532 with all 8 pins solderedNote the flux covering the pins and the package. This will be removed during cleaning.

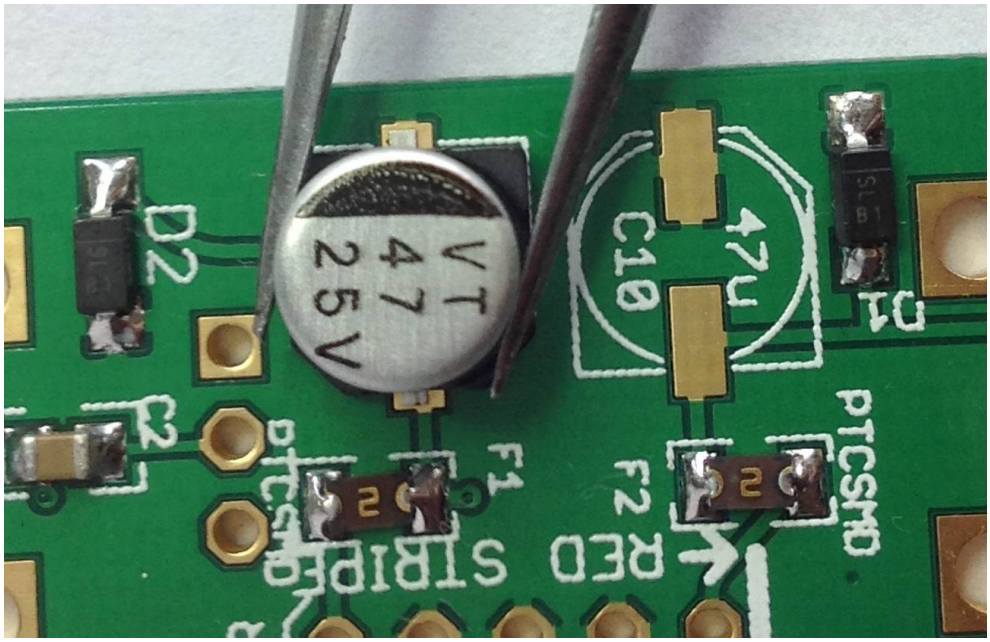
Repeat this process for IC2.

Once both IC's are soldered, thoroughly clean the PCB around both ICs, especially if you've used a flux pen. Take your magnifying glass/loupe & inspect the soldering on the IC pins. Check no pins are shorted. Using solder wick to repair any joints that are shorted.

Now it's time to do the two 47uF electrolytic capacitors, C9 & C10. These are polarised devices, so they have to be installed correctly on the PCB, but getting the orientation correct is easy to do with these. The positive end of the device has two diagonal sides in it's base, the negative end is flat. The shape of the base is also clearly marked on the silk screen on the PCB, so it's easy to get the orientation right.

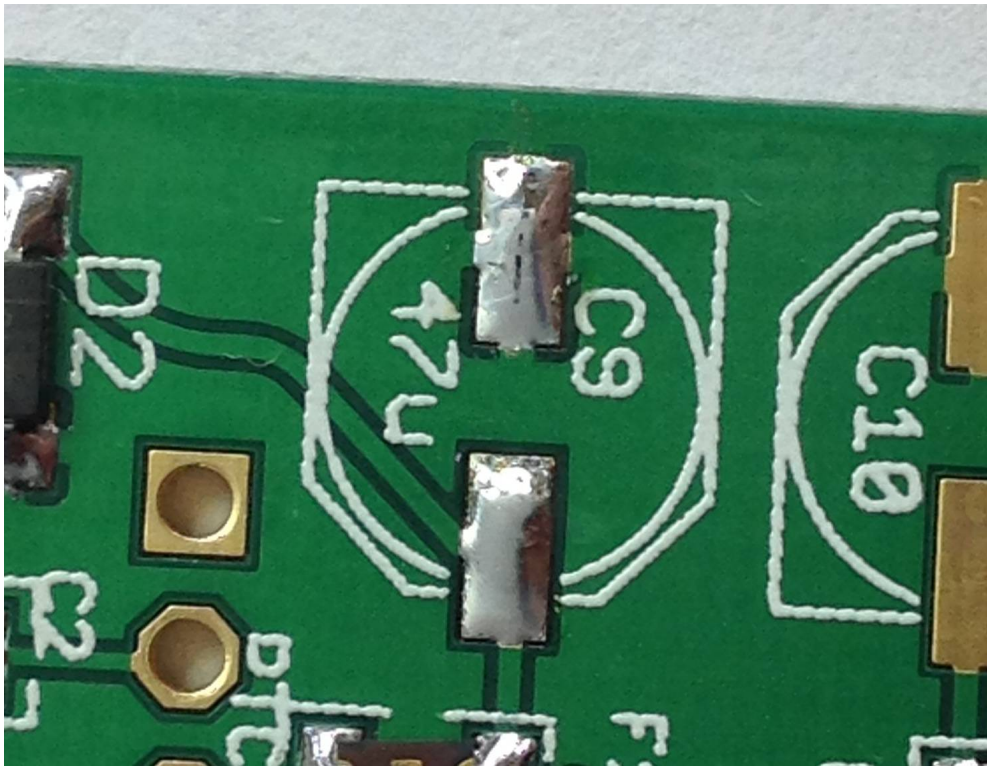


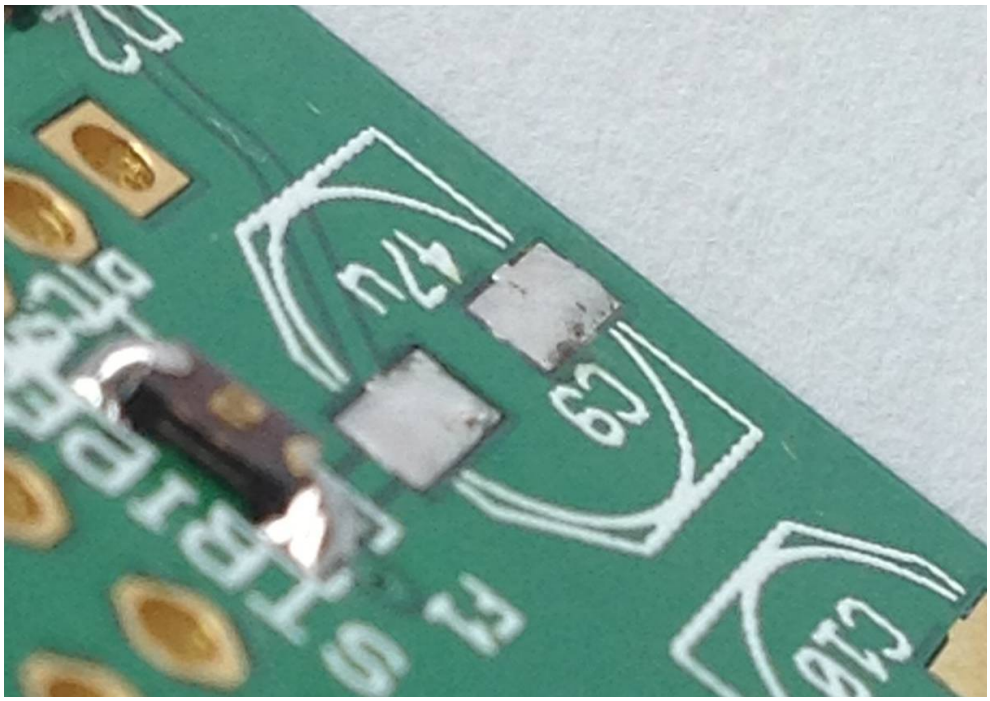
Again, remove one of the devices from the tape, offer it up to the PCB. Note how the pins sit on the pads. You'll probably notice there isn't a huge amount of room to get the soldering iron on the pad with the component on it. We have a technique for this.



Apply a small amount of solder to each pad. You don't want to apply too much here, as you want the device to sit squarely on the pad. Too much will prevent this. So once you've applied the solder,

remove most of it with some solder wick. Don't worry if it's not perfect, all we're looking to do it tack the device down for now.

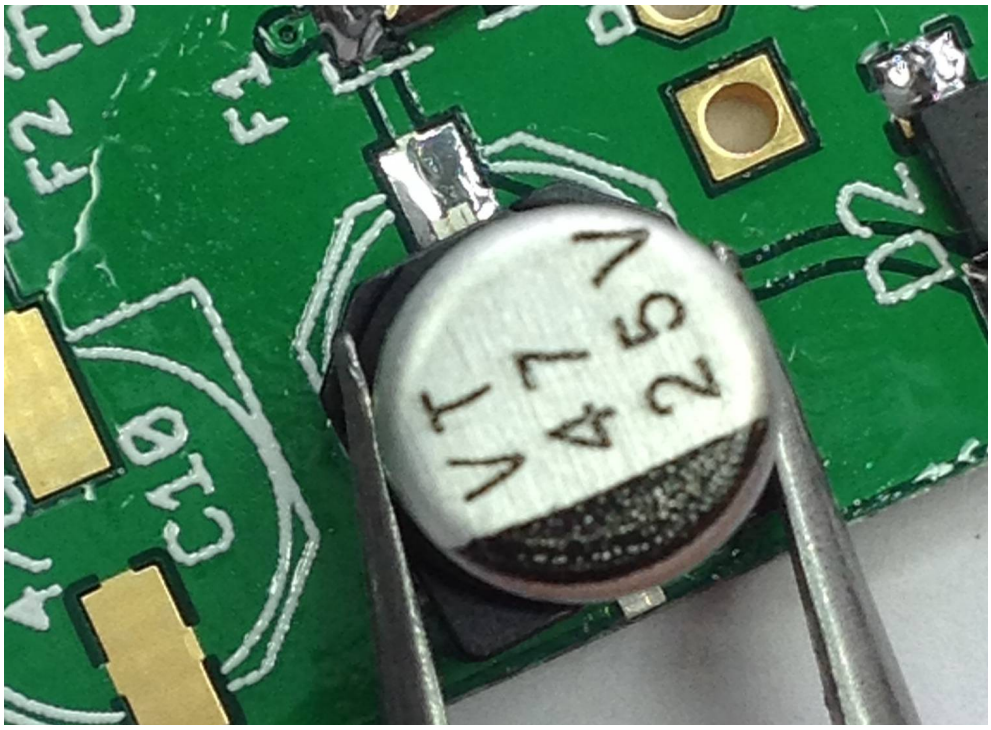




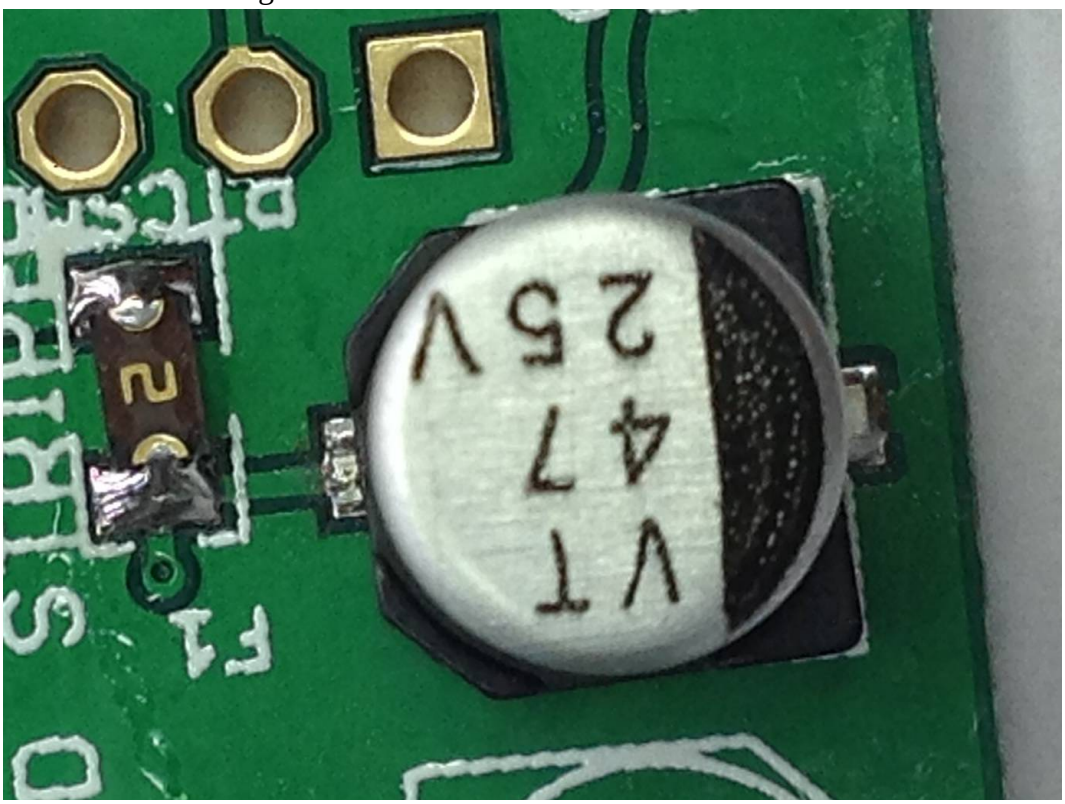
Now, if you're using a flux pen, apply some flux to the PCB pads and also to the capacitor's pins. The flux will help the solder flow into the joint between the pad & the capacitors leg.

Position the capacitor so that the sides of it's base are correctly aligned with the silk screen, but so

that one pin is at the centre of the device, just at the edge of the pad it'll be sat on once the device is installed.

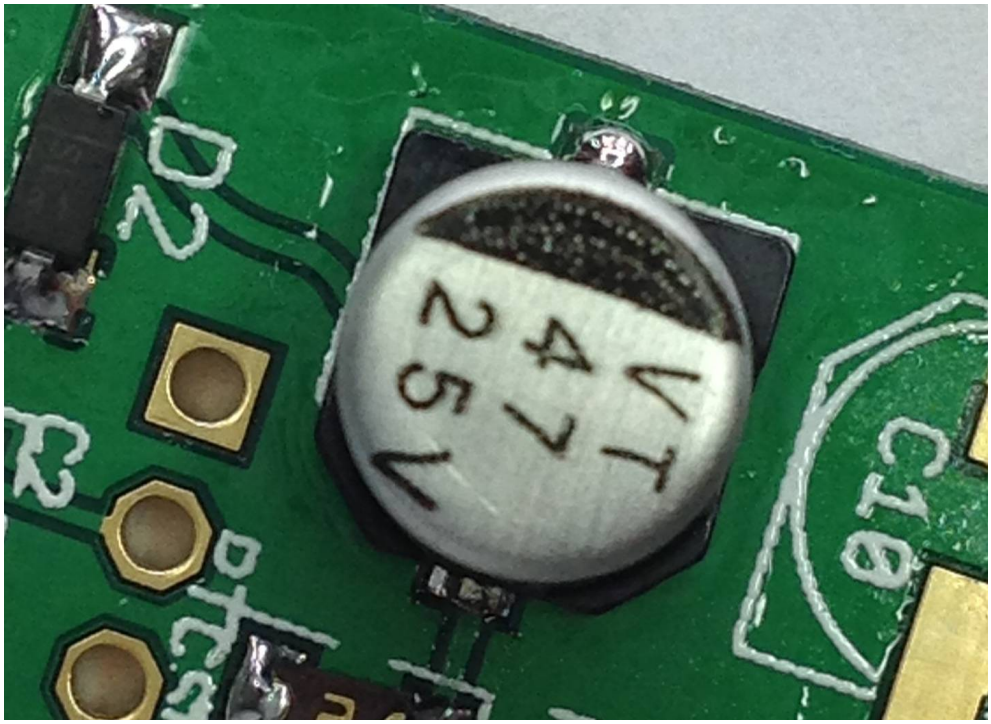


Apply the tip of the soldering iron to the other end of the pad. Once the solder on the pad melts, push the capacitor forward along the pad with the molten solder until it's pin comes into contact with the soldering iron tip and the capacitor is now aligned with the silk screen on the PCB. Remove the soldering iron.

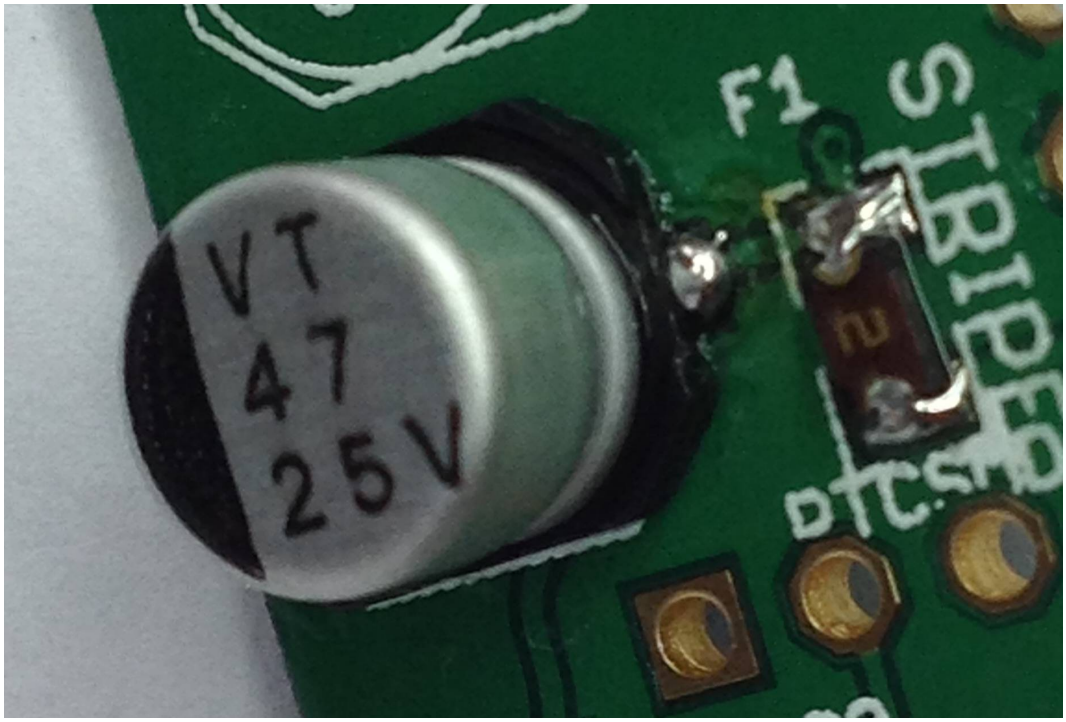


Check the capacitor's sides still sit squarely with the markings on the PCB's silk screen. If the capacitor is miss-aligned, re-apply the soldering iron to the pad & gently twist the capacitor until it's correctly aligned. The pin on the other end of the capacitor should be roughly in the middle of it's pad.

Now solder the second pin. This will be tricky as there isn't a lot here to get the tip of the soldering iron in contact with. Try to put the tip of the iron on the edge of the pin of the capacitor and the pad, whilst at the same time applying solder. A small amount of pressure on the iron may help. Applying some more flux here before you solder will help the solder flow down into the joint between the capacitor pin & the pad. You'll probably find that you'll melt the plastic base of the capacitor slightly. Don't worry about this. It can't be helped with such a small amount of the capacitor's pin protruding from the base.



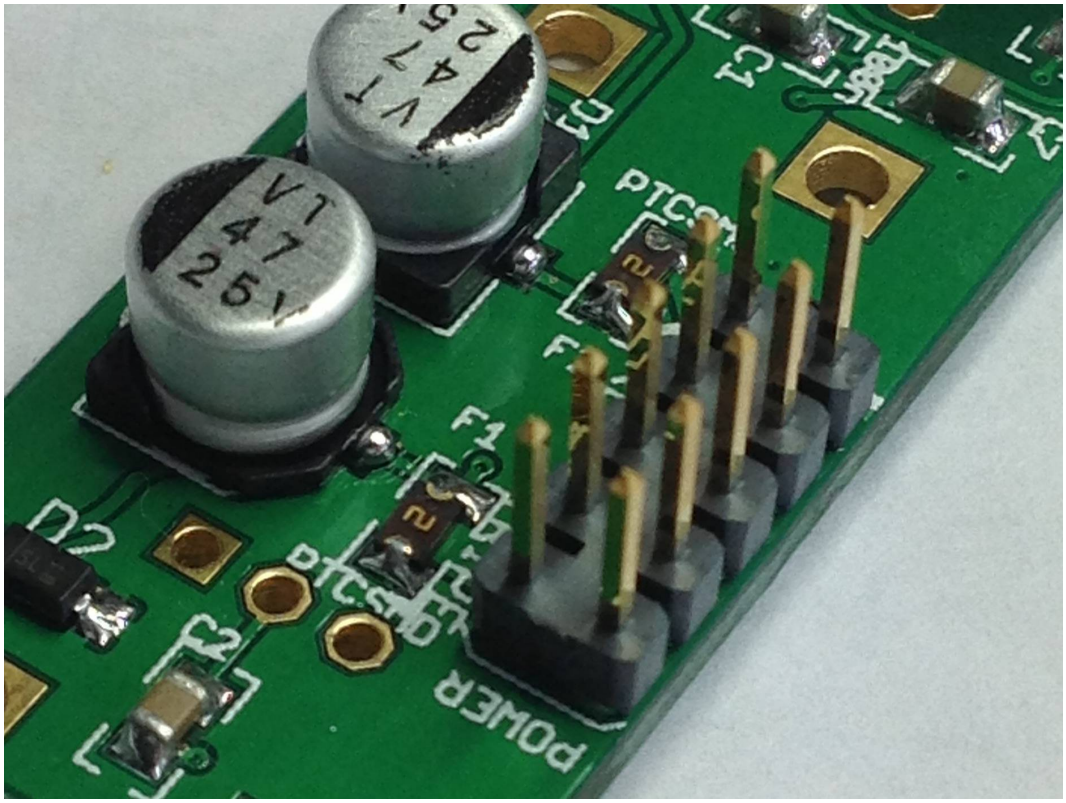
Once you've done that pad, go back to the first pad, apply some fresh flux if you're using it & add some more solder to the joint using a similar technique you just used on the second pin.



Repeat this process for the second electrolytic capacitor.

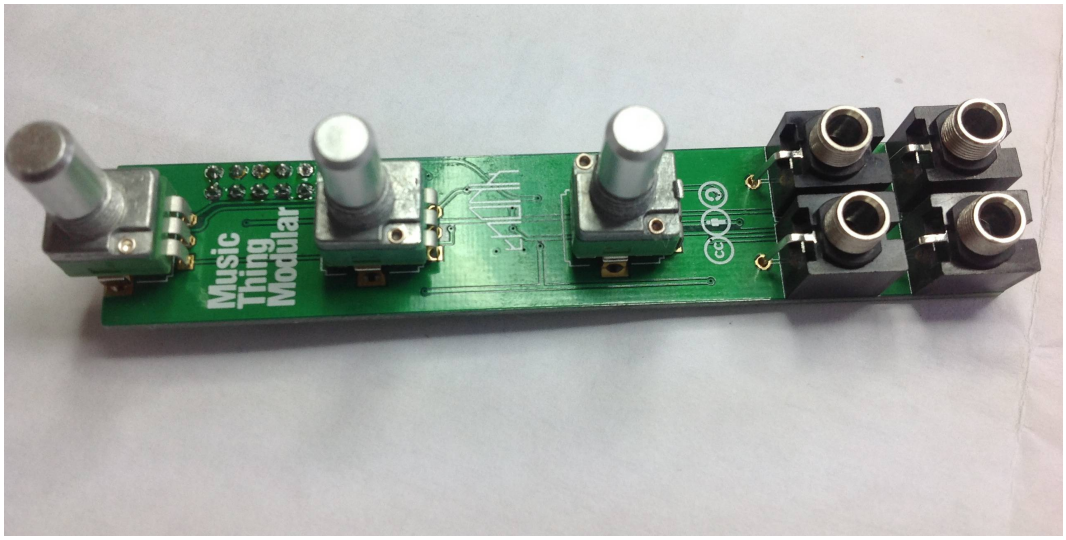
Thats the hard part over. Take some time to clean the board again, & inspect the joints for the electrolytic capacitors with the magnifying glass/loupe.

Now install the power connector. This is mounted on the same side of the PCB as the resistors etc. Solder one pin. Check the header is sat squarely on the PCB before soldering the other pins. Clean the flux residue from the PCB.

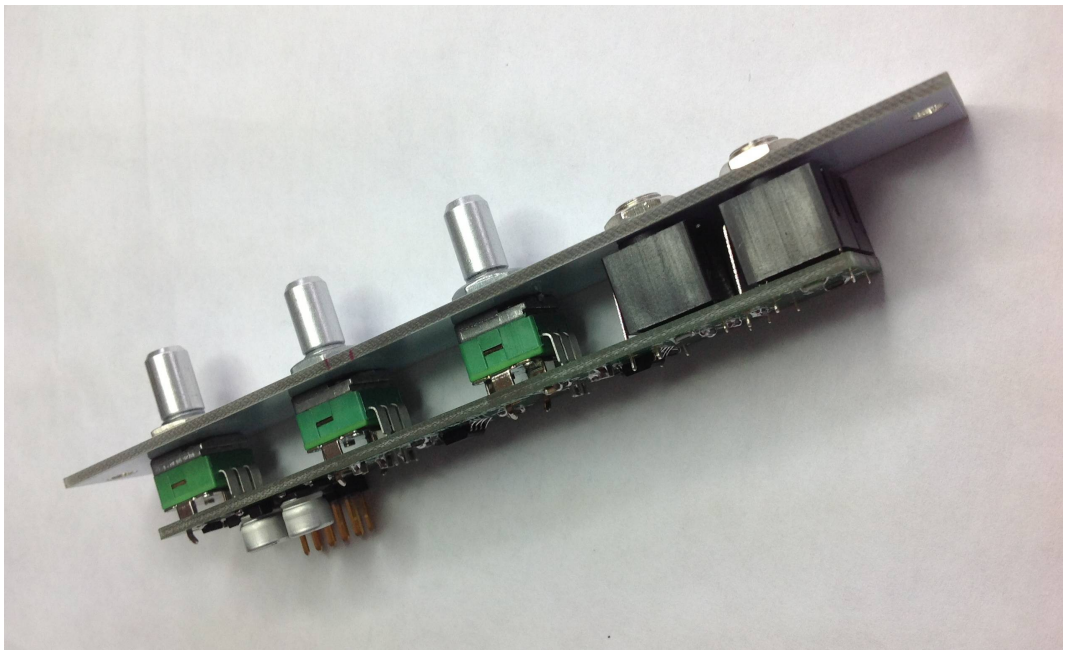


Now fit the pots and the jacks. These are mounted on the reverse side of the PCB to the components. Start by removing the nuts and washers if they're still fitted. If the pots have the small tab on them, remove it with a pair of cutters.

Fit the pots and the jacks to the board, but don't solder them.



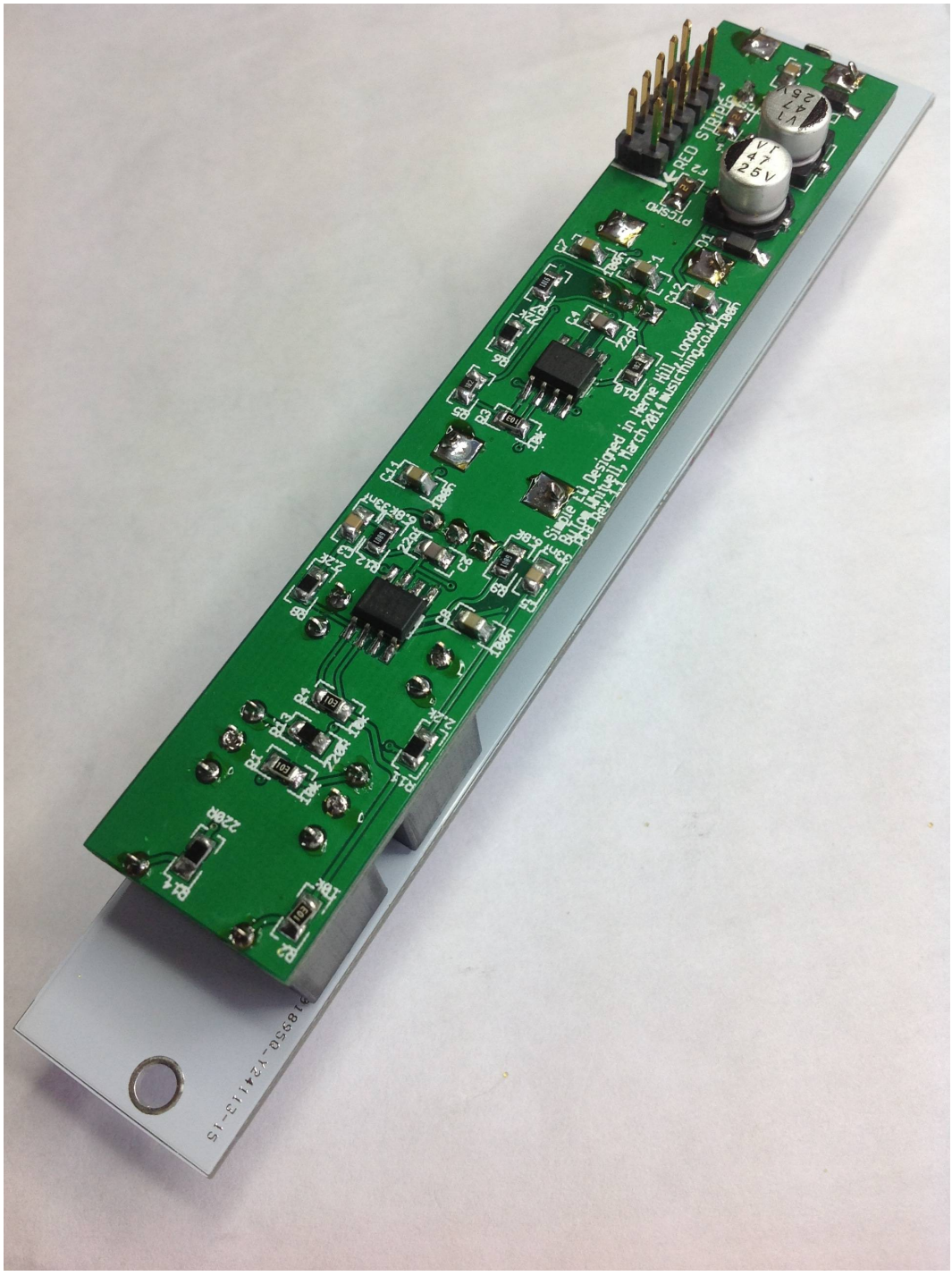
Fit the panel over the pots & jacks, then fit the washers & nuts onto the pots and jacks. Do the nuts up finger tight. The retaining clips on the pots should keep everything in place whilst you do this. The panel should now be parallel to the PCB.



Turn the assembly over so the component side of the PCB is facing upwards. Solder one of the Tilt jack pins nearest the bottom of the PCB. Turn the assembly on it's side & check that the jack is in contact with the PCB. If there's any gap between the jack & the PCB, apply the soldering iron to the pin whilst squeezing the panel & PCB together so that the jack is sat flush to the PCB. Solder the

bottom pin of the other Tilt jack.

Go back & check the panel & the PCB are parallel for one last time. This should be the case now if the pots are still fully clipped into the PCB. Solder the other pins of the jacks, the 3 pins and 2 retaining clips of each of the pots.



And thats the soldering done. All you need to do now is tighten up the nuts on the jacks and on the pots, then fit the knobs.

BOM

Qty	Value	Parts	Package / Description	Thonk / Mouser SKU
2	220R	R13, R14	R0805 resistor	667-ERJ-6ENF2200V
2	1.8k	R5, R10	R0805 resistor	667-ERJ-6ENF1801V
2	2.2k	R8, R11	R0805 resistor	667-ERJ-6ENF2201V
2	6.8k	R9, R12	R0805 resistor	667-ERJ-6ENF6801V
1	9.1k	R7	R0805 resistor	667-ERJ-6ENF9101V
4	10k	R1, R2, R3, R4	R0805 resistor	667-ERJ-6ENF1002V
1	22k	R6	R0805 resistor	667-ERJ-6ENF2202V
2	22pf	C4, C6	C0805 capacitor	81-GRM21A5C2E220JW1D (5% C0G NP0)
1	4n7	C2	C0805 capacitor	81-GRM2165C1H472GA01 (2% C0G NP0)
2	33nf	C3, C5	C0805 capacitor	81-GRM40X333J50D (5% X7R)
5	100n	C1, C7, C8, C11, C12	C0805 capacitor	81-GRM40X104K50L (50V, 10% X7R)
2	47u	C9, C10	Polarised Capacitor 25V or better, 6.3mm dia	667-EEE-FK1V470P (35V)
2	NE5532	IC1, IC2	SO08 op-amp	595-NE5532D
2	1N5819	D1, D2	SOD123 Diode	621-1N5819HW-F
2		F1, F2	0805 Resettable Fuse PTC	652-MF-PSMF020X-2
1		POWER	2x5 2.54mm Pin Header	649-67996-410HLF
4			Thonkiconn PJ301M Jack	http://www.thonk.co.uk/shop/3-5mm-jack-sample-bags/
3	B10k		Alpha B10K 9mm vertical linear potentiometers	http://www.thonk.co.uk/shop/alpha-9mm-pots/
3	Knobs		Black 1900h knobs (6.35mm round shaft)	http://www.thonk.co.uk/shop/knobs-davies-1900h-clone-metal/
1	Eurorack power cable		10-way to 16-way	http://www.thonk.co.uk/shop/eurorack-power-cables/

Build document & images by Graham Biswell, <graham@gbiswell.com>