BugBrand Workshop Oscillator Machine

Welcome!

We're going to put together a little circuit board noise machine and, along the way, get an introduction to some electronics and soldering techniques.

Don't worry if you've never done any electronics before – many people with zero prior knowledge have successfully built this machine in only a few hours. Also, while this handout contains quite a lot of information, much of it may go over your head for now – electronics can be seriously confusing to begin with (I certainly found this!), but hopefully you can come back to this information at a later date and gradually the pieces should begin to make sense. It is also fine, of course, just to build the machine and enjoy it!

All the parts are included in the kit – all you need is a soldering iron, some solder and a pair of wire cutters (more details in Sections 1).

The machine we'll make contains:

- Three oscillators – the sound sources
- A mixer – to blend the oscillators together
- A comparator – to keep the output stable when we use the starve control
- CMOS overdrive & tone – to shape the sound
- Jack output and mini-amp
- Circuit bends – 13 body contacts + power starvation

Parts of this document:

1. The parts of the kit & tools required
2. Soldering & building
3. Examination of the circuit workings
4. Further reading & final notes

Plus, at the end: Full Schematic / Parts Placement / Large image of Completed Board / Bill of Materials

If you have questions while building, probably the best place to ask is on the Muff Wiggler Forum Thread: http://www.muffwiggler.com/forum/viewtopic.php?p=25190
Or you can contact me directly: tom@bugbrand.co.uk

Good Luck!
& thanks!

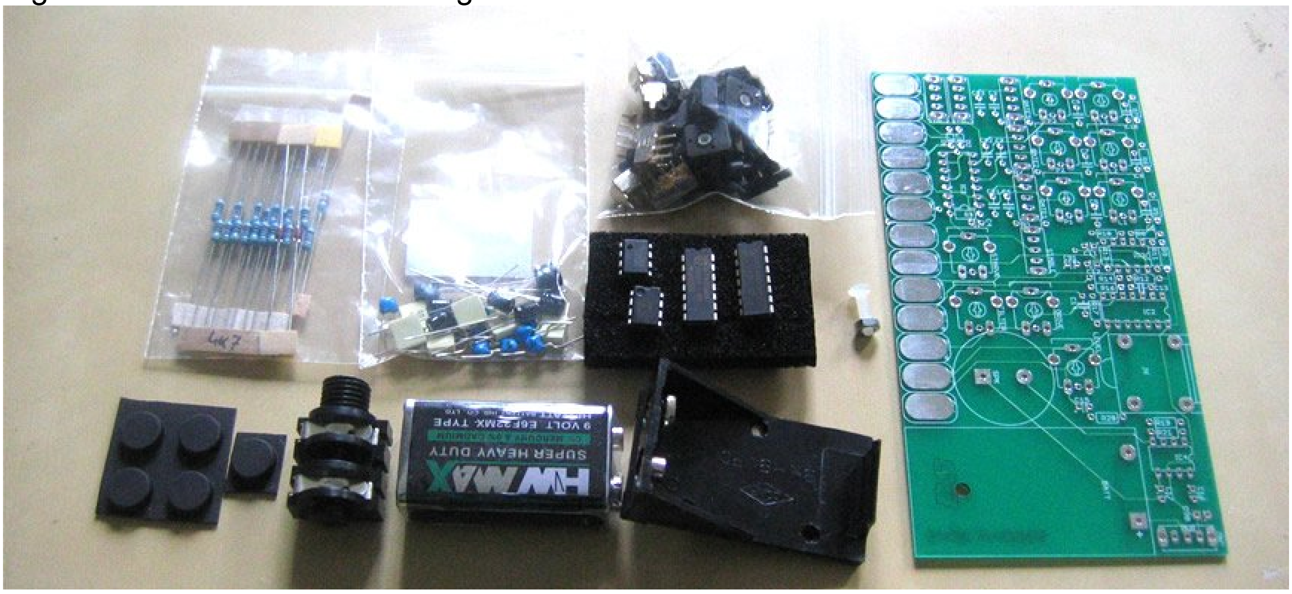
Tom Bugs
Nov.2008
www.bugbrand.co.uk

This document (in colour) can be downloaded from here:
http://www.bugbrand.co.uk/docs/bugbrand_workshop_osc_machine2.pdf
(rev2 – April2009)

BUGBRAND

1 – Parts of the kit & tools required

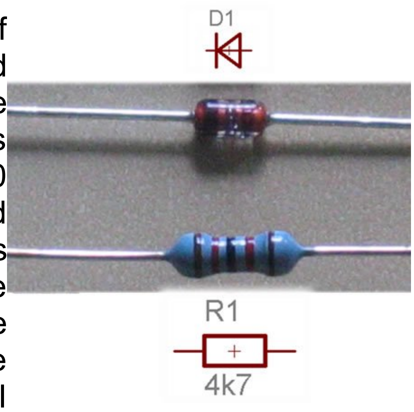
Carefully open up the main bag of parts – be careful as there are some small bits within! I'd recommend that you don't open up the smaller inner bags for now – wait until the building stage. Let's examine what we've got:



Contents (from top left): Resistors, Capacitors, Dials/Switches, ICs + Holders, Bolt, PCB, Rubber Feet, Jack Socket, Battery + Holder

You may want to have the circuit schematic sheet on hand now so you can begin to compare the parts discussed with what is drawn in the schematic – we'll introduce some symbols and terminology. By the way, a schematic is just a diagrammatic representation of the electronic circuit – it shows which parts connect together, though it doesn't show the actual layout of the printed circuit board (PCB) – more on that later. Let's also note that much of this introduction is quite specific to this kit but should still be relevant to electronics in general.

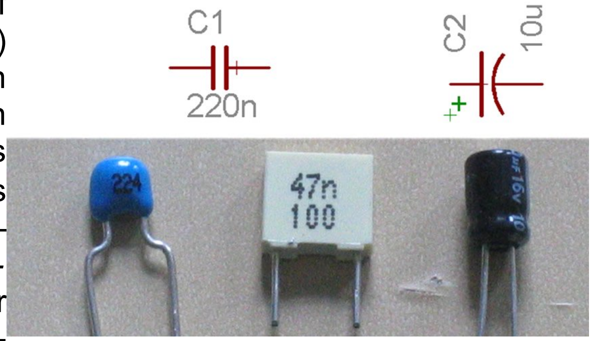
We'll begin with the ***Battery*** (which comes with a holder and bolt to attach the holder). This will be the power for the circuit – the battery is a 9volt supply and we'll use the terminology of *V+* for the positive connection and *Ground* (Gnd) for the negative connection. On the main schematic all points with the Gnd symbol are physically connected together in the circuit (and the same for the V+ points).



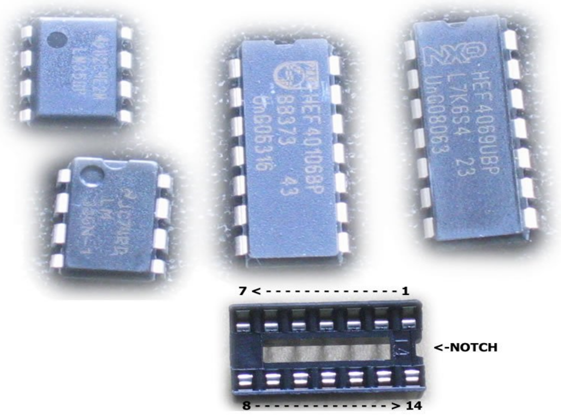
Now take the ***Resistor*** bag (which also contains a couple of diodes). Resistors oppose the flow of electrical current and come in many different values measured in *Ohms* (Ω) or, more usually, thousands of Ohms – kilohms (kΩ) – or even millions of Ohms – megohms (MΩ). By the way, we usually write 4700 ohms as 4k7 (we know that they're in ohms so there's no need to write the Ω symbol). If you look closely you will see various coloured bands around each resistor – these identify the particular value, but to make things easier I've supplied the resistors on paper strips which state the particular values (note – to avoid confusion, keep the resistors on the paper strips until you're about to use them!). Resistors are *non-polarized* components– it doesn't matter which way around they go.

In the bag there are also two ***Diodes*** – these allow current to flow in one direction only. Note the black ring around one end as this shows which way around to use them. These particular diodes are 1N4148 silicon diodes – the most commonly used in our sort of circuitry.

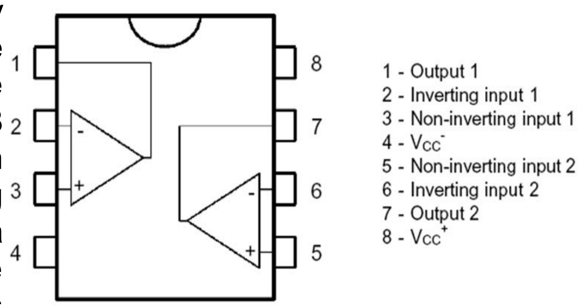
On to the ***Capacitors*** – there are three types of various values in the bag. Capacitors (or caps) store electrical energy and are measured in fractions of Farads – our typical values are in picofarads (pF), nanofarads (nF) and microfarads (μf or uf) - 1000pF equals 1nF and 1000nF equals 1uf. We have two types of low value caps here – the blue *dipped-ceramic* and the yellow *polyester-box* type (these types are pretty similar for our purposes) – note that these types are both non-polarized. There are also the black cylindrical *electrolytic* caps. These ones are polarized and *must* be used the correct way around – they have a positive side (the straight line on the schematic symbol and the long leg on the actual cap) and a negative side (the short leg on the cap – also there is a white stripe on this side with a minus symbol). We'll discuss more about the use of these capacitors when we look at the circuitry in the next section.



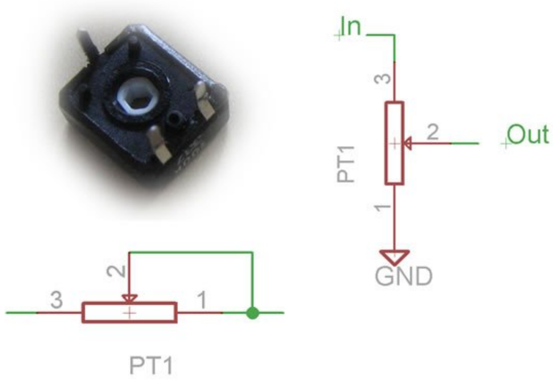
The next parts to consider are the chips or ***Integrated Circuits*** (ICs). These come in a little piece of foam along with IC holders for them – ICs can be sensitive (to heat or static electricity) so we generally attach a holder to the PCB first and only add the ICs once the circuit is complete (this also allows easy replacement if something breaks). There are literally thousands of different types of ICs available today and within the fairly nondescript black packages they actually contain miniature electronic circuits (composed of parts like resistors, capacitors, transistors etc). ICs have a number of pins (or legs) – typically 8 or 14 in our types of circuit – and these must be connected the right way around for correct operation (indeed, a chip will likely be killed if connected the wrong way around!). Examining an IC you will notice a notch at one end (or sometimes just a dot) – the pin to the left of this is Pin1 and the numbering from here goes down that side and then up the other.



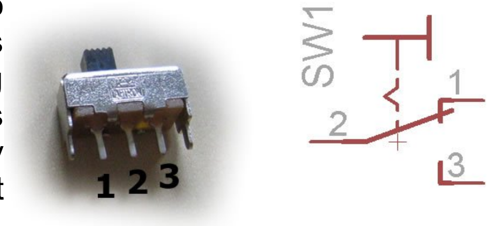
It is beyond our scope to look into the vast variety of chip functions available, but let's look at one type that we're using and see how the package and schematics hook up. We'll take the LM358 dual *Operational Amplifier* (OpAmp) as an example. OpAmps are useful electronic building blocks, each schematically represented by a triangle with two inputs and one output, and we can see from the diagram that the LM358 contains two of these. The two remaining pins are used to supply power to the chip. We'll look some more at the functions of OpAmps in the following section.



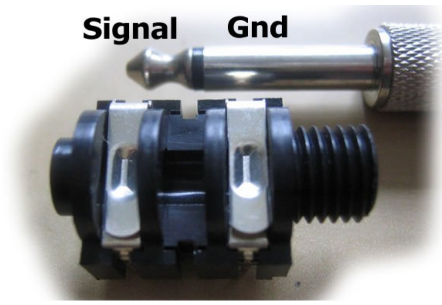
There's one bag of bits still to look at and this contains the dials or ***Potentiometers*** (Pots) and the ***Switches***. Looking first at the Pots – these are parts we use to control the behaviour of the circuit. While the ones we're using are small, they're pretty much the same as big dials on HiFi systems or synths. Pots are variable resistive elements with three pins and we'll use them in one of two ways. The first way (shown on the right) is called a *Potential Divider* (you can check Wikipedia for a more detailed explanation). There is a fixed value resistance between the outer pins (1 and 3) and then a *wiper* that moves between them as you move the dial. The way shown is used to attenuate a signal – ie as a volume control – turned one way the full signal is passed while turning the other way will make no signal pass. The second application involves tying two of the pins together to give a variable resistor, a useful means of controlling the behaviour of a circuit – more details on this in the circuitry section.



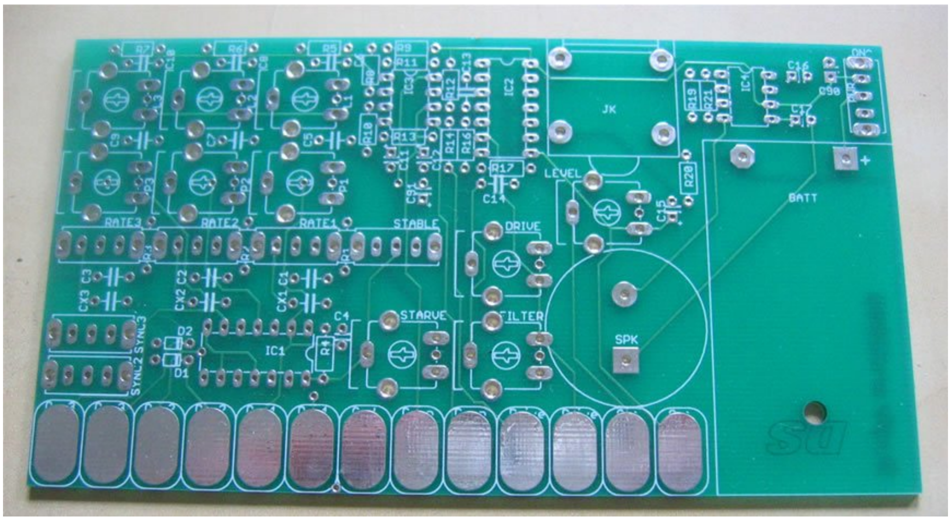
Mechanical switches are common in day to day life so should require little introduction. These particular ones have three terminals (the other legs are just for adding stability) – when the switch is pushed one way then pins 1 & 2 are connected together, while the other way connects pins 2 & 3, thus allowing us to select different routes within the circuitry.



There's not much left to examine – the ***Speaker*** should be self-explanatory but lets just look at the ***Jack Socket***. This is a switched mono 1/4” socket (ie for standard guitar cable). It has four legs, two on each side. The signal goes out on to the *tip* of the jack plug and the *sleeve* connects to the circuit Gnd. Note also that when no jack plug is inserted the legs on either side are connected – this allows us to automatically switch off the mini-speaker when a cable is plugged in. When the plug is inserted the horizontal pieces of metal are lifted up and thus the legs on the lower side are left unconnected.



Finally lets take a quick look at the ***Printed Circuit Board*** (PCB). This has been specifically designed for this project – laid out directly from the schematic diagram and fabricated in a far-away land! Its actually quite a complex procedure to do such a design and for most one-off circuits you'd simply use a material called stripboard (look it up) but for replicatable projects it really makes sense – everything is laid out nicely and building errors are minimized. If you examine the PCB you'll see *tracks* on the top and bottom of the board joining the different parts together. The silver coloured *pads* for the components make soldering easy (hopefully!). Now, people have asked why we make this project directly on the PCB without using a box. I've found that these on-board-circuits are strong and reliable. The time saved by avoiding putting it in a box is quite considerable and it also makes the project a good deal less complicated. Over the years I've been building things I've found that the most time consuming part of making devices is the boxing – wiring up pots and other controls. The wiring is also a place were faults can easily develop and my aim for this project has really been to make something that is almost guaranteed to work first time – the one thing that kills a buzz for electronics really quickly is when you finish building something and then it doesn't work!



Tools

There's another factor in my choice of mounting everything on the PCB – hardly any tools are required for the building! All that is required is a soldering iron (plus stand), something for cleaning the soldering iron (either a dampened sponge or brass wool tip cleaner), some solder (naturally!) and a pair of wire cutters (ideally a small pair though larger ones will do the job). There are more notes on soldering and where to get an iron at the end of this document in the appendix – have a check!

It is worth just mentioning how to set up a place to work – clear a good area so that things won't get lost (as mentioned before, keep things in their bags until you're going to use them) and find yourself a good source of light as this will help you do good soldering. It's also worth locating yourself somewhere with good ventilation, perhaps next to an open window – while the fumes from soldering aren't hazardous some people find them irritating.

2 – Soldering & Building
In the interest of keeping things interesting (!) we're going to move straight on to building before examining the workings of the circuit – but we'll come back to that in the next section.

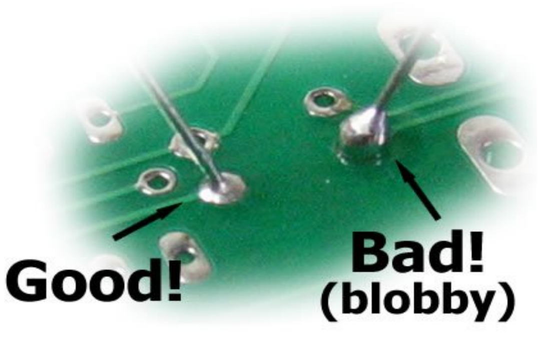
First let's talk about soldering. This is an area that often puts people off electronics, but with a little info its actually very straight forward! Bad soldering is probably the largest factor in crushing the enjoyment of electronics! Poor techniques will result in bad joints which don't work or are intermittent – so pay close attention to the soldering and your project should work first time.

It should go without saying that the iron is ***HOT*** and should be handled with care both to avoid burning yourself and items / people around you!

Soldering is a process where a metal alloy (solder) is melted to join metal surfaces together. For this we need *heat* – provided by the soldering iron. When you're starting out there really is no need to shell out on an expensive iron – all the irons I use in my teaching workshops cost under £5 (off eBay) and work just fine. What *IS* important is to keep the tip of the iron *clean* – a dirty tip, even on a decent iron, won't allow good soldering!

Cleaning – a clean tip should appear *shiny.* If the tip is dull / dark grey then it needs to be cleaned before soldering. The tip of a hot iron will gradually *oxidize* forming a thin layer of oxide over the tip which isn't good for heat transfer. We use a cleaning sponge or brass-wool cleaner to remove this layer and then melt a little solder onto the iron tip (a process known as *tinning*). After this the tip should appear nice and shiny – if not, repeat the cleaning process another time. The iron's tip should always be tinned when sitting in its holder – I always clean and re-tin the tip when I put down the iron after soldering and then clean it again when I pick it up.

Soldering – the aim of soldering is to heat up both the component lead *and* the PCB pad. Once they're heated enough, solder is melted onto the join and it flows around the component lead to form a solid seal. We must do this relatively quickly to prevent damage to the component or board (certainly no longer than 5seconds but preferably around 1second). We need to apply enough solder to fully cover the seal, but not too much as to make a bulbous joint (as this can hide problems). The join should appear clean and shiny – not a dull grey – and should be slightly concave.



So, follow this order each time:

- Clean the tip
- Tin the tip
- Heat the joint of lead and pad
- Flow the solder
- [Solder further joints if required, cleaning the tip as necessary (c. every 5 – 10 joints)]
- Clean and re-tin the tip before replacing the iron in its holder

If you do notice a bad joint then it is always possible to *reflow* the solder and fix the problem – either just reheat the joint until it flows properly or desolder it and reapply fresh solder.

Just to note – solder wire generally contains a core of a substance called flux. This can sometimes splutter a little when soldering and may leave a little residue around the solder joint. Flux is also responsible for the smoke that occurs when soldering – this smoke is generally harmless but may cause some respiratory irritation, especially for asthmatics, so if possible work somewhere with good ventilation (by a fan or window).

Desoldering – if you make a mistake you may have to remove some solder and this is actually a little tricky with these fabricated PCBs – so try to avoid mistakes in the first place! There are two methods possible – a solder-sucker or desoldering braid.

A solder-sucker is vacuum pump which sucks up molten solder. You depress the plunger, use the iron to melt the solder to be removed, place the tip of the sucker by the melted joint and then press the button to suck. Unfortunately, this technique tends not to work so well on fabricated PCBs due to the though-hole plating.

Desoldering braid draws up molten solder by capillary action. You put the braid on top of the solder to be removed and press the iron tip on top, melting the solder into the braid. It is generally necessary to pull the braid across the join to remove all the solder – a technique that takes a little practice!

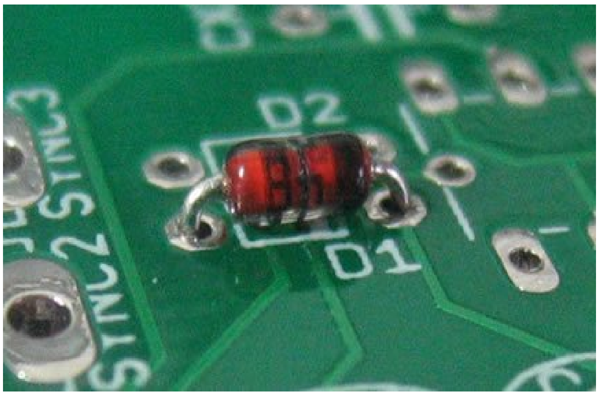
Ok – finally the time comes to build...
(you may want to turn on your soldering iron at this point to let it heat up)

But first:

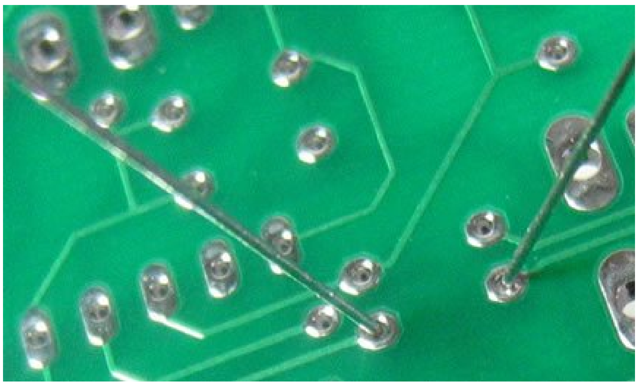
- Don't rush! Double check everything ***before*** soldering – ask if you have problems! Better to move slowly than to make errors.
- Keep that iron ***clean!***

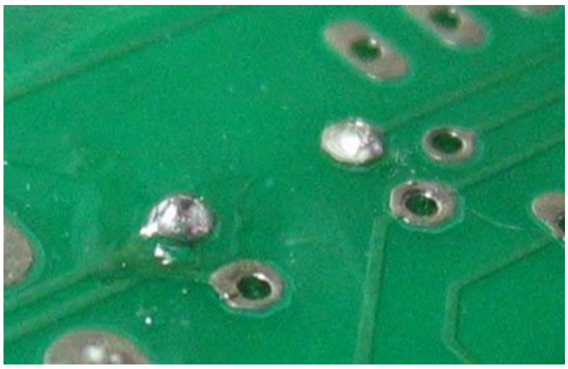
Proceeding in a methodical manner is of vital importance – do things by stages so that you don't confuse yourself and to avoid components getting mixed up (keep things in their bags until you'll use them). We generally build from lowest profile (height) component upwards – so we start with resistors and move from there...

We'll start with the two diodes – examine them and observe the black band around one end. Now locate the positions for D1 and D2 on the PCB and note that there's a line marked on them – this shows the correct orientation of the diodes (these are polarized components and must go the correct way around). You will also notice that one *pad* is round, the other is square – the band on the diode should go to the square hole. Remove one of the diodes from the paper strip and bend the leads at right-angles as shown – the bends should be tight to the body so the diode will fit snuggly into the PCB and fit right up to the board.



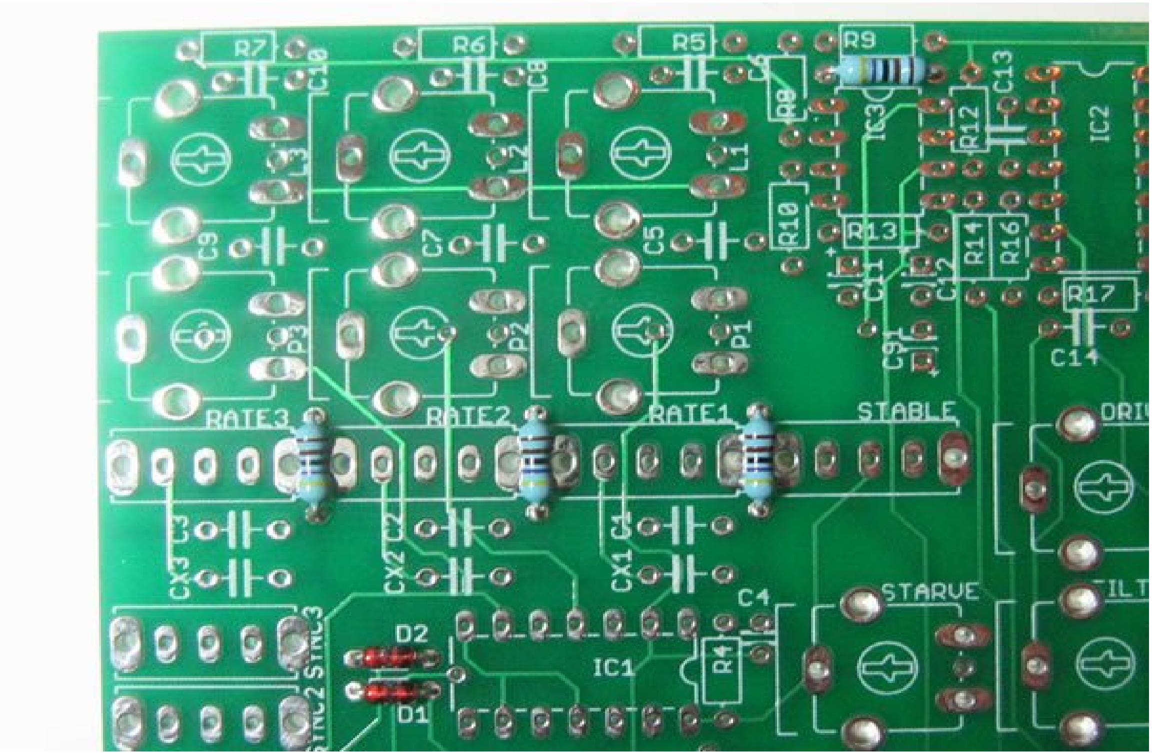
Turn the board over making sure the diode stays in position – you can splay the legs out slightly to prevent the component moving about. Now have a go at soldering, remembering the order: clean tip -> tin tip -> heat join -> flow solder -> clean, tin and replace. If all goes correctly you should have two nice clean solder joints. If you've done that ok then you can snip the leads off just above the join – nice and neat.



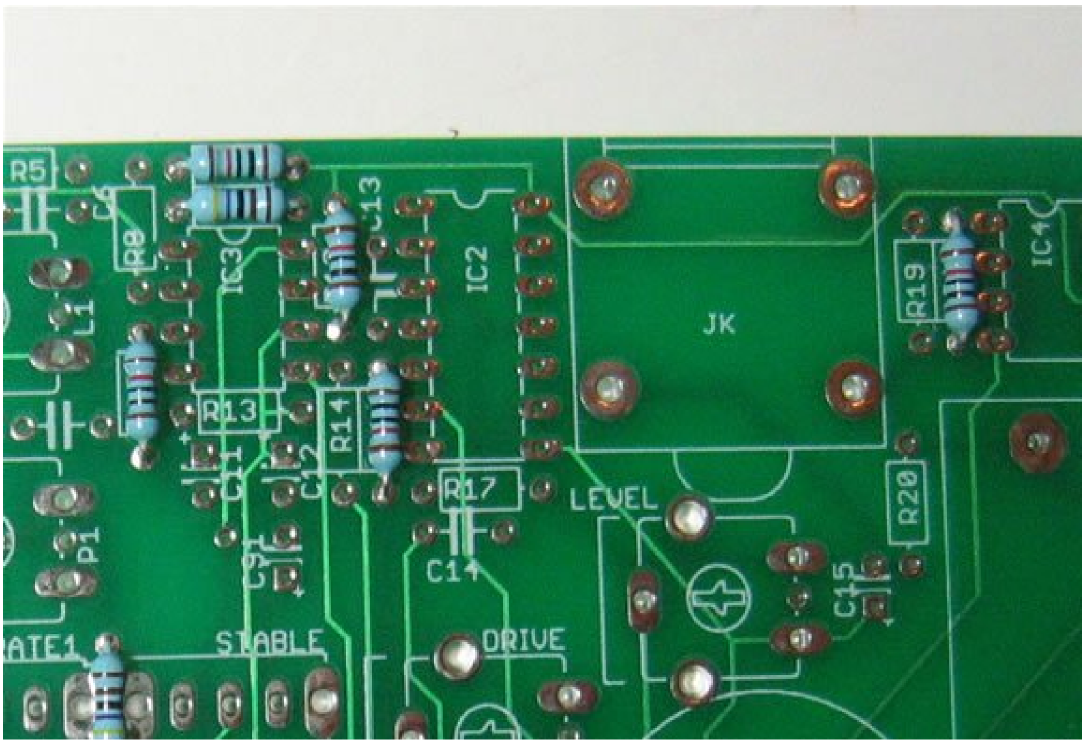


Do the same for the other diode – hopefully you'll be getting a feel for how the solder and iron work and feel... From there we move to the resistors – you may prefer at this point to start placing several components at a time before soldering them all. Though – don't put them all in at once or you'll find all the legs get in the way!

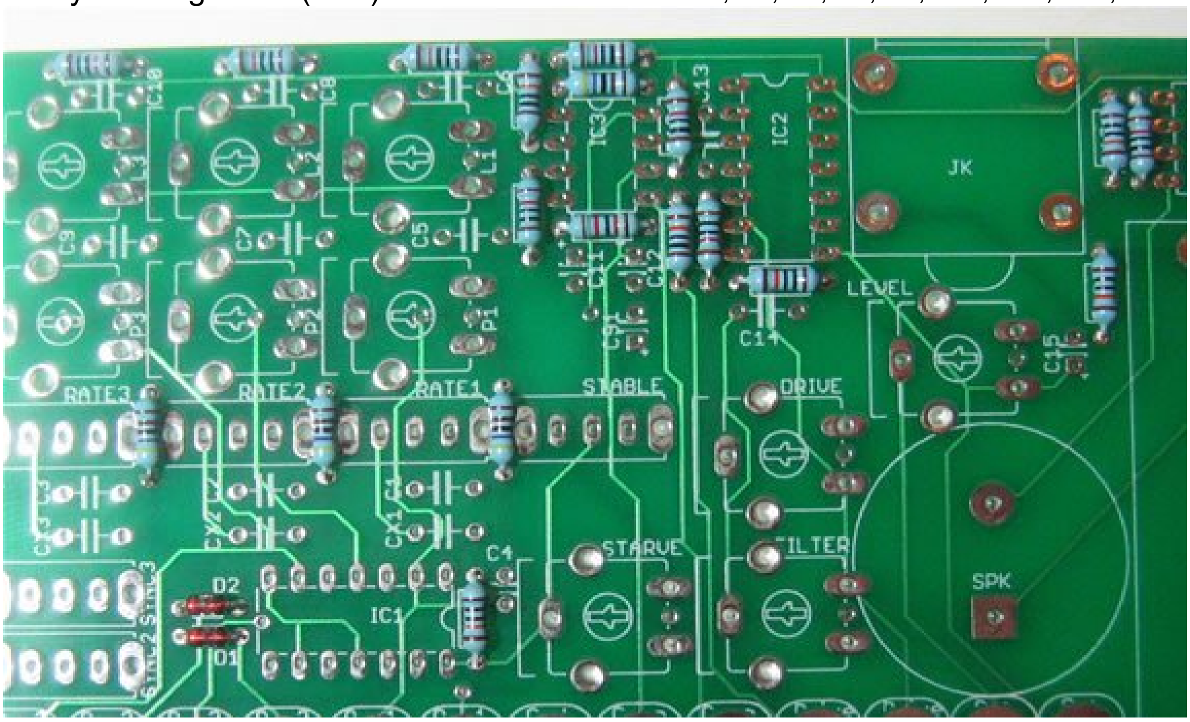
Take the four 4k7 resistors and locate their spaces – R1, R2, R3 & R11 – as shown below. Do the usual steps of bending the legs close to the body, put them in place and solder. Note that resistors are not polarized – it doesn't matter which way around they go. I find when I'm soldering that I turn the board round to find the best way to get the iron tip to the solder joint. Also be careful not to inadvertently solder up any extra holes with a careless movement of the iron!



Next its the 10k resistors - R9, R10, R12, R16, R21:

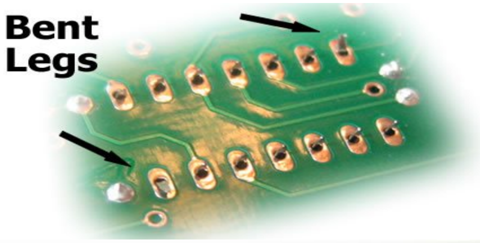


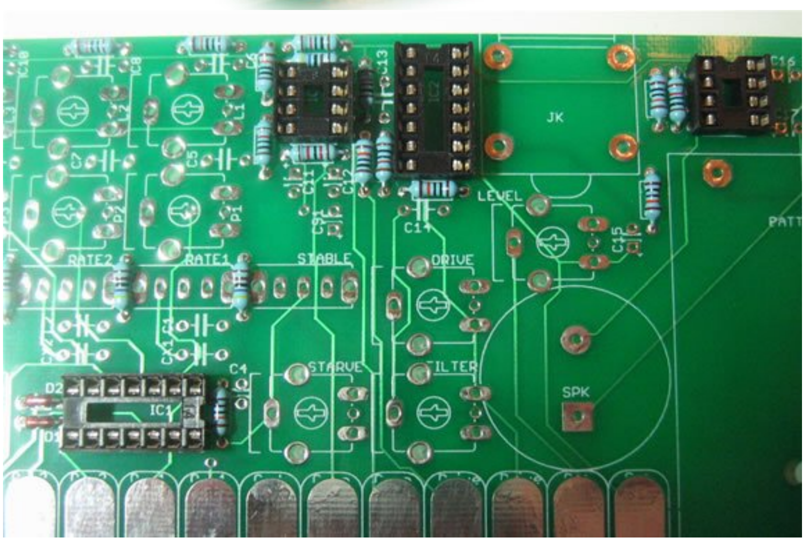
And finally the single 12k (R13) and the nine 100ks - R4, R5, R6, R7, R8, R14, R17, R19, R20:



Hopefully all your resistors are now in place and properly soldered!

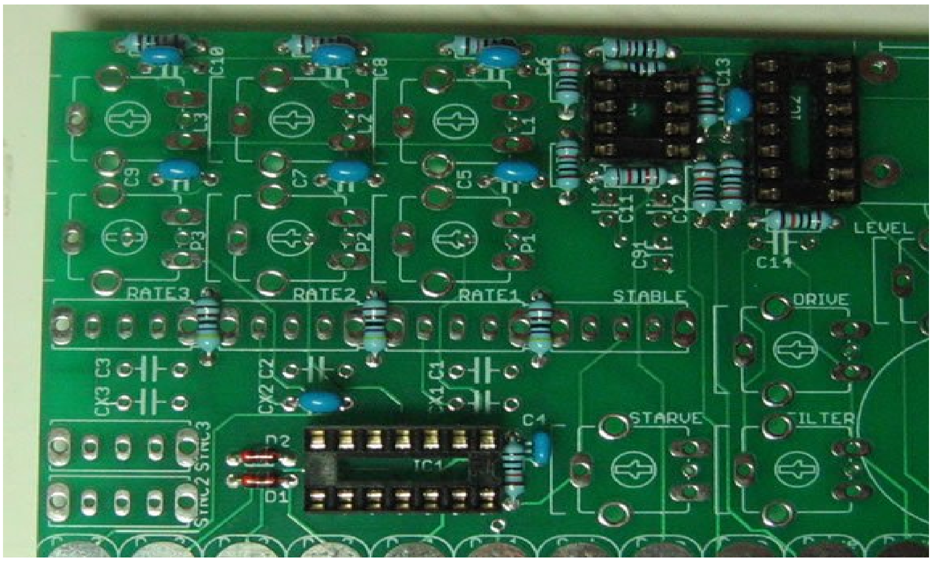
Next come the holders for the ICs which are held in the piece of foam. You should have two 8pin ones and two 14pin ones. Note that the notch at one end of each holder and, identifying the placements IC1-4, line up the holders with the PCB markings. Now – you'll probably notice that when you turn the PCB upside down to continue soldering that all the IC holders fall out! I suggest holding the holder in place with a finger, turning the board upside down and then bending out a pin on each side to hold the holder in place before soldering (do this one by one for each IC holder). Also make absolutely sure that all the legs have correctly gone through their holes (sometimes legs get bent under the holder) – if you start soldering before noticing such a problem then you've got a right task trying to desolder things and sort out the problem!



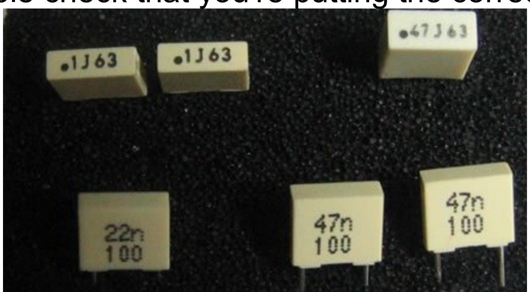


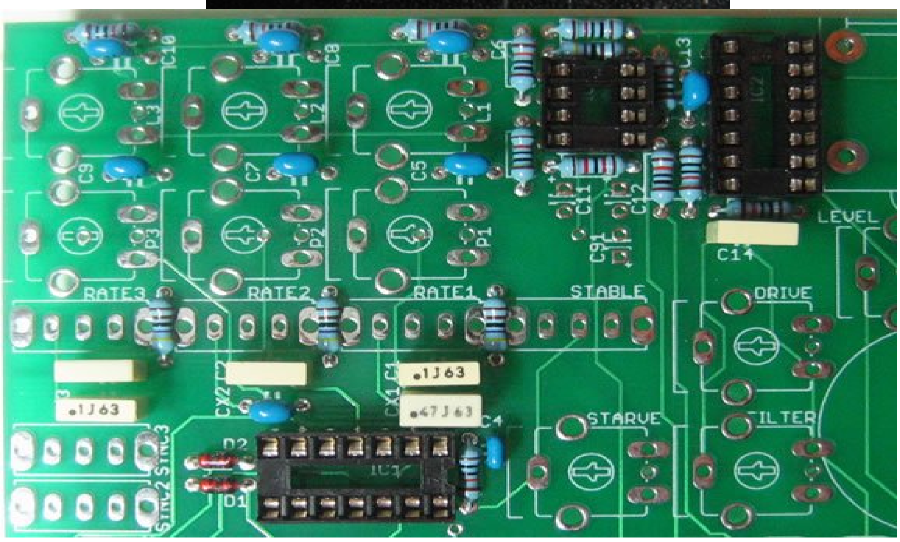
Capacitors come next – find the bag and empty them out. There are three types – the blue dipped ceramics, the yellow poly boxes and the black tower electrolytics. We'll put each group in separately, so start with the nine blue ceramics. Examining them you'll notice some numbers printed on the body – there should be eight the same with 224 and one with 221 marked (and narrower legs). In electronic terms, 224 means 22 plus four zeroes: 220,000 or 220nanofarads (C5, C6, C7, C8, C9, C10, C13, CX2). 221 means 220picofarads (C4). Note that these caps are not polarized – they can go in either way around, so locate the positions, put the caps in, solder and trim....

[in a few kits I may have replaced the 220n dipped caps with poly box caps (they function just the same in this circuit) in which case there will be eight extra box ones with the id mark .22j]



On to the poly box caps (again, not polarized). These have a slightly confusing number system (this actually varies a lot between different types and makes of capacitor) so go slowly here. You should have one 22n cap (C3) and two 47n caps (C2, C14) – both clearly marked. The other ones have the confusing .1J and .47J labels – that means 100n (C1 & CX3) and 470n (CX1). Double check that you're putting the correct cap in the correct place!

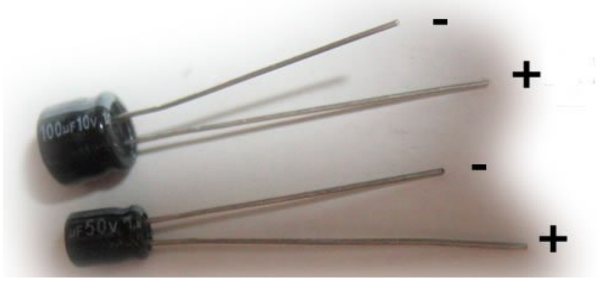


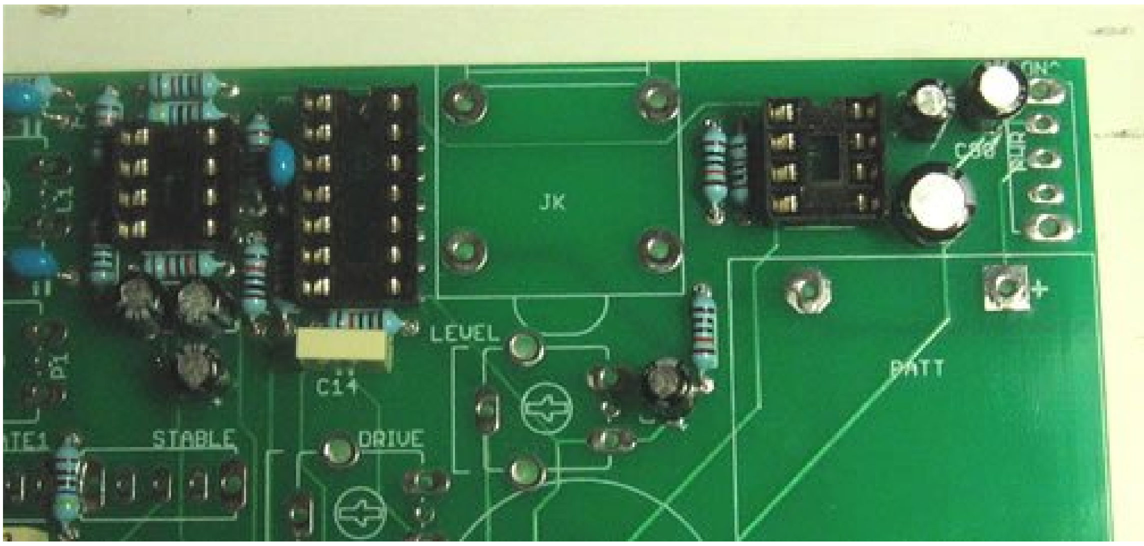


Now, the final electrolytic caps ***are*** polarized and so ***must*** be put in the correct way around (they can pop if connected back-to-front!). If you examine them you will see that:

- the value and voltage rating are marked in small letters on the side (eg 10u 16v – you can ignore the voltage marking)
- next to this is a vertical white(ish) strip with minus signs marked – this shows the *negative* lead which will go to the *round* pad
- the legs are also of different lengths – the short one corresponds to negative and the longer one is positive (goes to the square hole)

If it is hard to read the numbers off the side you can probably identify the different caps by process of elimination! There is one 1u (C15), four 10us (C11, C12, C16, C91) and then slightly larger caps for the one 47u (C90) and one 100u (C17).
Remember – the long lead goes to the square hole! Note also that for the 10u caps – C11, C12 & C91 are close together, but C91 faces in the opposite direction.

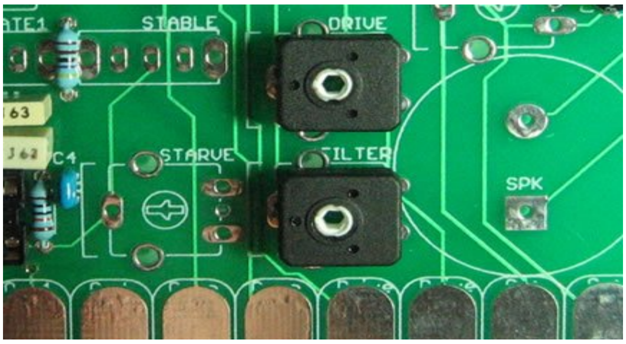


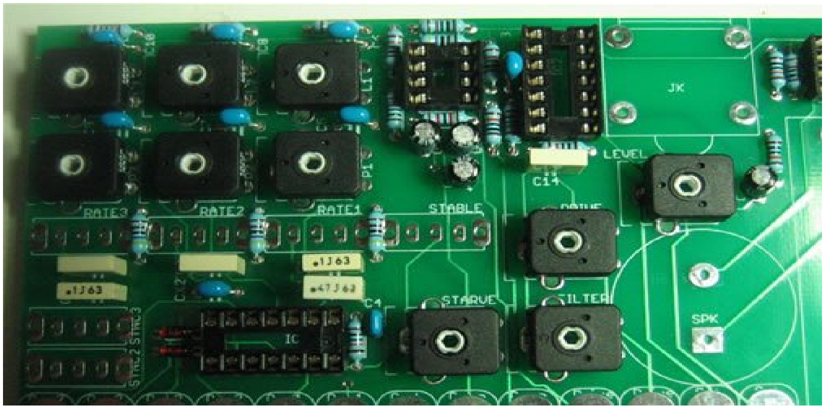


Time for the controls – the pots. You can leave off the spindle dials until the end, so put them back in the bag and sort out the different value pots. There should be one with 10k marked, one with 1M marked and then eight remaining all with 100k.

You'll probably find that, like the IC holders, the pots often don't want to stay in place before soldering, so here's a technique. Begin with just one pot, the 10k which goes in the *Filter* position. You should be able to turn the board upside down without the pot falling right out, but it may still move just a bit. To deal with this we solder just one leg to begin with. Then, pick up the board, still keeping it upside down, and put a finger of your left hand onto the pot from underneath. While pressing on the pot, reflow the single solder connection until it melts

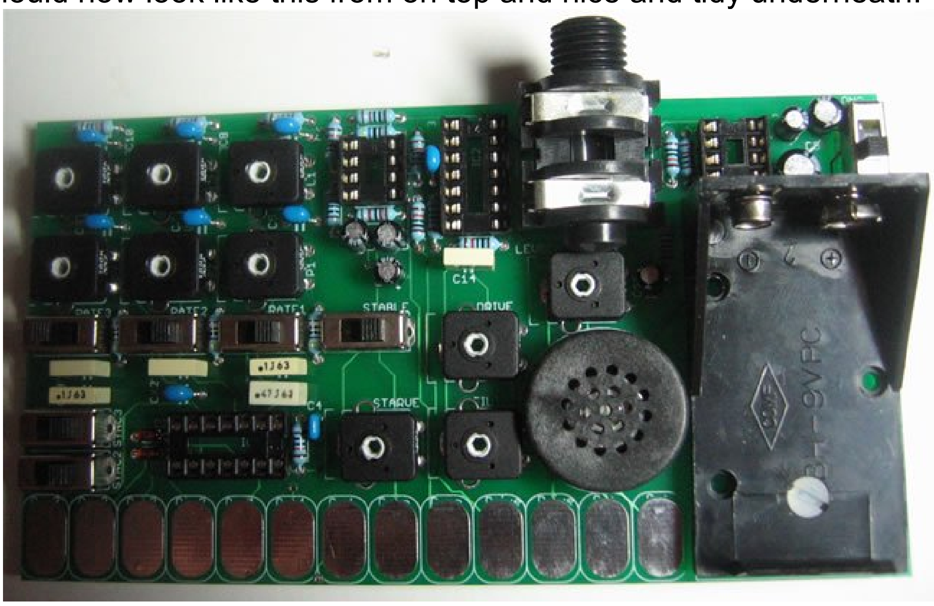
– at this point the pot (which you're pressing) will pop into the correct position! Once that's done, you can solder the remaining legs. Use the same technique for the 1M pot for the *Drive* control and then fill up the remaining positions with the eight 100k pots.
[Note – there are a few extra and unused holes around the pot positions – you can just ignore them]



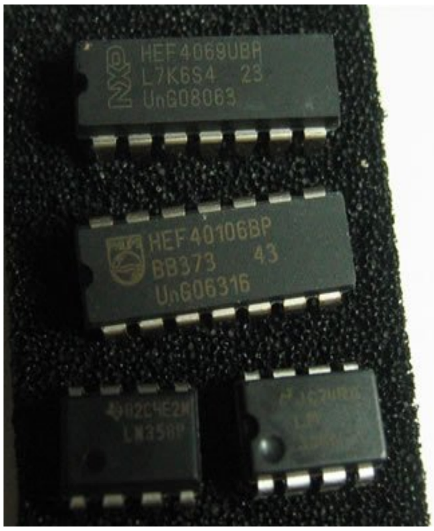


This method of soldering one leg, then pressing and reflowing the solder is used again for the final few components. We'll first do the seven switches, followed by the mini-speaker (you can bend the legs on that), then the jack socket and, finally, the battery box (which definitely does need to be pressed down during soldering). Once all those are attached successfully you are almost done (and maybe switch the iron off now!). Turn the board over again and, after trimming any stray legs, thoroughly eyeball all the joints to check that they are all good – just reflow or resolder any that don't look right. You can also add the spindles to the pots and the little nylon bolt and metal nut to hold the battery case firmly on the board. (leave the rubber feet off for now though and don't put in the battery yet)

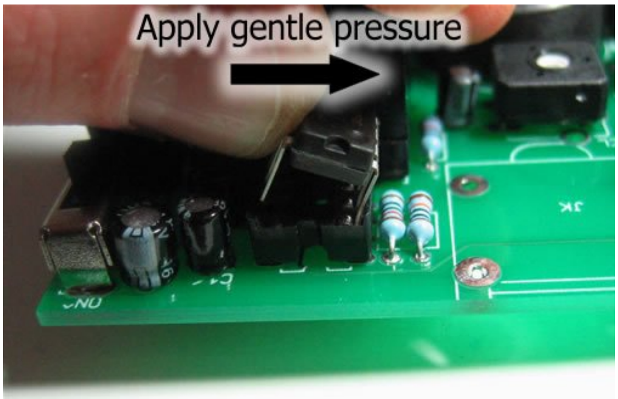
Your board should now look like this from on top and nice and tidy underneath!



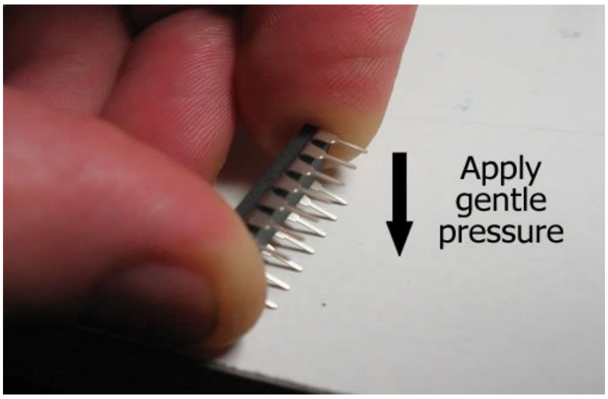
Now that all the soldering is done we can finally put the chips in. This is actually a little fiddly – ICs come with their legs slightly splayed meaning they don't always slot straight into the holders. You should also note that the two larger ICs are static sensitive – you need to discharge any static charge from your body ***before*** taking the chips from the foam. To ground yourself, simply touch something metal like: a radiator (an unpainted part), the metal case of a computer or something similar.



Lets quickly identify the chips – you should have two 8 pin devices, an LM358 (IC3) and an LM386 (IC4). Reading the codes off the top of the chips you'll notice there are lots of other letters and numbers – just ignore them! (they are date codes etc) The two larger chips should be identified as a 40106 (IC1) and a 4069 (IC2). Notice also the notch or dot at one end of the chip – this identifies the correct orientation and should match the notch of the corresponding IC holder. Make absolutely sure you place chips the right way round – powering up a chip the wrong way round will generally kill it instantly... Ok, the methods:



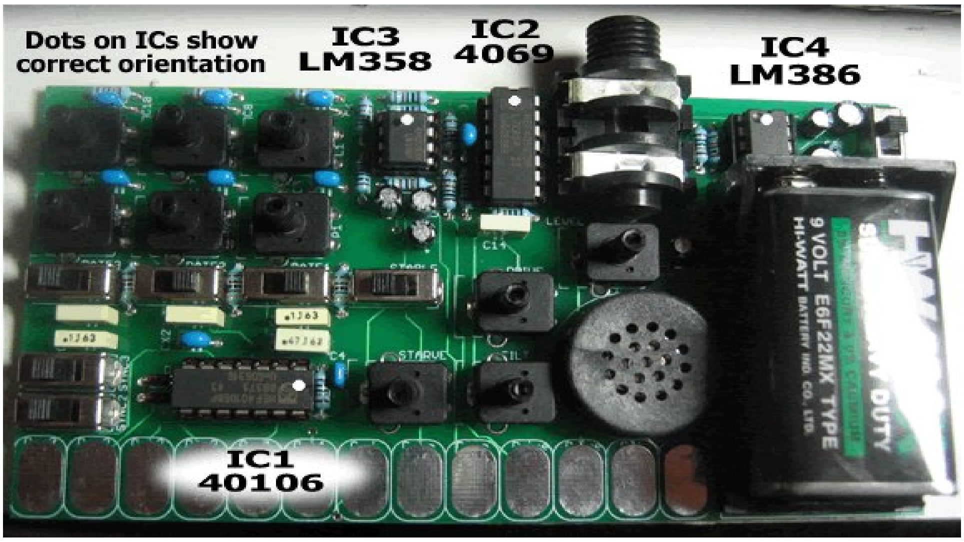
Technique 1 – pick up the chip and orientate it. Gently position the legs of one side into one side of the IC holder. Holding the chip at an angle of about 35deg, apply a gentle pressure in the direction shown so that the legs bend very slightly – don't push too hard or the legs will all bend out of shape! Gradually bring the other side of the IC down to locate the rest of the legs into their sockets – once things are in position, apply a final downward pressure to fully seat the chip.



Technique 2 – pick up the chip, holding it lightly at each end. Turn the chip on its side so it sits on one set of legs then apply gentle downward pressure to slightly bend the legs. Be careful to apply pressure evenly at each end and also to avoid bending the upper pins with you fingers. The legs need only be bent less than a millimeter so be gentle! Turn the chip over and do the same to the other side. The legs should now be ready to slot easily into the holder – orientate the chip lightly in the socket, double check all the legs are seated correctly and only then apply downward pressure.

Ok – time for the moment of truth! Unwrap the battery and clip it into place (make sure the polarity is right before switching on!). I'd say give one final check over before switching on, but you've probably already launched straight on in and are hopefully already making a hearty racket!

You can finish things off by attaching the rubber feet to the underside.

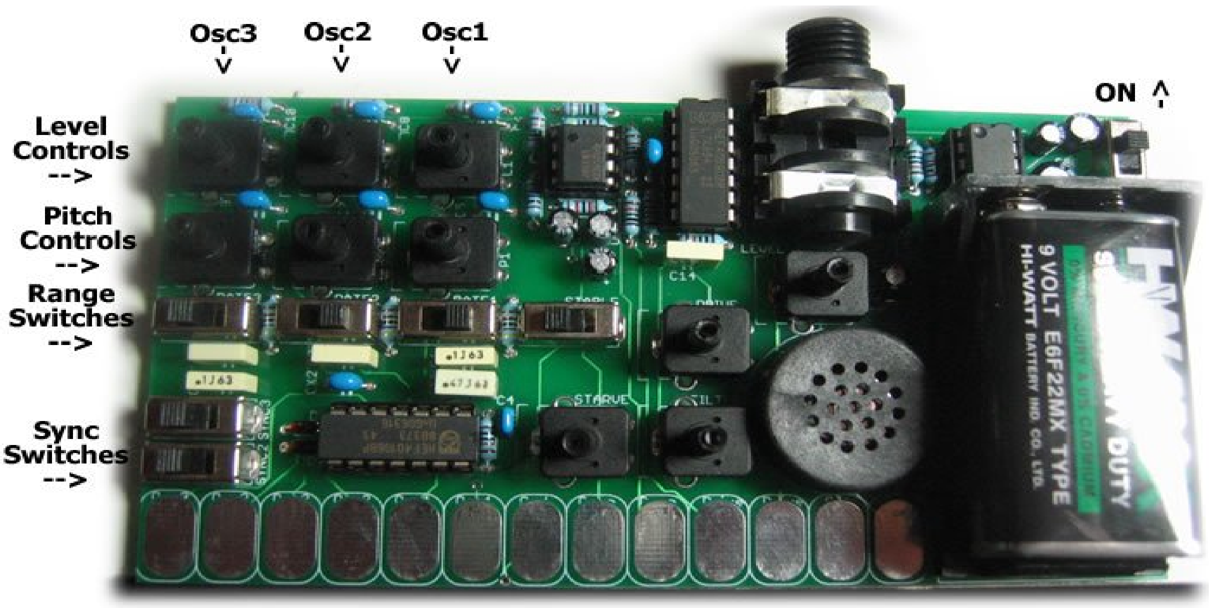


What happens, though, if it doesn't work? That's a tricky one and frustrating too... There are several areas to start checking:

- double check the orientation of the ICs and that the chips are in the correct positions.
- bad soldering – this is probably the most common mistake when starting out. Check and check again that all the joints are good – it can often take a while to spot (maybe take a break and come back a bit later). Pay attention to joints that look overly bulbous, as underneath the solder it may be that the component isn't actually poking through at all... If things look suspect then it may be good to desolder and then resolder to be sure.
- check also that all the leads are properly snipped close – I've seen short leads left which then bend over to cause a short circuit.
- In a similar manner, check that adjacent pads haven't been accidentally soldered together (something called a solder bridge) – again, desolder and repair when things look suspect.
- Check all the solder joints once again!

Section 3 – Workings & Circuitry

I always reckon that the best way to learn an instrument like this is to just dive in and discover! Controls are labeled (..a bit small..) so that may give some clues too. But, there'll come a time when you may want to know more. So lets look at the controls and then at the circuitry that's making it all tick.



What we've got is: 3 Oscillators -> Mixer -> Comparator -> Distortion -> Output / Amp

Oscillators: Each has a *Range* switch (Lo rate = left, Hi rate = right), a *Pitch* dial and a *Level* control that sets the mix balance (though behaves a bit weird too!)
There are also two *Sync* switches – these, when switched to the right, synchronize each Osc to the rate of Osc1. When switched left Sync is off. When Sync'd, the Osc must be running at a higher pitch than Osc1 or it won't make sound!
For more Sync info – look on the internet for an explanation.
The *Starve* control effects all three Oscs – turned fully clockwise means full power. This is coupled with the *Stable* switch – switched left gives the instable mode (more chaotic).

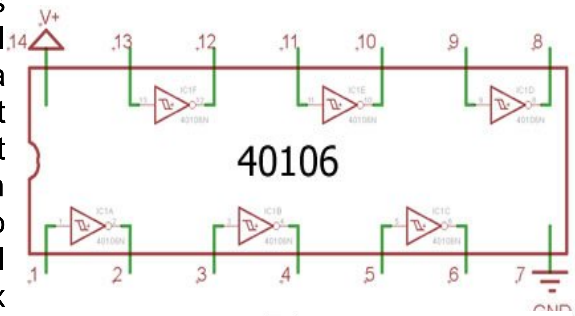
You will find that the sound and behaviour change quite a lot dependent on how loud you push things. The machine also becomes much more stable when sound is taken from the 1/4” output jack (which mutes the mini-amp) – though I really like the character of the speaker sound!

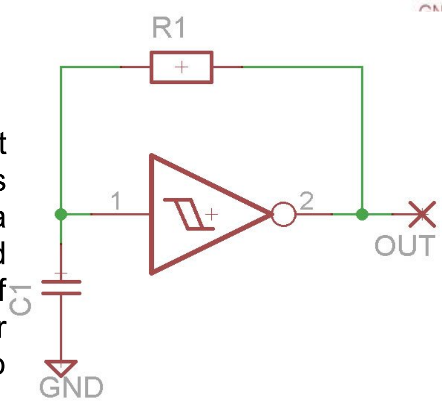
Experiment with the touchplates – these connect you straight to various points of the circuit. Lick you fingers, press several keys at once and explore!

Time now to get a closer look at the schematic – the scarily large diagram of components.. The good thing is that we can actually break it down into several much more manageable chunks. This is often the way in circuitry – you take several building-blocks and group them together into a larger piece. Many of these building blocks can also be used on their own or regrouped in different ways. You can find a lot more info about things on the internet, in particular by finding the *datasheet* for each chip (just search on Google) – these contain functional data and example circuits.

Just remember – points on the schematic with the little downwards *Gnd* triangle are all connected together (and the same for points with *V+* or *V_Strv*) – it just makes it a lot clearer not to actually draw lines joining all the points together!

Ok – let's start with the *Oscs* section which makes use of the 40106 chip. This chip has a technical name of *Hex Schmitt Inverter* (!) - this translates to a chip which, internally, contains six (hex) Schmitt Trigger *gates.* If we look at the 40106's datasheet we can see the internal layout of the chip – what pin does what. This chip can actually be made to produce, very simply, six squarewave oscillators! All you need is a power source, six resistors and six capacitors. We, though, add a bit more on top to give more flexibility.





Let's look at the basic Osc setup, focusing just on the first gate between pins 1 & 2. If we hook a resistor between pins 1 & 2 and then a cap from pin 1 to ground, we'll get a squarewave output from pin 2 – simple as that! The speed or frequency of the squarewave depends on the values of the two components – a bigger capacitor gives slower speeds (and vice versa), and a large value resistor also gives slower speeds.

This setup will just give us a fixed frequency output – to get a variable pitch we simply change R1 from a fixed resistor to a variable one (we actually also add a small value fixed resistor so the overall resistance doesn't reach 0 ohms). Examine the main schematic and you'll see that's how we've set up the Oscs – each one has a small value resistor (4k7) plus a 100k variable resistor between the gate pins. We can then see that there are switches to add in an extra capacitor for each Osc – this shifts the whole operation to a lower frequency range.

There's just one more part in the Osc section – Sync. What we're doing here with C4, R4 and gate IC1F is to make a very narrow pulse from the Osc1 squarewave – this can then be used to *reset* the other oscillators (the Sync points on the schematic join up).

Moving on to the *Mixer* – this is a very simple implementation of *summer* to combine signals and may be useful as a little standalone circuit. Each input section has a variable resistor set up as a potential divider – ie a variable attenuator – turned one way the signal is completely attenuated, while the other way the full scale signal is passed. The two caps around each pot, known as *DC coupling caps,* are standard in such designs, but we won't discuss their function. Each input signal passes through an input resistor (standard 100k) – the ratio between the input resistor and the opamp feedback resistor (R8 on the schematic) sets the output level. In this case the values are all 100k which means the gain is 1 – the level that goes in comes out the same the other end. We'll skip R9 and R10 other than to say these set up a mid-point voltage reference or *bias*.

The *Comparator* looks pretty similar to the mixer, but note that the signal is applied to the +ve opamp input and there's no feedback resistor. The purpose of the comparator is to transform the input signal, which may be of low level due to the Osc's power starvation, and turn it into a squarewave of full amplitude again – this makes sure the output level is pretty constant. With resistors R12 & 13 we set up a reference level and *compare* the input signal with this – the output of the comparator goes *high* when the input is greater than the reference and *goes low* when the input is smaller.

The *Distortion* section is an interesting little sub-circuit that can be used to emulate an overdriven valve amp – its great as a guitar stompbox! The surprise is that the chip used, the 4069, is actually a *digital* IC. This means that the outputs can either be *On* or *Off* which would suggest that we'd get a squarewave out of it - this would sound rough, not smooth like a valve amp! Well, with a resistor across the pins of one of the 4069's gates (which are laid out the same as those of the 40106) we do, in fact, get a nicely rounded output signal. Varying the resistor value (eg R16 and pot Drive) changes the *gain* and pushes the circuit into more overdrive – greater resistance gives more gain and vice versa. Another modification is to decrease the input resistor value (eg R14) to increase the gain.

There's also a simple tone control set up within the Disto section with the *Filt* pot and cap C14. This is a simple *Low Pass Filter* – it attenuates the high frequency parts of the signal – dependent on the resistance and the capacitor value. Turning the dial to 0 ohm resistance allows all frequencies to pass, while turning it to full 10k brings in the filtering action. The value of C14 could be changed to alter the frequency range – increase the value for a lower cutoff frequency and vice versa.

We discussed the action of the *Out* jack a bit in the first section – when no cable is plugged in, the signal passes straight through to the final Mini-Amp section and also the ground connection is made. The LM386 is a simple little power amp chip used in the final *Mini-Amp* section (a very standard setup is shown) – you can actually use a very similar setup to make a mini guitar amp head which can drive a full Marshall speaker cab.

Lets just finally look a little at the *Power* section. Each chip needs power to the appropriate pins and we generally add an electrolytic cap (C90) to stabilize the supply. By putting a variable resistor in series with the powerline we can create a starvation effect – this only works well, in this particular setup, effecting the supply to the 40106 Osc chip. This starved power also has a stabilizing electrolytic cap, but a switch is added to eliminate it if desired, resulting in much more chaotic Osc operation!

Ok – that was a very brief overview of the circuitry. As we said right at the start – you don't need to really understand what is going on to build or have fun with the circuit, but hopefully this overview can give some directions if you want to find out more.

4 – Final & Further Details
Well done if you've successfully built the Workshop Osc Machine! Hopefully you've enjoyed the experience and picked up some useful skills along the way. While the workings of the electronics may not be clear yet, you've hopefully realised that soldering isn't too tricky when you follow some basic approaches.

There is a wealth of further information available on the internet – dive in (if you want!) - it's the best way to learn!

Some useful / interesting websites:
www.bugbrand.co.uk – the BugBrand website with many sounds and some further electronic bits'n'pieces
www.cadsoft.de – home of the software, Eagle, that I use for schematics and PCB layout. A freeware version is available
http://electro-music.com/forum/index.php?f=112 – the EM DIY forum – a great place to ask questions
http://www.muffwiggler.com/forum/index.php – the MuffWiggler forum – home to the W.O.M. discussion thread and a great source of synthness

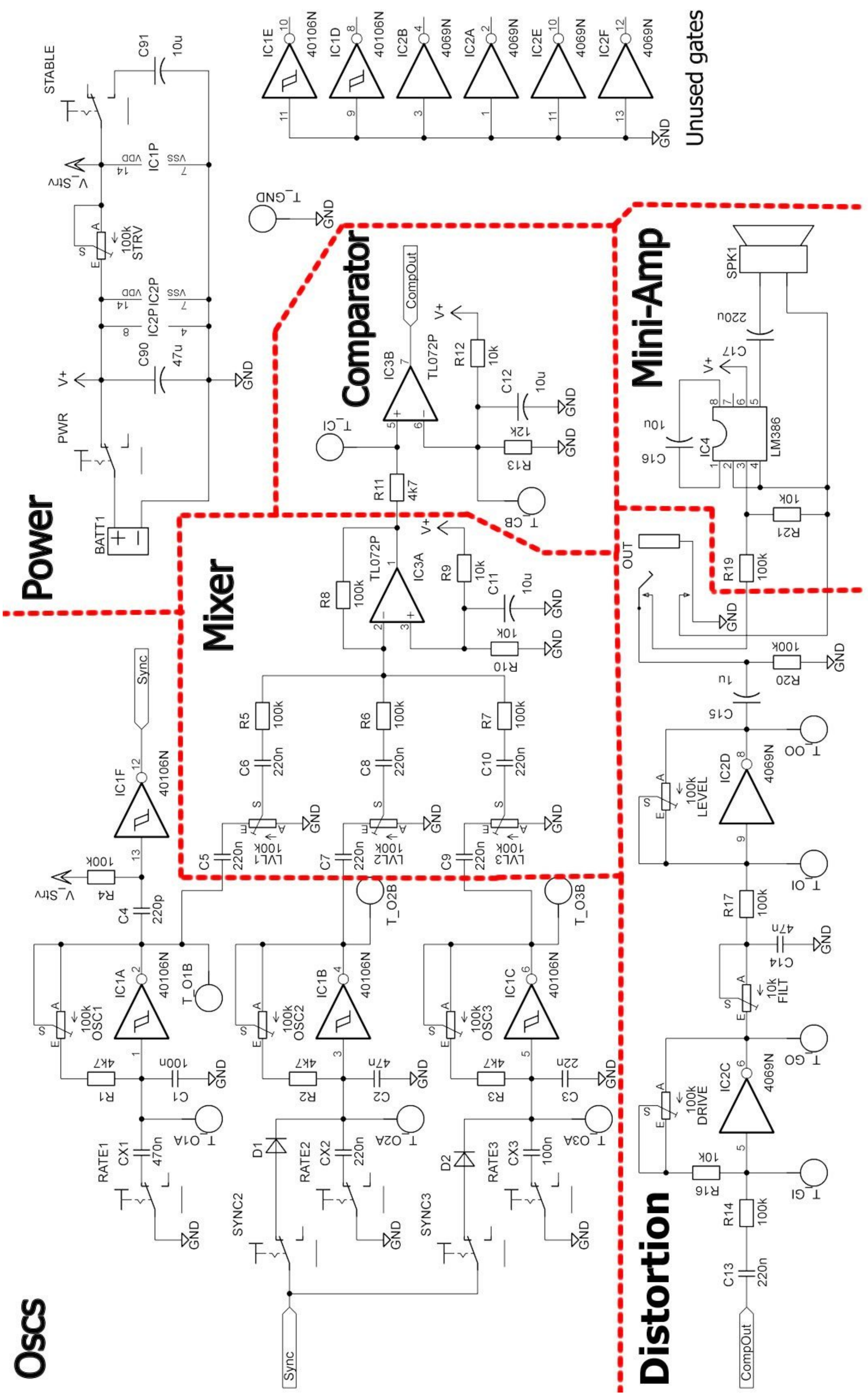
Parts:
www.rapidonline.com – a really great place to get parts in the UK (much better prices and stock than M*plin!)
www.futurlec.com – Thailand based webshop with very low prices (takes a while for delivery though)
www.mouser.com – good for the US

Further good electronics:
www.musicfromouterspace.com – Ray Wilson's site has a load of great info and circuits. In particular there is another simple little project (with PCB available) called the Wacky Sound Generator or you can get the more complex synth project, the SoundLab Mini-Synth.
http://www.cgs.synth.net/modules/ - Ken Stone's synth designs – a bit more complex to build, but very high quality & lots of useful info – PCBs are available.
http://www.geofex.com/ - lots of good info (but a bit all over the place)
http://bleeplabs.com/thingamakit/ - Bleeplabs' fun project with light control chaos!

Bill of Materials:
(shown in order of placement)

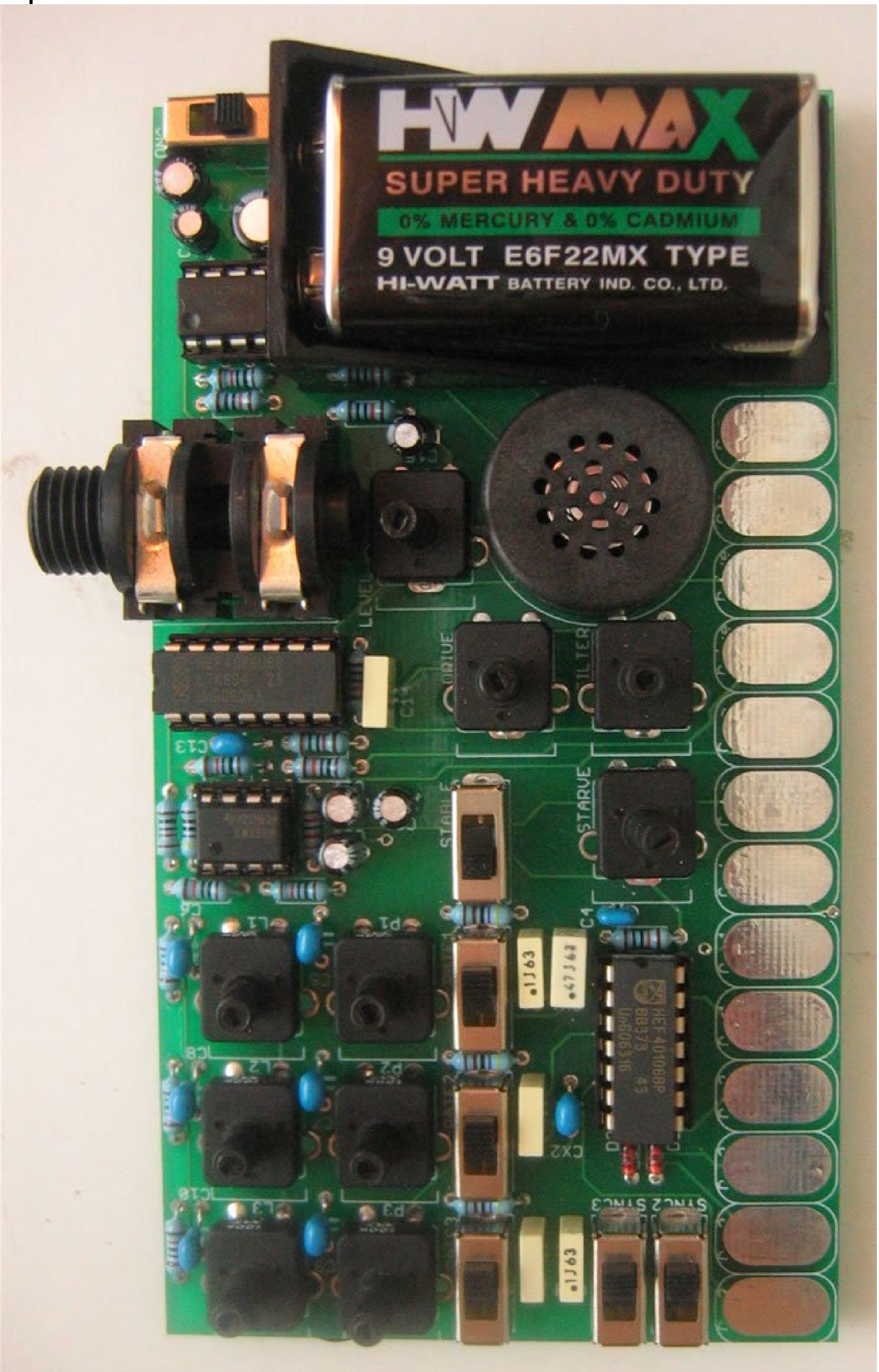
Qty	Value	Device	Parts
2		Diode 1N4148	D1, D2
4	4k7	Resistor	R1, R2, R3, R11
5	10k	Resistor	R9, R10, R12, R16, R21
1	12k	Resistor	R13
9	100k	Resistor	R4, R5, R6, R7, R8, R14, R17, R19, R20
2	8pin	IC Holder	
2	14pin	IC Holder	
1	220p	Dipped Cap	C4
1	22n	Poly Box Cap	C3
2	47n	Poly Box Cap	C2, C14
2	100n	Poly Box Cap	C1, CX3
8	220n	Dipped Cap	C5, C6, C7, C8, C9, C10, C13, CX2
1	470n	Poly Box Cap	CX1
1	1u	Electrolytic Cap	C15
4	10u	Electrolytic Cap	C11, C12, C16, C91
1	47u	Electrolytic Cap	C90
1	100u	Electrolytic Cap	C17
1	10k	Miniature Potentiometer	FILT
8	100k	Miniature Potentiometer	LEVEL, LVL1, LVL2, LVL3, OSC1, OSC2, OSC3, STRV
1	1M	Miniature Potentiometer	Drive
7		Miniature Slide Switch	PWR, RATE1, RATE2, RATE3, STABLE, SYNC2, SYNC3
1		Miniature Speaker	SPK1
1		Mono Jack Socket	OUT
1		Battery	Holder, Nut/Bolt, PP3
1	40106	CMOS IC	IC1
1	LM358	Dual OpAmp IC	IC3
1	4069UB	CMOS IC	IC2
1	LM386	Power Amp IC	IC4
6		Rubber Feet	

Schematic:



Power
Oscs
Mixer
Comparator
Mini-Amp
Distortion
Unused gates

Completed Board:



Parts Placement: