BugBrand WorkshopCrusher
DIY Kit Instructions v.1 October 2014

Welcome!

The WorkshopCrusher DIY kit provides all the parts to build a compact Analogue Sample-rate Reducer device. The kit design and documentation aim to make the process as straight forward and informative as possible, suitable even for complete beginners to electronics.

The first steps in electronics can be seriously confusing and a major aim here is to provide a positive first experience! The kit has been designed with clarity and simplicity in mind, while avoiding cutting corners at the electronic level – you absolutely do not have to understand how things work for now (but plenty of information is given for those who want to learn more).

I believe that soldering is one of the key foundations for DIY electronics, so we will give particular focus to this. Poor soldering is one of the main reasons why DIY projects either do not work or work erratically – few things crush enthusiasm like completing a project and finding it doesn't work.

Sections:

1. The Device & the Kit
2. Tools & Techniques (Soldering)
3. Building Guide
4. Circuit Analysis
5. Bill of Materials (BOM) / Parts Placement / Schematic

Many thanks to Steve @ Thonk (www.thonk.co.uk) for making up the kits so clearly and efficiently! In the unlikely event of missing parts, please contact him directly.

For technical or general questions, there is a build thread for the WorkshopCrusher at the MuffWiggler forum:
http://www.muffwiggler.com/forum/viewtopic.php?t=124057

The latest version of this documentation can be found at:
http://www.bugbrand.co.uk/docs/workshopcrusher.pdf

Good luck & thanks!
Tom Bugs
October 2014
www.bugbrand.co.uk

1 – The Device & The Kit:

BugCrusher Backgroud

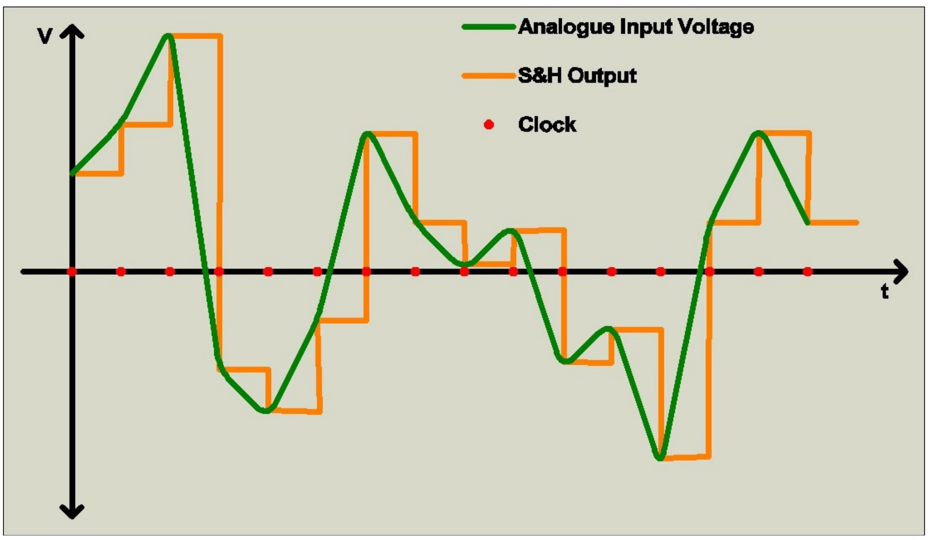
Back around 2006, I came up with the idea of running Sample & Hold at audio rates (though it has surely been done for years) and decided to call the processor the BugCrusher as I imagined it as a bit-crusher type effect. This is something of a confusing title, though – bit-crushers are, typically, a combination of bit-depth reduction (quantizing along the Y-axis – Amplitude) and sample-rate reduction (quantizing in the X-axis – Time).

Bit-depth reduction is a digital process (or overly complex in analogue form) whereby a constantly variable analogue signal is quantized to a limited number of voltage steps, known as the bit-depth. For example, CD audio is 16bit (2^{16} = 65536 possible voltage levels). This conversion process is controlled by a clock (eg. 44.1kHz for CD) – each time a clock occurs, the analogue voltage is sampled and quantized to one of the possible digital values.

This sampling can, however, be achieved in the analogue realm by using a Sample and Hold (S&H) circuit and a variable rate clock. [S&H circuits have long appeared in synths, but are typically scaled to LFO rates]. Initially I used the AD781 S&H Amp – a very high quality device, but with several drawbacks. Not only was it expensive and quite hard to find, it also required Bipolar Power (+ and – power lines – modular synth standard, but rarer in pedals etc.).

The two original schematics I published using this chip are still around online – one is fairly bare bones with a manual clock (not a million miles from the design today) while the other tacks on voltage control for the clock (not very elegant). After a few years I worked out how to make my own S&H circuit using, importantly, simple electronic switch ICs which allowed me to move to using unipolar (single line) power.

Since those beginnings, each time I've developed a new production model I've tried to refine the design. The Workshop Crusher continues the line of the previous MicroCrusher pedal with just a few minor tweaks and updates. I feel I've been able to boil it down nicely to the point today where it is simple but effective and stable.

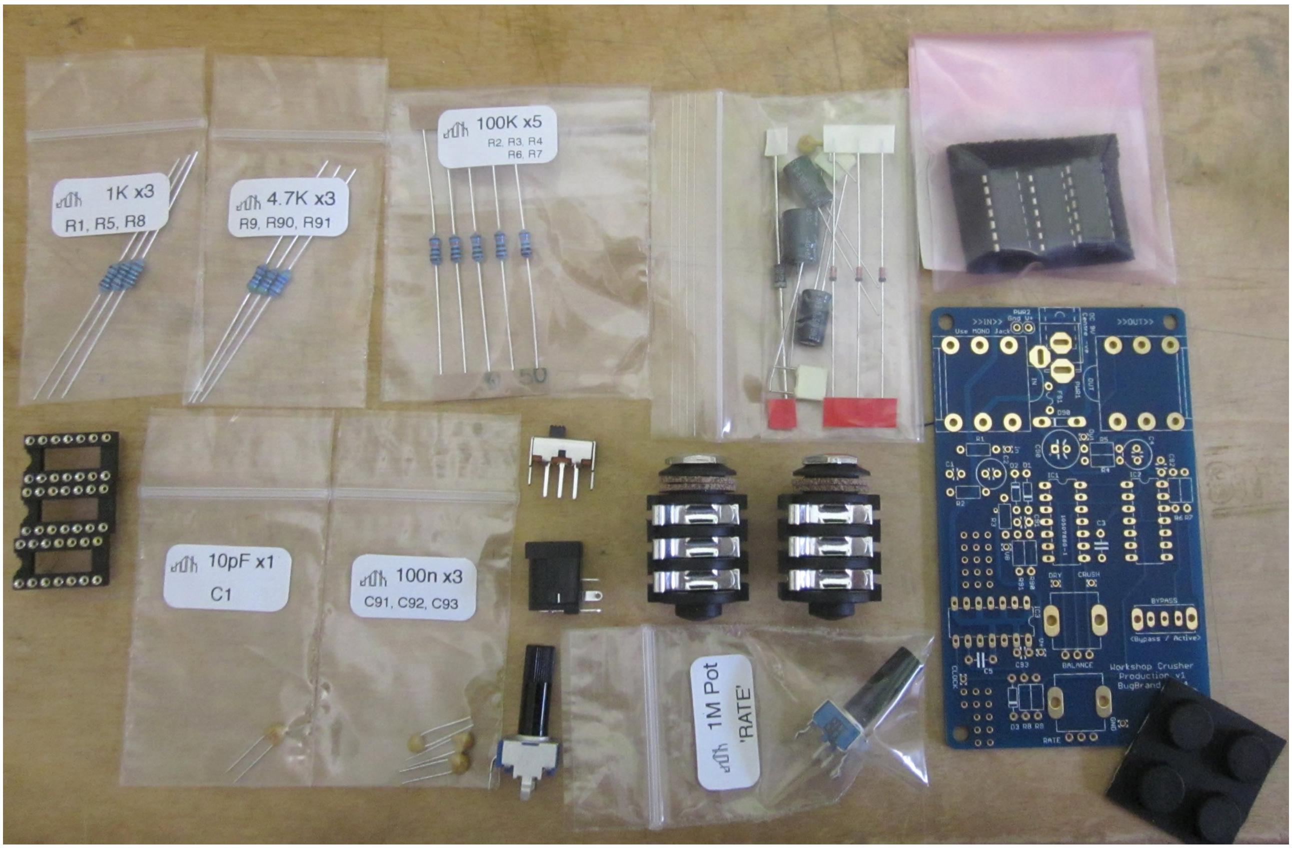


Schematic / Printed Circuit Board / Bill of Materials
The schematic is a diagrammatic representation of the electronic circuit showing what parts, values and power connections are used. While it shows what will make up the circuit, it does not show the actual location of the parts on the Printed Circuit Board (PCB). I see the schematic as a readable diagram and it actually contains all the information you need to populate the PCB.

The PCB mechanically holds and electrically connects the components. It is designed direct from the schematic, with each part having a PCB footprint of pads and silk-screen legend. Copper traces form the connections between parts and can be seen under the blue solder-mask layer. All solder pads are gold-plated (ENIG) to prevent oxidation and are 'plated through hole' (pads on the topside and underside are joined together). The entire kit is lead-free (RoHS compliant).

The Bill of Materials (BOM) provides a convenient list of parts, values and quantities. It can be seen as a checklist for use during the build process.

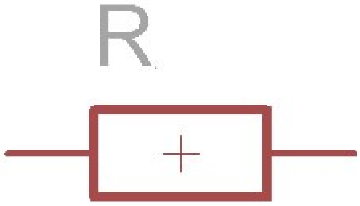
The Kit
Be careful not to lose or mix up parts! Keep parts in their bags until needed.



Thanks to Steve @ Thonk for packaging so clearly – it *really* helps – you can save the joy (headache) of sourcing your own parts for a future project.

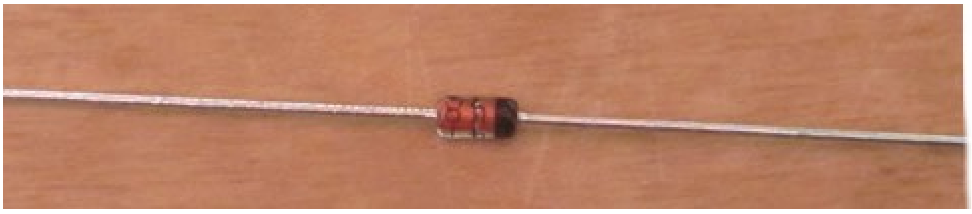
Resistors

These passive components regulate or resist the flow of current and are measured in Ohms / Ω [1KΩ = 1 kilohm = 1000Ω / 1MΩ = 1 Megohm].
The ones in the kit are blue metal film ones, rated at 0.25W and with 1% tolerance (very standard parts).
Coloured bands on the body show the resistor value and tolerance. Frankly, I've never learned the system, finding it far too fiddly, and instead make sure to keep different values carefully separated in bags or parts drawers.
Resistors are not polarized and you'd have to be pretty rough to break one.



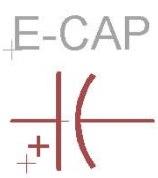
Diodes

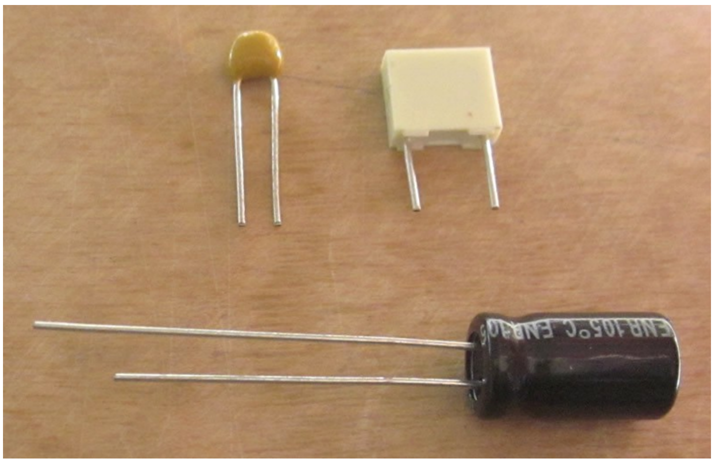
Diodes are like electronic valves that only let current flow in one direction.
Notice the ring marked on one end which shows orientation (cathode). The kit contains three 1N4148 (standard small signal silcon diodes) and one larger 1N4001 (rectifier diode). These are often used in circuits as protection elements (eg. Reverse polarity protection).



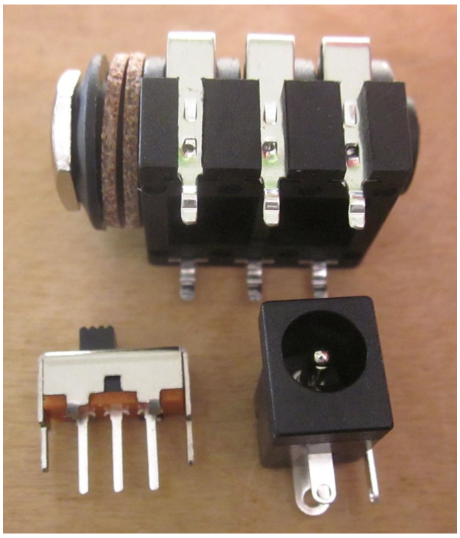
Capacitors

Capacitors store electrical charge and are measured in fractions of Farads / F.
[1uF = 1 microFarad = 0.000001F / 1nF = 1 nanoFarad = 0.001uF /
1pF = 1 picoFarad = 0.001nF]
The kit contains three common varieties:
Ceramic (dipped multilayer) – general purpose, low cost – used for filtering and power decoupling – typical values from 10pF to 1uF
PolyBox (metallised polyester) – more stable than ceramic, used for timing etc. - typical values from 1nF to 1uF
Electrolytic – larger capacitance from 1uF and upwards. Commonly used to condition power supplies and typically polarized.





Mechanical Parts

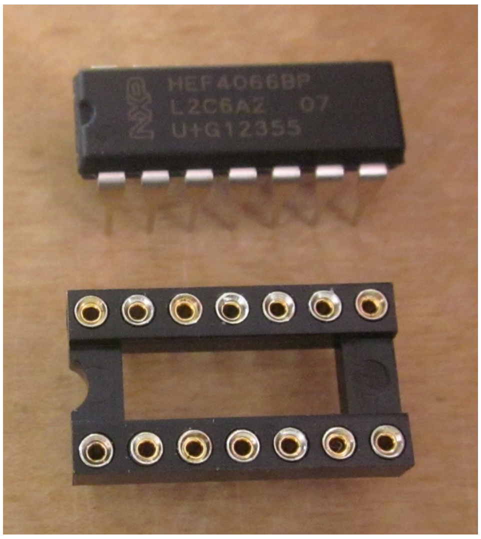


Switches are used to make or break connections. The kit includes a miniature slide switch with three signal pins (+ two mechanical mounting pins for added stability).
The centre signal pin is known as the common – either one of the outer pins is connected to it depending on which way the slider is pushed.

The DC-socket is a standard for power entry, accepting a 2.1mm DC plug.

The 1/4” Jack Sockets allow you to connect to and from the device. We use a stereo version despite the circuit working in mono as this allows us to implement simple power switching.

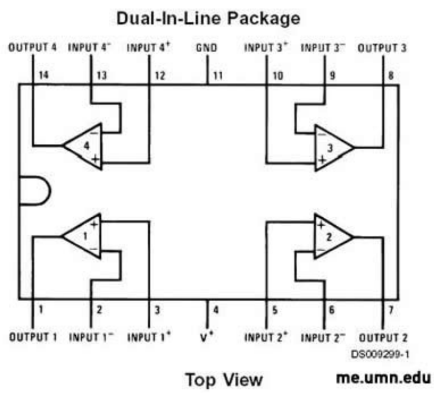
Integrated Circuits



ICs have revolutionised electronics over the last 50 years or so. They contain sets of miniaturised components, printed on to silicon plates and then packaged inside plastic cases with pins to allow access to the internal circuitry.

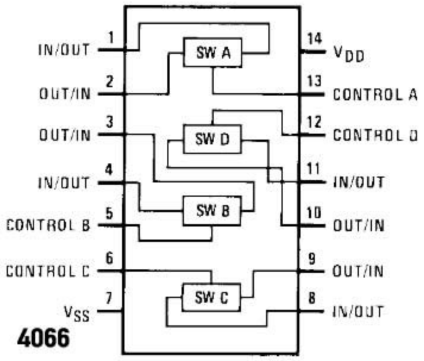
There exist countless varieties of ICs. Devices may be split into main classes such as logic, micro-controllers, voltage regulators, etc, but these can be subdivided further by different functions (eg. fixed, low-dropout or variable regulators) and then further still by variations of actual packaging (DIP, SO, QFP, BGA, etc). And then the 'same' chip may also be offered by several different vendors (eg. Texas Instruments, National Semi, etc). Datasheets should be readily available for all ICs via an internet search.

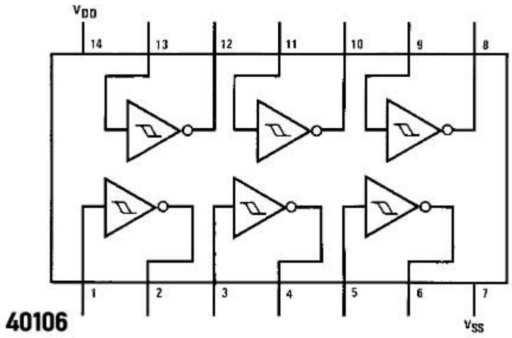
The kit contains one operational amplifier (op-amp) chip and two 4000 series CMOS logic chips, both in 14pin Dual Inline Package (DIP) form. Note we also have IC sockets because ICs are typically sensitive to static discharge and/or heat – a socket allows us to put the IC in after everything else has been built and also allows easy replacement if a repair is required in the future.



The TL074 is a quad low-noise JFet input op-amp – a commonly used part for DIY circuitry. There also exist single (TL071) and dual versions (TL072) along with the TL061/2/4 (similar but low power) and TL081/2/4 (general purpose). Op-amps are widely used to provide circuit functions such as voltage buffering, amplification, mixing, etc.

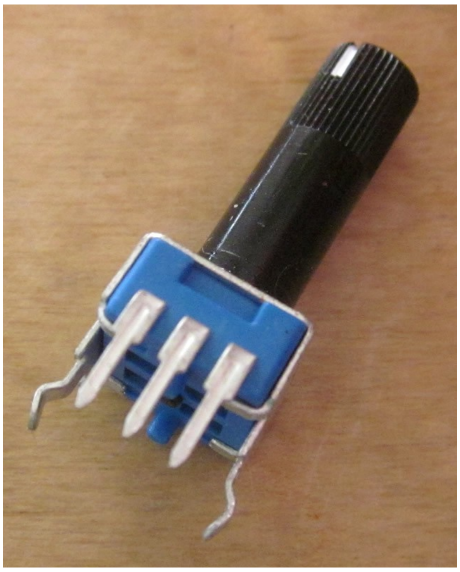
The 4000 series CMOS logic family contains a host of ICs dealing in 1s and 0s – digital functions such as ANDs, ORs, Inverters, Shift-Registers, etc.
The two chips we use actually blur the digital boundaries. The 4066 Quad Bilateral Switch has digital control pins but the switched connections can be analogue. And the 40106 is used in this instance as a squarewave oscillator with continuously variable (analogue) rate.



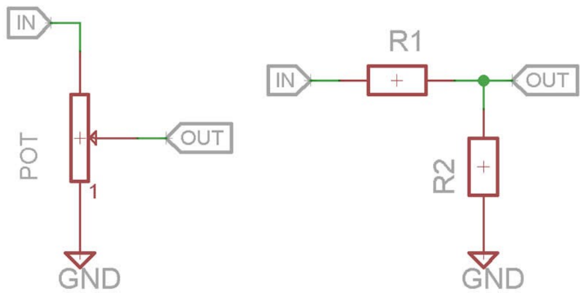


Potentiometer

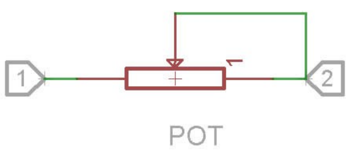
Pots are one of the main 'user control' elements we come across in circuits. They have three signal pins (+ two body pins for mechanical strength) and have a value given in Ohms. There are two main ways to make use of them.



The first way is known as a *potential divider* and crops up in things like amplitude controls. Between the outer pins there is a fixed resistance value (the value of the pot itself). The middle pin (the wiper) moves towards either end, thus dividing that resistance – at one end R1 will be zero, while R2 is maximum, and vice versa. This arrangement is used for the Balance / Blend control in the WorkshopCrusher circuit.



The second way joins two adjacent pins together and forms a *variable resistor.* Now at one end the resistance will be zero, while at the other end it will be the full value of the pot. This arrangement is used for the Clock Rate control.



2 – Tools & Techniques

As everything is mounted direct to the PCB, building is relatively straightforward – the location of parts is clearly labelled and there are no flying wires to install (often a place where errors can creep in). But another bonus is that very few tools are needed. All you really need is a soldering setup and a pair of wire-snips – though a pair of pliers may come in handy too. For the wire-snips (and pliers) you probably want a pair that is relatively small – electronics work is generally quite fine, so you don't want something too big and clunky.

The Soldering Setup

Soldering irons are available from a few pounds up to.. a lot! Interestingly, many of the cheap ones can be used just fine in the short-term (*if* they are kept clean) – they may not last as long (and probably won't be repairable) and definitely lack many features found on better irons, but they can have their place when you're starting out.

The main factors are that they must have a relatively fine tip (eg. 1 mm – *not* one of those monsters used for plumbing) and the tip should be in serviceable condition.

Nicer irons may have features such as:

- silicone cable which won't melt if it comes into contact with the iron
- variable temperature
- a convenient on/off switch
- replaceable tips
- higher power

The tip size is important as you want the maximum contact area to the pad and component – too fine a tip will not be able to transfer heat quickly enough, while a tip that is too large will overhang the pad and can damage the PCB. When the tip is in contact with a pad and component, heat will be transferred from the iron and it will need to be re-energised to keep its temperature constant – an iron with a higher power can do this more effectively.

You will also require some sort sort of stand to hold the iron when not in use and a tip cleaner – either a dampened sponge (an approach I never liked as it drops the tip temperature) or some low-abrasive brass wool.

Think also about your workspace – clear a good area, employ decent lighting (a high power daylight bulb is ideal), think about ventilation and how you position yourself (both in terms of being able to clearly see what you are doing and so that you aren't going to bugger your back).

Solder
Solder is an fusible metal alloy used to join together metal pieces, such as components and PCB pads. While solder used to typically be made of 60% tin and 40% lead, these days most of the electronic industries worldwide have moved over to lead-free processes (PCBs, components, solder, etc.), due in part to the RoHS initiatives in the EU.

All parts in this kit are lead-free and while leaded solder could be used (and can still be legally sold, especially for repairs etc.), I feel that progress is moving very much towards lead-free. Some people maintain that leaded solder is easier to use (due to the lower melting temperature), but this should not be an issue with decent soldering technique and reasonable tools.

While we're on the subject, although lead is certainly poisonous and you should wash your hands after using leaded solder, it should be understood that breathing in the fumes from soldering will not give you lead poisoning as lead would only vapourise at much higher temperatures.

The fumes from soldering are actually flux (a chemical cleaning agent in the core of the solder wire) burning off and it should be noted that fumes from rosin flux have been linked to occupational asthma. When doing a lot of soldering you may want to open a window, invest in fume extraction and/or find a solder that uses a rosin-free flux.

Cleaning the Iron
It cannot be overstated how important it is to learn how to keep the tip of the iron clean – without a clean tip, not only will the job of soldering be difficult, leading to poor joints or damaged components, but the life of the tip may also be degraded. The job of the soldering iron is to transfer heat to the join of the PCB pad and the component – if the tip is not in good condition then heat will not transfer effectively.

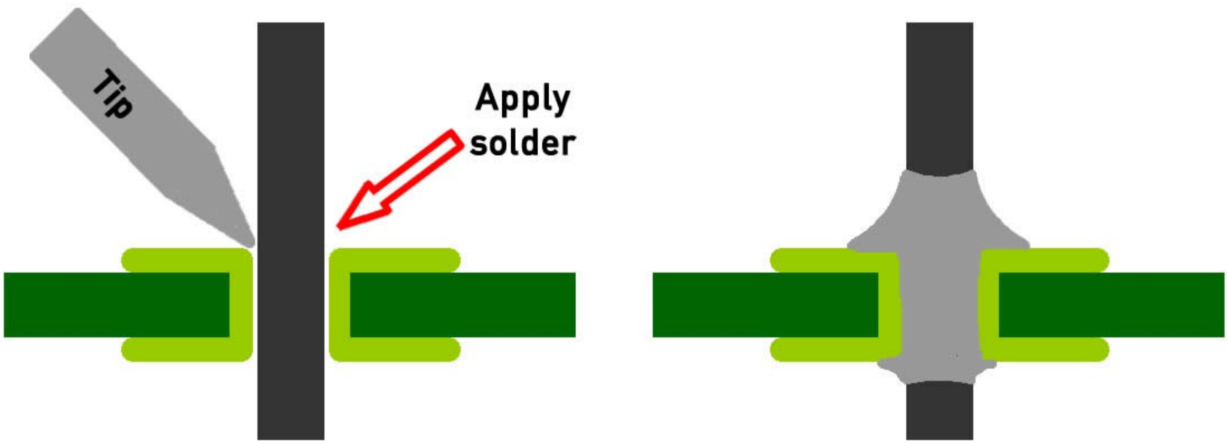
Oxidation occurs when the heated tip is in contact with the air (ie. Whenever the iron is on) and appears as a surface dullness covering the tip. Solder will not adhere to an oxidized tip (known as de-wetting) and the layer of oxidation acts as a barrier to heat transfer. Cleaning is achieved by first wiping the tip in some tip-cleaning brass wool (or on a damped sponge as mentioned above) and then re-coating the tip with fresh solder, a process known as *tinning the tip*. We mentioned flux a moment ago – the flux core of the solder wire actually helps remove any oxidation which is one of the reasons tinning is so important. A correctly tinned tip should have a thin, shiny coating of solder.

Cleaning / tinning should be done when you first pick up the iron to make a solder joint, after every few solder joints (as required) and, importantly, *before* putting the iron down again. This fresh coat of solder when the iron is on but inactive protects the tip – a tip without a coating of solder will oxidise quickly. Note that if you will not be soldering again for a few minutes then you should turn down or turn off the iron (hence why temperature control or an easy on/off switch is useful).

If excess oxidation has occurred it may be overly difficult to clean the tip using this method (though you can always repeat the steps two or more times if required). In such instances you must use either a specially formulated tip-tinner compound (a mix of flux and solder) or a low-abrasive tip-cleaning bar – hard abrasives such as sand-paper or files should never be used as they will severely damage the tip. Note these reconditioning techniques should be last resorts and only used occasionally as they do wear out the tip more quickly than regular cleaning.

Soldering
Once the importance of cleaning has been understood, you are ready to move on to soldering.

- First the iron must be switched on and allowed to heat up fully – this may take a couple of minutes with low cost irons.
- When the iron is fully heated, clean and tin the tip.
- Place the tip of the iron to the join so that contacts both the pad and the component.
- For regular components, allow around 1 second of heating before applying solder. Larger pads and components will require slightly longer times to properly heat (or a higher iron temperature)
- Solder should be applied to the *opposite* side of the joint. As it begins to melt it will flow towards the heat and should flow neatly right the way around the joint. Enough solder should be added to completely cover the pad, but should be added gradually as too much too quickly may not flow correctly, forming a poor joint.
- Once the solder has flowed, remove the solder wire, then the tip of the iron, taking around 2 to 3 seconds for the entire sequence of heating and soldering. A correctly soldered joint should appear slightly concave and should be relatively shiny (more so for lead solder).
- Further joints can be soldered now if required, before cleaning the iron and replacing it in the stand. [Remember to never leave the iron inactive without a fresh layer of solder]



The process certainly takes some practice, coupled with background knowledge. Here are some extra pointers:

- The component / joint must not move during soldering or else the joint will be poor quality. Fix the component in place by, for example, splaying the legs (as described in the building guide) and keep the PCB stable by placing it securely on your work surface, perhaps using a finger to stabilise it (or some people like to use a PCB vice).
- As mentioned, you should apply solder to the side opposite to the tip, but you can start the process off by momentarily touching the solder to the tip then instantly moving it round to the opposite side. Do not try to solder by applying the solder direct to the tip.
- You should never apply excessive pressure with the tip as it can cause damage (to the tip or PCB) and does nothing to aid the transfer of heat.
- Think about how you position your hands and how they can add stability. For right-handed people, you may use your left hand to hold the solder while also using a finger to stabilise the PCB, while the right-hand holds the soldering iron. Resting your wrists on the work surface can greatly increase how steady your hands are.
- Do turn the PCB around to find the best 'angle of attack' for soldering, especially if there are many leads sticking up to get in the way.
- Decent lighting is essential to see that your soldering is effective.

Bad Solder Joints

You should quickly learn how a decent solder joint looks – shiny, slightly concave and fully covering the pad as mentioned.

If the heat transfer is ineffective due to tip oxidation or poor contact, you can get what is called a *cold solder joint* where the solder forms a blob on the tip, pad or component lead, thus not forming a correct joint. While a partial joint may be made, this can quickly become intermittent.

If too little solder is applied then, again, the joint may become intermittent. Too much solder can either hide problems or could potentially bridge across to other pads, causing a short circuit.

Luckily, each of these errors can easily be corrected by remelting the solder and adding a little more fresh solder if required. A good final step in building is to examine all joints and retouch any which don't look quite right.

De-soldering

If a joint has to be completely redone, for instance if a component has been inserted in the wrong position, then de-soldering will be necessary. With plated through hole PCBs this tends to be tricky, so it is best to proceed slowly during building to avoid mistakes. If unavoidable then either a vacuum pump or braid can be used.

With the pump, the plunger is pressed down before melting the offending solder – once molten, pressing a button releases the plunger and the solder is sucked up by a vacuum action. This isn't always effective, however, on plated through hole PCBs. Special heated vacuum pumps can also be purchased which may stand a better chance.

Desoldering braid acts to remove solder by capillary action – the solder is melted, the braid is dragged through it and the solder wicks up into the braid. There is quite a knack to this though.

In both cases you must be careful to avoid over-heating the PCB during the process. In extreme cases it may be necessary to sacrifice the offending component entirely – cut the legs off, removing the main component body, then remove the legs with tweezers while applying heat to melt the solder. Once the legs have been removed it will be easier to remove any solder remaining.

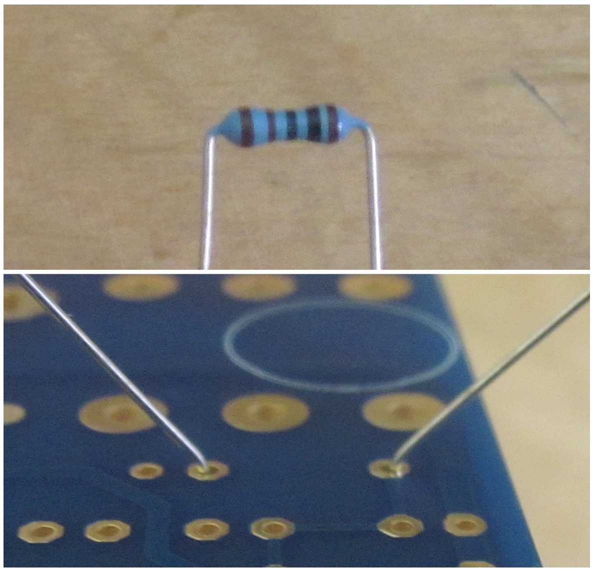
3 – Building Guide:

Keep the BOM and schematic on hand through this section. The kit is built up in the order shown on the BOM – we build *up* with low profile components first as this helps keep the PCB level during soldering. All components go on the top side of the PCB (the side with the white silk-screen legend).

You are advised to keep the parts in their bags until needed to minimise the risk of losing anything. Remember that it is not a race! You should check & re-check before soldering as it is hard to correct things once soldered (see above). For some parts, especially large or multi-pin ones, it can be a good idea to solder just one lead first, then double check everything is correctly in position / flush before soldering the remaining pins. Take extra care with any polarized parts which must be inserted in a particular orientation.

Resistors

Begin with the 1k resistors (bag of 3). Bend the leads to right-angles just beyond the body of the resistor. Identify where each resistor is to go (they are not polarized) and insert the legs – the part should slide neatly into place to sit flush against the PCB. Hold the resistor in place with a finger and slightly splay the legs on the other side – this helps keep the resistor firmly in place before soldering. Insert all 3 resistors before turning the board over, placing it on your desk and soldering the 6 legs. Once soldered, the leads can be neatly cut with wire-snips just above the solder joint.



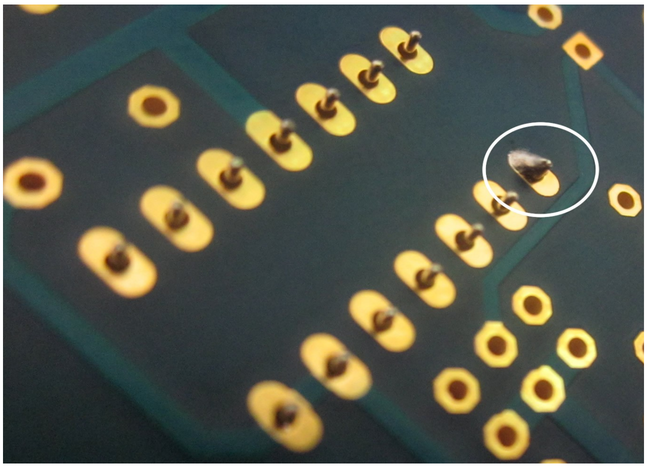
Repeat with the 4k7 and 100k resistors.

Diodes

These should be approached in the same way as the resistors, but noting the orientation – each diode has a band marked on one end (black for 1N4148, silver for 1N4001) which corresponds to the white stripe marking on the PCB. Note that D1 and D2 go in opposite directions.

IC Sockets
Do NOT use the ICs themselves in this stage – they are inserted into the holders in the final step after everything else has been completed. Note the notch at one end of the socket which corresponds to the notch marking on the PCB legend.

As these are 'turned-pin' sockets, they will not stay in place when the PCB is turned upside down, so you need to tack-solder one pin of each socket first. Hold the IC socket flush to the board with one finger and turn over the board, melt a small blob of solder onto the iron, then quickly dab it on to one pin and pad to hold the socket in place. This is, of course, bad soldering technique, but it is a useful temporary measure – once the other pins have been soldered you can correct the soldering on the 'tack' pins. Double check that the ICs are correctly aligned and are fully flush before soldering – mistakes on ICs / sockets are very hard to remedy due to the number of pins.



You can snip the pin ends after soldering, but it is not absolutely necessary.

Capacitors & Fuse
The dipped ceramic [mustardy yellow colour] and poly-box capacitors [pale yellow] are not polarised. You may want to splay their legs slightly after insertion to hold them in place.
If you get them mixed up:
Ceramic – the 10p is marked 100 [10 and no 0s], while the 3 x 100n are marked 104 [10 and four 0s – 100000]
Poly-Box – the 1n is marked 1nK100 while 4n7 is marked 4n7K100
[the K means 10% tolerance, the 100 means DC rating 100V]

Next insert the Fuse [mustard yellow, 5mm lead spacing].

For the electrolytic capacitors, the two smaller 10u ones are non-polarized and can be inserted either way around, while the 220u one *is* polarized. The PCB footprint has a square pad that also has a small '+' sign marked for the positive side of the capacitor. On the capacitor itself, one lead is longer than the other – the longer lead goes to the square hole. [Alternatively, note that one side of the capacitor has a white stripe with '-' signs – this side corresponds with the round pad]

Switch, Sockets & Potentiometers

The switch should stay in place once fully inserted. Note that the outer pins are simply mechanical strengtheners – the signal connections are the three smaller central pins. The switch is not polarized.

Tack-soldering is again needed for the DC socket. It has a small amount of 'play' so ensure it is neatly aligned with the PCB edge when you hold it in place.

The 1/4” Jack sockets should click in and hold themselves in position, but do ensure they are fully flush with all pins showing correctly before soldering.

The pots should also click into position for soldering – solder one signal pin (the three small, round pads), check that the pot is absolutely vertical, then solder the remaining pins (including the larger body supports).
The pot value is marked underneath with 103 (10,000) corresponding to 10K and 105 corresponding to 1M.

Final Checks, Rubber Feet

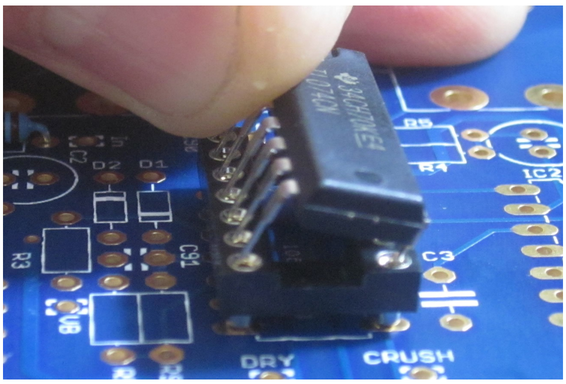
It is now a good time to re-check all solder joints by visually examining the underside of the PCB, retouching any joints that don't look right. Also trim any excess leads left from previous stages.

Once you are satisfied, add the four rubber feet to the underside of the PCB in the marked positions.

ICs

Extra care must be taken when inserting the ICs. Firstly, some of the ICs may be sensitive to static discharge so you must dissipate any charge on you by touching a grounded metal object (eg. An unpainted part of a central-heating radiator.

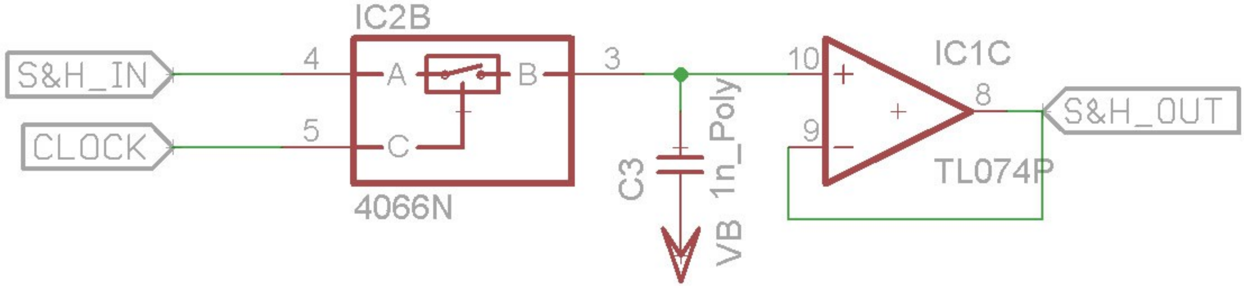
Identify which IC is which – all three ICs are 14Pin DIL, but have different code markings (confusingly, there are several other codes on each chip to signify things like date of production – these can be ignored).
Note the notch on one end of the IC which corresponds with the notch on the IC socket (and the PCB legend).



There is quite a knack to inserting ICs – their legs come slightly splayed as standard and careless insertion can lead to pins becoming bent (which makes it harder then to correctly insert the IC – or a pin can bend under the body of the chip). My preferred approach is to part-insert one side of pins, holding the IC at a slight angle. Apply gentle pressure to uniformly bend the pins until you can insert the other side of pins.

4 - Circuit Analysis:

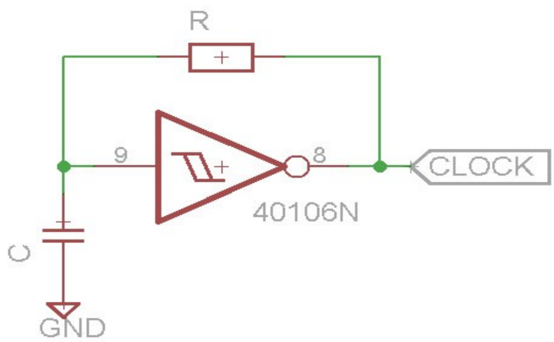
Sample & Hold Section



The S&H core is made up of an electronic switch (one section of the 4066 quad switch), a storage capacitor and an op-amp buffer. When the clock input goes high, the switch is closed, the input signal is connected to the storage capacitor and the voltage output tracks the input. As soon as the clock goes low, the instantaneous voltage on the storage capacitor is held (sampled) with the op-amp buffer stage keeping this voltage constant until the next clock occurs.

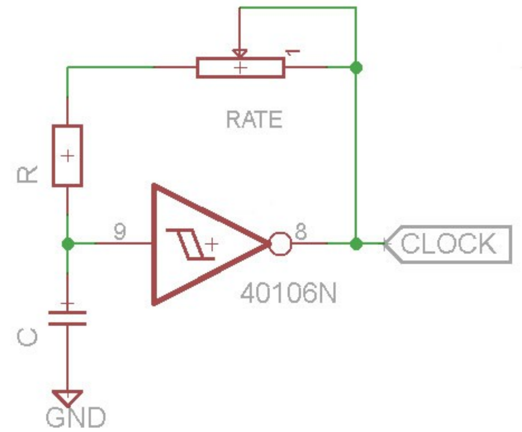
Clock Circuit

The clock is built from one section of the 40106 Hex Schmitt Inverter (the other sections are unused & their inputs tied to ground). The 40106 chip can be used to make pretty much the simplest electronic oscillator – you just need a resistor and a capacitor (and a power supply).



This circuit will produce a 50/50 squarewave output (from the output point marked *CLOCK*) with the rate determined by the values of Resistor R and Capacitor C.

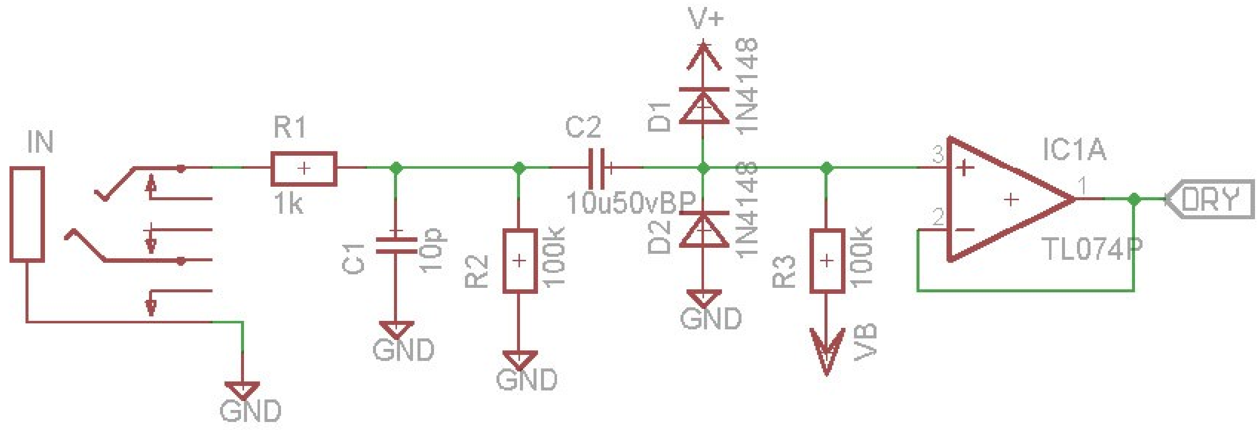
You can easily make the rate variable by replacing the fixed resistor with a variable one (+ small fixed value resistor to prevent short circuit).



[You could also change the range of the oscillator by changing the capacitor value – you can easily add a switch to add a larger value capacitor in parallel with the original value (which would slow down oscillations and lower the overall range)]

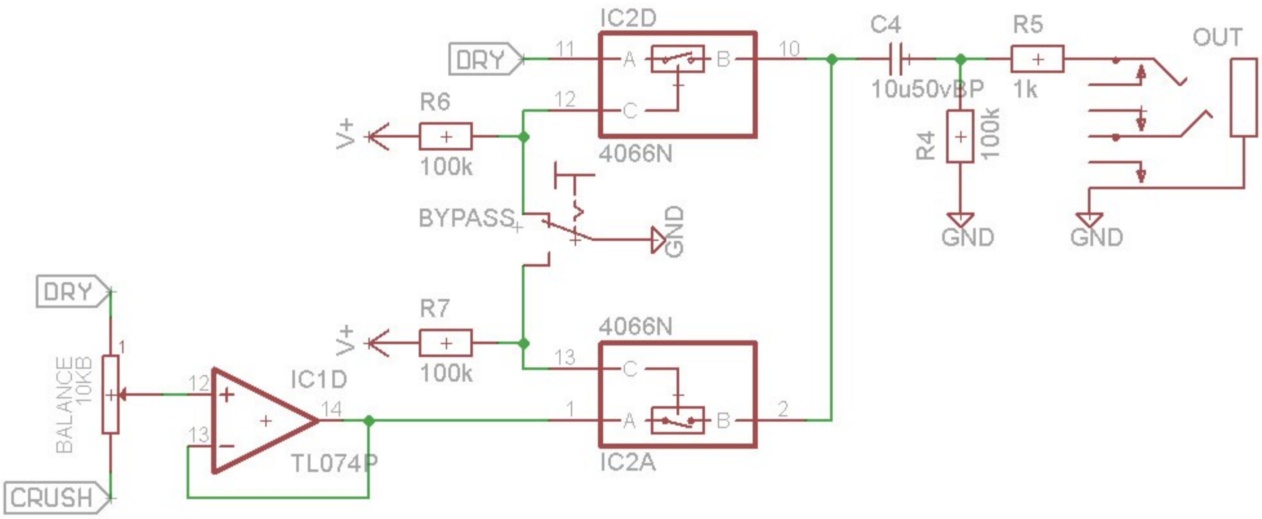
It is important that the clock pulse is as short as possible, this is achieved by feeding the output back to the input via a diode. With this in place, as soon as the output goes high, the feedback instantly causes a reset and the pulse duration is minimised.

Input Section



The input section is a standard, unbalanced, unity-gain input buffer that aims to condition the signal for the rest of the circuitry. Resistor R1 and Capacitor C1 form a low-pass filter to remove any incoming RF signals – these are placed as close to the input socket as possible. C2 blocks any DC voltage with R2 providing a path to drain any charge that does build up, thus avoiding thuds when making connections. R3 biases the input to the VB midpoint (described below) and diodes D1 and D2 protect in case of any voltage swings beyond the power rails. The op-amp section is set up as a standard non-inverting unity gain follower – note that each stage in the overall design keeps both the gain and the phase constant.

Balance, Switching & Output

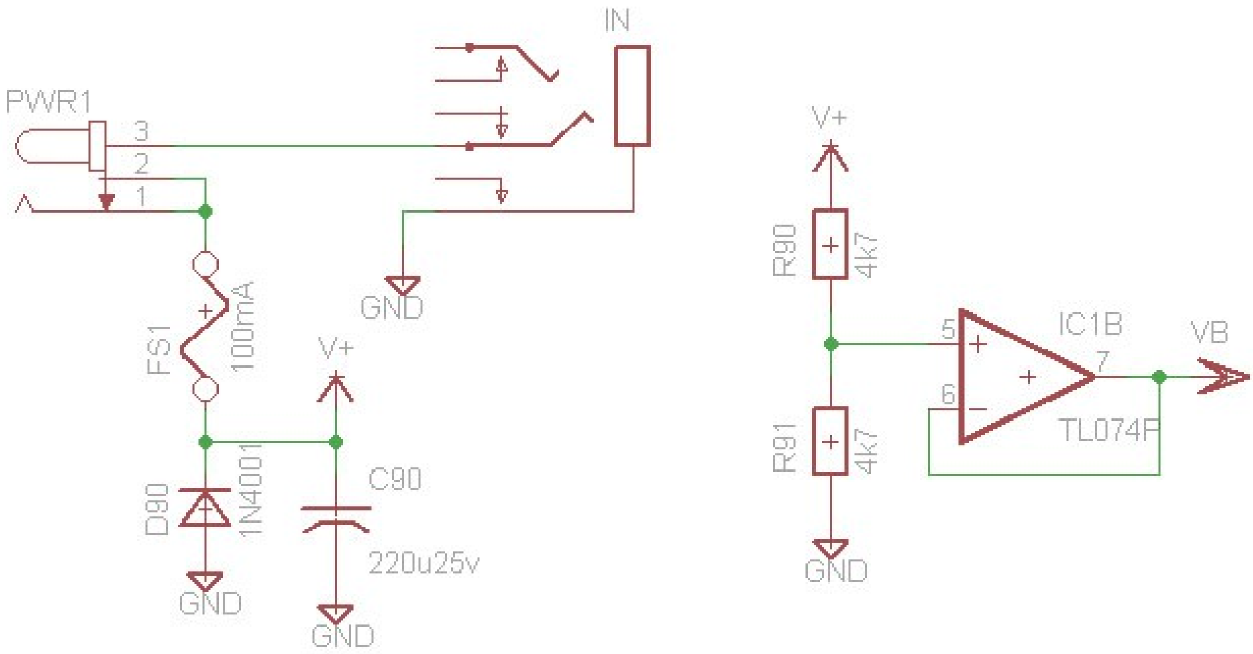


The blend control is made with a pot and a buffer amp. On one end is the dry signal, on the other is the S&H output. The op-amp buffers the voltage at the wiper.

The bypass switching makes use of two of the remaining switch sections in the 4066 IC (the remaining section is unused with the control input tied to ground). The switching is arranged so that while one switch is closed, the other is open and vice versa, and so selects either the (buffered) Dry input signal or the signal from the Balance stage. Notice that each switch control port is pulled high by the 100k resistors (thus closing the switch) while the Bypass switch takes one or the other side low (opening the switches).

C4 and R4 at the output form a passive RC high-pass filter to block the internal DC bias voltage. R5 acts as a basic circuit protection by current limiting in the case of a short circuit.

Power



The V+ and 0V power lines come in via a 2.1mm DC power socket with centre-negative polarity (Boss style). This was designed for typical 9V operation, but the main reservoir cap, C90, is spec'd to 25V so the circuit should be fine running direct off +12V or +15V (unipolar – no negative rail).

A power switching trick (quite standard in guitar pedals) is implemented with the input socket. The 0V input route is only connected when a (mono) jack is inserted – mono jacks have a single connection joining the 0V / sleeve at the ring position, thus making the required connection.

The power has a resettable poly-fuse and power diode to protect from reverse polarity – if power is connected backwards, an excessive current will flow through the diode, heating up the polyfuse and making it go open circuit, thus protecting the main circuit.

The VB (the positive rail divided by 2) is generated by two equal value resistors buffered by a spare op-amp section. This provides the 'central reference' for op-amp stages (R3 bias and C3 storage cap connections).

Note that each IC has a standard 100n decoupling capacitor in-line across the power close to each V+ input. The 0V and V+ lines for the 40106 clock section are fed (from the main reservoir cap) separately from the rest of the circuit.

Further Ideas

A few extras have been added to try to help with DIY expansion / experiment:

- Two small prototype pin areas have been added
- 3.5mm mounting holes at each corner
- Test points [Dry, Crush, Clock, etc]
- Extra power header position [bypasses jack socket switching]

During prototyping I experimented with an idea from Synthmonger on Electro-Music.com (http://tinyurl.com/kuvm559)which aimed to add Voltage Control (VC) to the clock in a simple manner (a transistor & a couple of resistors). I tried it out and it certainly worked, but I couldn't manage to tame it to a manageable range for my liking (without adding extra circuitry/complexity). [Note - To experiment along these lines you would need to change the connections to the Rate dial which would involve cutting traces on the underside of the PCB.]

The circuit could be used in a modular system directly by powering just from the V+ and 0V power lines with the input and output circuit capacitors blocking any DC input signals.

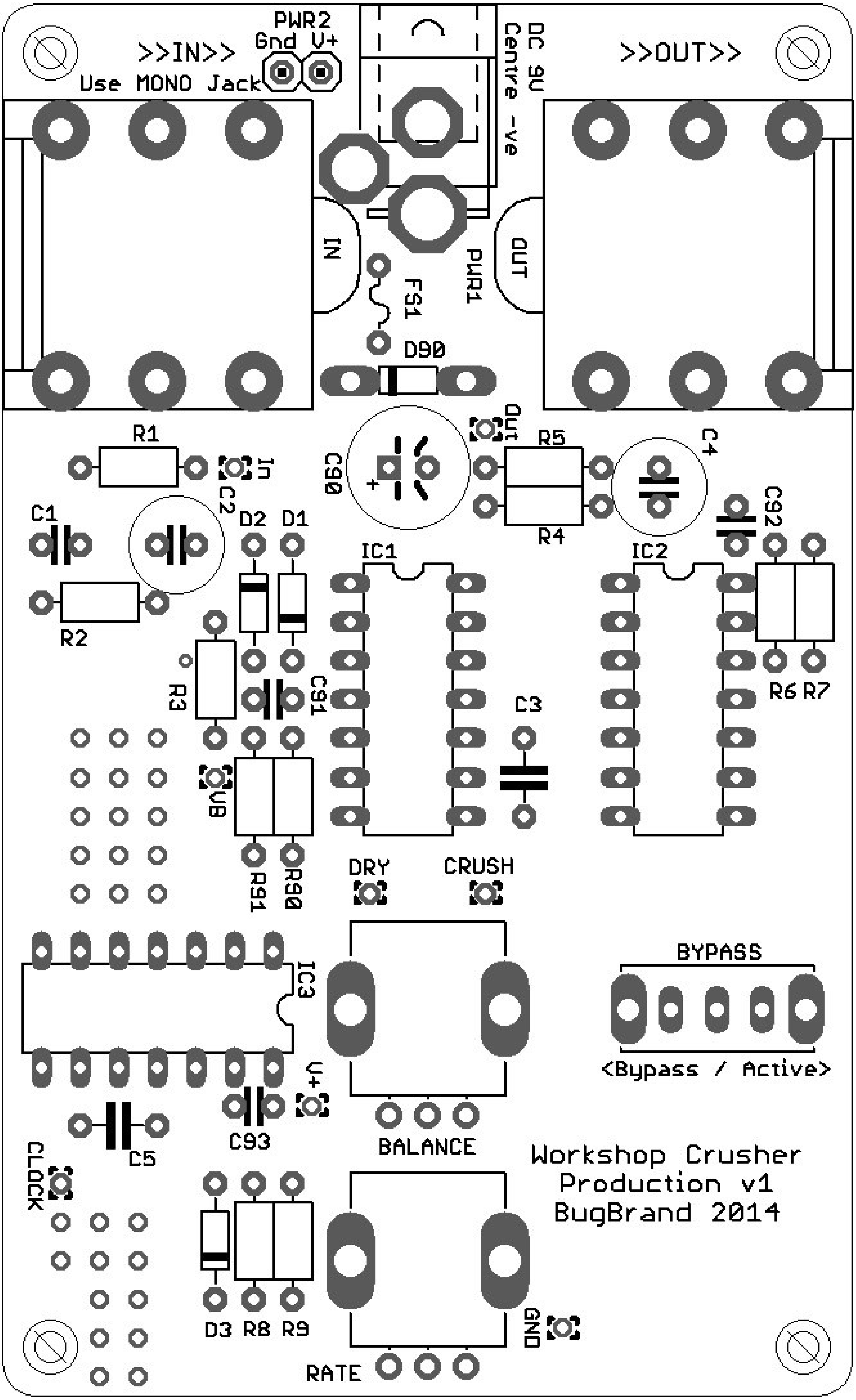
If I were re-designing this more for a modular system, the circuitry would be changed to use bipolar power and I would upgrade to an electronic switch such as the DG412 (the 4066 can only take maximum 15V power).
The approaches of the S&H section could be used at sub-audio / LFO rates (think typical 'random generator' S&H effects) by changing the clock timing capacitor (the storage cap value should be increased also).

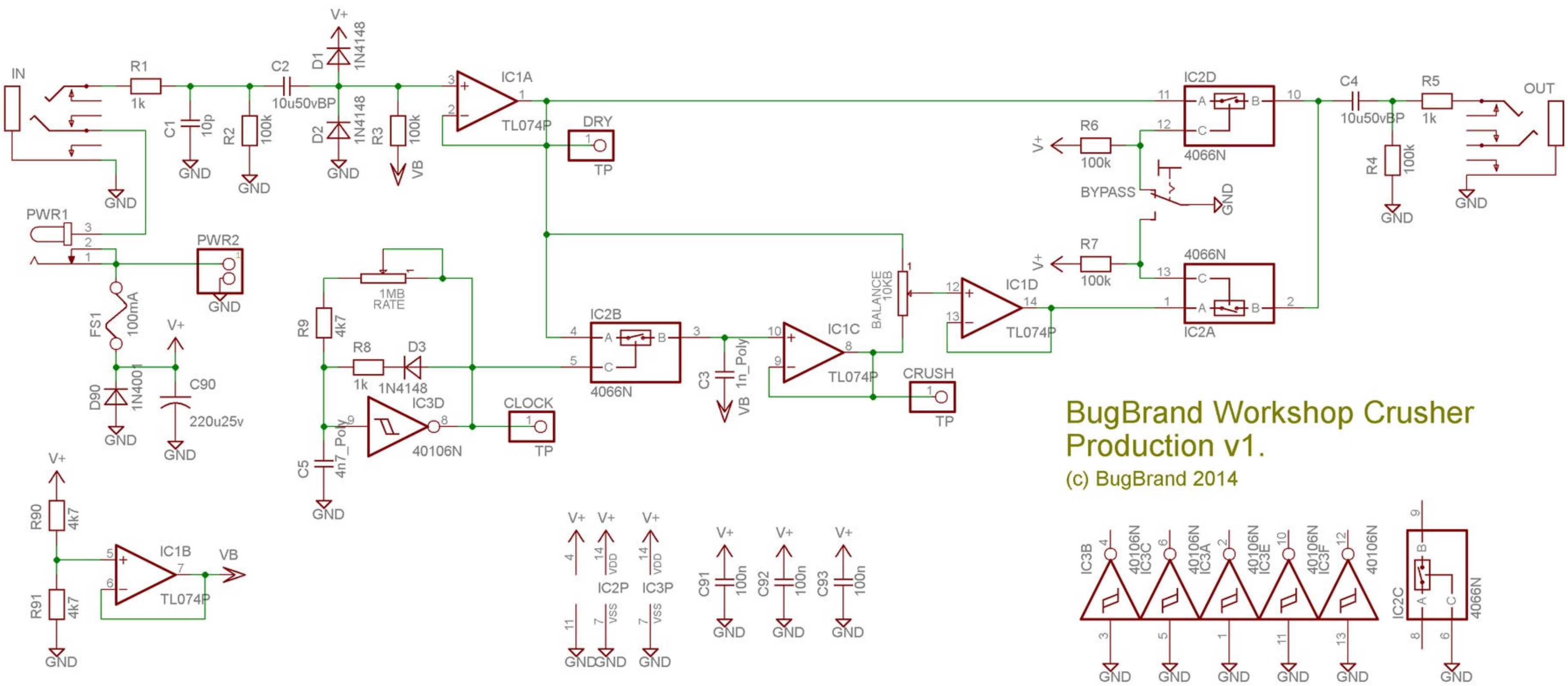
Well done! Hopefully you now have a fully working WorkshopCrusher and have learned some useful techniques. If you have encountered problems and need help, or if you have questions and ideas, then do visit the Muffwiggler build thread mentioned back at the start of this document.

Thanks must go to Richard Scott for the initial asking which got the balls rolling, for Sam Weaver and Ricardo Climent for organising the festival where this project was launched, and to Steve @ Thonk for making and selling the kits.

5 - Bill of Materials (BOM) / Parts Placement / Schematic

Qty	Value	Device	Package	Parts
3	1k	RESISTOR7,5PCB	R-7,5	R1, R5, R8
3	4k7	RESISTOR7,5PCB	R-7,5	R9, R90, R91
5	100k	RESISTOR7,5PCB	R-7,5	R2, R3, R4, R6, R7
3	1N4148	DIODED-7,5	D-7,5	D1, D2, D3
1	1N4001	DIODED-PWR	D-PWR	D90
3		IC Socket	DIL14	IC1, IC2, IC3
1	10p	Dipped Ceramic 2.5mm	C-NP-2,5_SCN	C1
3	100n	Dipped Ceramic 2.5mm	C-NP-2,5_SCN	C91, C92, C93
1	1n_Poly	Metallised Polyester Box 5mm	C-NP-5_SCN	C3
1	4n7_Poly	Metallised Polyester Box 5mm	C-NP-5_SCN	C5
1	0.1A	Resettable Fuse 5mm	RST_FUSE	FS1
2	10u 50V BP	Non-Polarised Electrolytic Cap	C-NP2,5_SZ125	C2, C4
1	220u 25V	Electrolytic Cap	C-2,5_SCR16	C90
1		Miniature Slide Switch	SDPT	BYPASS
1		DC Power Socket 2.1mm	SPC4077B	PWR1
2		Rean NYS215 1/4” Stereo Skt	JACK_SKT_1	IN, OUT
1	10KB	9mm Miniature Potentiometer	ALPS9MM	BALANCE
1	1MB	9mm Miniature Potentiometer	ALPS9MM	RATE
1	TL074	TL074	DIL14	IC1
1	4066	NXP HEF4066B	DIL14	IC2
1	40106	NXP HEF40106B	DIL14	IC3
4		Rubber Feet 11.5x3mm		


>>IN>>
Use MONO Jack
PWR2
Gnd V+
DC 9V
Centre -ve
>>OUT>>
IN
PWR1
OUT
FS1
D90
R1
In
C90
Out
R5
C4
C1
C2
D2
D1
R4
C92
IC1
IC2
R2
R3
C91
R6 R7
C3
VB
R91
R90
DRY
CRUSH
IC3
BYPASS
<Bypass / Active>
V+
C5
C93
BALANCE
CLOCK
Workshop Crusher
Production v1
BugBrand 2014
D3 R8 R9
GND
RATE



BugBrand Workshop Crusher
Production v1.