HOT IRON

"Journal of the Constructors Club" ©

Upton Bridge Farm, Long Sutton, Langport, Somerset TA10 9NJ

Issue 2
December
1993

I am very pleased to be able to tell you that we now have about **50 members** of the **Constructors Club**. This is an excellent start - much better than I had hoped for and I am particularly pleased to have a small but significant number of overseas members. Please spread the word to your friends interested in electronics and amateur radio. I emphasize again that I hope many of you will contribute to the journal - even if its only a small item or idea almost certainly others will be pleased to see it - so send in your contributions please!

For this issue, the construction theme is slightly stronger than last time with very welcome contributions from a trio whose arms were more pliable than others! Thank you Gentlemen. I have included G3GC's note on soldering because good soldering is fundamental to all electronic construction and it is often the cause of problems. My own note on "Dead-bug" construction is intended to encourage experimentation without the need for etched printed circuits - the technique is very effective from an rf viewpoint and lends itself to very easy changes. Alternatively you may prefer the smart effect achieved by G4GVM, but it does require you to have sorted out the circuit and sizes - you can still change values provided the footprint on the PCB does not change. Interestingly G0HDJ has contributed a note on filters for direct conversion rigs, a topic which has occupied my spare moments quite a bit recently in connection with the new rig I am working on; I have in mind to include a tuneable switched capacitor filter with a very steep slope comparable to that achieved in the best IF filters in superhets. I need some advice on the type of filter, which is acceptable for CW without undue ringing effects (Bessel, Buterworth, Chebychev or Elliptic) - so if anyone can produce a reference I should be pleased to have it. At one stage I thought I might have to include a fancy filter in the Yeovil to get rid of interference from my latest frequency counter kit - however a fresh design using more modern chips and a change to CMOS logic has completed eliminated interference in the RX. This display kit is for direct conversion rigs or superhets or as test gear; there are two input channels which can be added or subtracted and will work to 60 MHz minimum - the prototype did 85 MHz! It suits all of my rig designs. I also now have a converter kit for the Yeovil so that it can be made into a three band transceiver covering 20, 40 and 80 metres.

Finally, following an inquiry about where to obtain parts from a member; if you do have difficulties, I can generally obtain any of the parts that get a mention in **Hot Iron.** Don't forget that membership of the **Construction Club** allows you to obtain kits in stages if this suits Father Christmas' pocket! For administrative convenience, I have decided that membership of the **Club** will always start on Sept. 1st of any year so that I have only to worry about subscriptions once a year! **Hot Iron** is 50 % bigger this time, I will try to keep it up but it needs your contributions. Tim Walford - Ed.

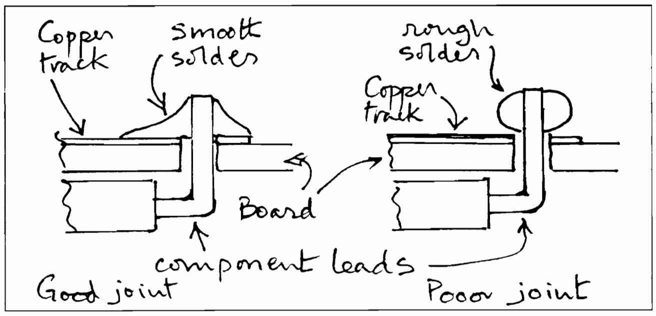
Soldering Techniques

Once one has learnt to solder, then it is one of the most simple and sure ways of making a good electrical joint and may, depending upon the particular circumstances, also be a good mechanical joint. The most important thing to remember about soldering is that "cleanliness is next to Godliness". This is closely followed by having the correct size iron at the right temperature, the correct solder, tinning and keeping the work absolutely still until the joint has set.

To obtain a good joint the work must be absolutely clean. Most components will have silver plated leads which (if new) take solder easily but enamelled wire can be difficult. The insulation of modern "enamelled" wire, which has a slightly pinkish colour, will " burn off" purely with the heat of a largish iron but do NOT breath the fumes. Old style enamelled wire needs scraping with a penknife or similar, followed by cleaning with a scrap of wire wool done well away from the equipment - do not use sand paper or emery cloth. After cleaning the wire down to the bare and bright copper it should be tinned as explained below. The copper of printed circuit boards does become tarnished in time unless it has been lacquered or roller tinned all over, again light wire wool rubbing will clean it up followed by thorough brushing. If the tracks are narrow and might peel off, rub along them. Where a board has been lacquered it is not necessary to remove the lacquer prior to soldering; it is also quite possible to solder direct to PCBs covered in freshly exposed photo resist (a greenish colour) but after a while such boards will tarnish and will need wire wool cleaning. Old components may also need cleaning and tinning prior to assembly. (See tip below.)

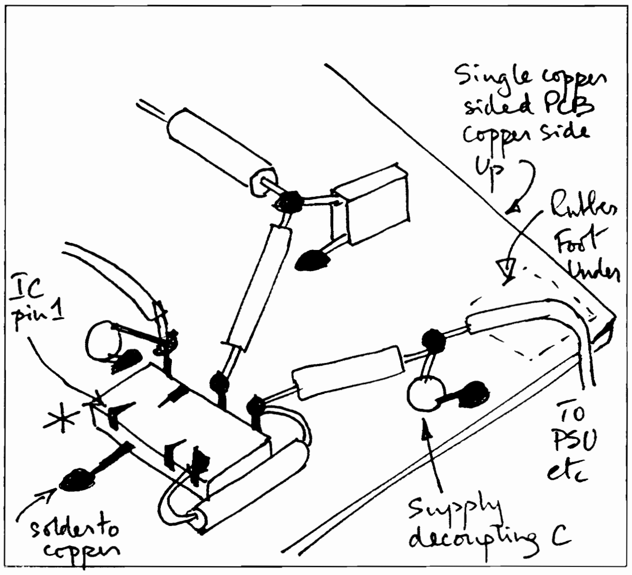
It is essential to have the right sized iron for the job and most irons will have a power to match their bulk and achieve the right temperature. For soldering components to PCBs, a small 2 mm pencil bit with a 18 Watt element is sensible; for more bulky jobs such as making connections to big earth tags (and burning off pink enamelling) a 3.5 mm bit in a 25 W iron is useful. For outside work on 12 SWG antenna wires a big 150 Watt iron maybe needed, particularly if its windy! A small iron would be quite useless since it has insufficient heat capacity and is quickly cooled by the work. For radio work you should use solder that is 60 % tin and 40 % lead; its melts at 188 ° C and is available with internal flux cores - size 22 SWG is suitable for most work and size 18 for external jobs. Do **NOT** use either plumber's solder or their flux since it is corrosive. The purpose of the flux is to make the solder flow more easily and it is essential to keep the parts being joined still while the solder is solidifying. Tinning is the process of initially getting a film of solder to adhere to the cleaned metal which makes the eventual joining much easier. It is done by placing the iron and solder together on the cleaned part to be tinned, whereon the solder will melt with a puff of smoke as the resin flux evaporates and the solder then flows easily over the metal leaving a bright thin film. Do not breath this smoke either! Usually the copper of PCBs is very clean and the solder will flow around the joint without tinning. Tinning is often needed where the joint is likely to be difficult or the parts are large compared to the size of iron.

Now to making the actual joint. Assuming that the work has been properly tinned, then it is merely necessary to hold the two pieces together and apply the soldering iron and melt in a little solder. This appears to require four hands but in practice the two pieces to be soldered are often held together mechanically (such as the components mounted in a PCB) leaving your two hands free to hold the iron and solder. If necessary a small vice maybe used and there are a number of small plastic ones with a suction grip to hold it to the bench top. Crock clips and clothes pegs have also been known to be used! The joint should be allowed to cool naturally (never blow on it or drop it in water!). When cool, it should look bright, shiny and smooth. If it has a matt or pitted look then it probably moved during cooling and will have to be reheated and a little more solder/flux applied. A good joint between a component lead and a PCB track is shown in section on the left - the poor joint on the right maybe due to insufficient heat or movement during cooling or poor adhesion of the solder to the parts due to dirt. It is not easy to describe all the techniques in words but practice makes perfect. If in doubt seek advice from an experienced amateur with a practical demonstration. I will end where I started by reminding you that for soldering "Cleanliness is next to Godliness". Eric Godfrey G3GC



Dead-bug Construction

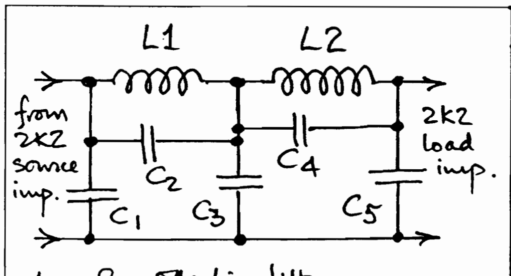
This unusual name derives from the technique of bread-boarding (or building a very early prototype) by mounting integrated circuits (the bugs) upside down with their legs sticking up in the air - thus having the appearance of dead animals. It is a very good method as it depends on a continuous sheet of copper, usually single sided printed circuit board, acting as a ground plane or continuous 0 volt line spread all over the circuit area. This is good for RF from a layout viewpoint and avoids having to worry about separate 0 volt wires for the supply. You dont have to do any drilling for component leads since they are all mounted or attached to the board by their earthy connections. As in any project, the layout of functional blocks is important and the simplest scheme is to lay the stages out in as long a straight line as is necessary and only bend round or back when space is insufficient. You must make certain that you avoid inputs and outputs being next to each other, especially when they are working at the same frequency. All leads are kept short to increase rigidity and avoid unnecessary lead inductance; the earthy component leads are soldered directly to the copper sheet using a hot iron for the shortest time sufficient for a good joint. The other non-earthy component joints are made up in the air, again with short leads, using just sufficient solder to hold them together. By the time you have added decoupling capacitors etc for the supply leads you will find that for most circuits, practically all the components will be firmly attached and the resulting structure is quite rigid despite having the most awful appearance! If you find that some component, such as a resistor to a supply line, is rather free floating in a mechanical sense it is very easy to secure it by adding an extra decoupling capacitor across the supply line at the resistor; one end of the capacitor is soldered direct to ground, the other to the common point between resistor and supply lead. These extra decoupling capacitors are seldom a problem electrically and generally are most beneficial, furthermore they are cheap and small so you can be generous with them. Occasionally you may feel that an extra capacitor to earth is not right for the circuit at the point that needs mechanical restraint, in this case consider anchoring the component to earth with a high value resistor, use the highest value you have in the junk box and it should be at least 100 times the circuit impedance at that point - if you dont know the impedance dont use resistors of less than a few MOhm! On the whole, extra resistors are best avoided since they might cause problems whereas extra supply decoupling capacitors are very unlikely to do so; practically any type of capacitor (and value) can be used but I recommend the use of 10 nF disc ceramics as they are small and cheap when bought by the bag! If the design needs integrated circuits they are mounted upside down by their supply pins, first decide the best orientation for the chip to give you the shortest input and output connections and mark the PCB with a star where pin 1 of the IC will be located. This avoids having to unsolder all the connections when you have later forgotten which end is which. Then carefully bend the chip's 0 volt pin back so that it points out sideways (actually needing a bend of just over a right angle) so that you can solder this pin direct to the ground plane. Then locate the supply pin(s) and solder a 10 nF decoupling capacitor direct from the pin to the ground plane, the supply lead is also soldered to this pin and run on to other supply connection points. The unused pins of the chip can be carefully bent towards each other to make more space for the other pins that have components attached to them; where there are several unused pins bend them slightly sideways as necessary to avoid them touching each other. The sketch alongside may help explain the general idea. Long interconnecting wires are best run flat in a rectangular grid pattern against the ground plane and can look quite tidy. Avoid running wires near any oscilators. If you have to make a structure, say to mount connectors or a control it is quite easy to cut and file PCB material to the right size and attach it by soldering along the joint between the ground plane and the back of the control mounting panel - if necessary add another piece of PCB offcut at right angles to the other two to give further rigidity. Finally the base PCB can be mounted on rubber feet. You will be surprised how easy it is to get good results. You can even rip it all apart and start afresh without any difficulties! Tim Walford.



Passive CW audio filter

Craig Douglas, G0HDJ writes, "One of my QRP rigs is a Hilltopper (designed by GM4JMU); it has an audio filter using passive components with a bandwidth suitable for SSB as well as CW. It works well without any ringing etc but an article in "Sprat" suggested that a lower frequency CW note at around 500 Hz is easier to listen to and aids discrimination in a noisy background. I built the 500 Hz version of this elliptic low pass filter and it is certainly very effective with a sharp cut-off. It is intended to be fitted in the audio line just before the audio gain control. The parts are not expensive; the inductors are available in the Toko 10RB and 10RBH series, available from Circuit and others. The table shows the theoretical component values and suggests the most suitable combinations of standard value capacitors. With a theoretical attenuation of some 40 dB for an increase in pitch of 50 Hz from

500 to 550 Hz, it should be very good at removing unwanted nearby signals. If you decide to build the 500 Hz version be aware that most commercial rigs are based on an 800 Hz offset and you may have difficulty working other stations. (It is of no consequence if you are receiving only.) I have included the figures for an 800 Hz low pass filter just in case!"



Low Pass Elliptic filter.

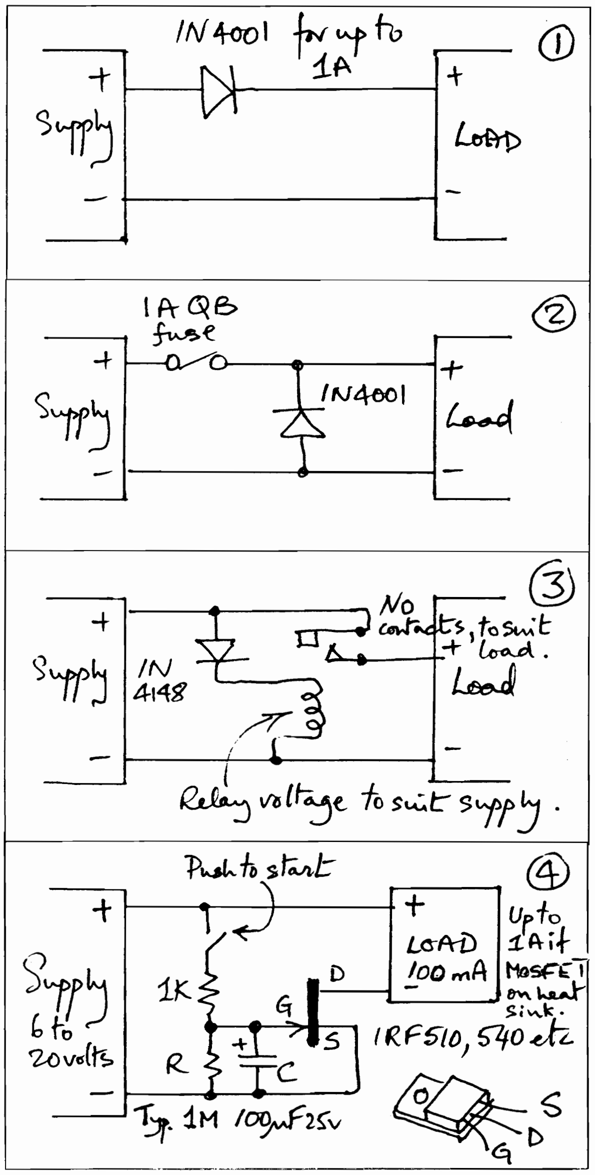
		C1	C2	C3	C4	C5	L1	L2
"500Hz" −40dB at 550 Hz	Theory	216 nF	40 nF	271 nF	121 nF	161 nF	708 mH	484 mH
	Actual	220 nF	39 nF	220 + 47	100 + 22	100 + 47/10	680 mH	470 mH
"800Hz" −40dB @ 870Hz	Theory	135 nF	25 nF	169 nF	76 nF	100 nF	445 mH	305 mH
	Actual	100 + 33	22nF + 2.2	100 + 68	68 + 6.8	100 nF	390 + 56	270 + 33

Construction Tip

Les Boddington G4JDC writes, "I have learned from bitter experience never to assume that the wires on components are clean and ready for soldering. I always gently clean them to clear off any oxidation by gently pulling the lead through the jaws of small pair of pliers. I know this may sound long winded and time consuming but it helps to insure a good joint." The Editor would comment that this is particularly necessary for those dreadful bags of a 1000 resistors that you get for 20 pence at rallies - they always seem to be covered in wax all over and have obscure values!

Reverse supply protection

Following my note last time on power supplies, I list some methods for protecting equipment from wrongly connected supplies. The first using a **series** diode, typically a 1N4001 for up to an Amp, only has the disadvantage that the supply voltage is reduced by about 0.7 volts and this figure does vary with load current. The second method of a series fuse and **reverse** diode does not suffer from significant volt drops but there is a very brief instant, on connection of wrong polarity supply while the fuse melts, when the load has a small reverse voltages applied to it. Only quick-blow fuses should be used. The third circuit has no performance drawbacks except that the relay does draw some extra current and it is more costly and bulky. The relay coil must match the supply voltage and its contacts must be adequate for the load current. The fourth circuit, which is intended as a timer for battery powered instruments to avoid a flat battery next time it is used, does also protect against wrong battery polarity. The time for which the load is connected to the battery varies with load and individual components but its simplicity makes up for the need for a little experimentation with C and R values to obtain the desired on time. As a very rough guide, the On time will be about CR seconds (C in microFarads and R in megOhms) but it depends considerably on the supply voltage and the characteristics of the MOSFET. A wide variety of n type MOSFETs can be used such as the IRF510, IRF540 etc. The supply can be up to about 20 volts. Tim Walford G3PCJ



Fabricating "one-off" printed circuit boards

Here is a method of making "one-off" PCBs which, with some care and attention to detail, can give a highly professional finish. For brevity I am tabulating the various steps. Read right through these notes first to help you understand what's involved. Do take care with the etching solution.

Preparation

1. You will need pencil, rubber, ruler, 0.1 inch graph paper, carbon paper, masking tape, very sharp knife such as a scalpel, 0.1 inch matrix vero-board, blank PCB, 2 or 3 small clamps or bulldog clips, ferric chloride etching solution (see later), the components, 0.8 mm drill, 2 x 3 inch lengths of 20 SWG, 2 mm ID brass shim tube, and various vessels.
2. Draw component placing and circuit layout on 0.1 inch graph paper, drawing it as viewed from the component side. The tracks will ultimately be on the other underside.
3. Try actual components on matching veroboard to check for tidy looking layout - alter as required.
4. Check and double check the circuit is correct with a sensible layout - avoid inputs near outputs etc.
5. Place carbon paper, carbon side up, underneath the drawing and trace over the pads and tracks. This will show the track layout as seen from the copper side, which will be a mirror image when viewed from the track side.
6. On the drawing, number each vertical line left to right and mark each horizontal line with letters top to bottom, a b c etc. List the numbers to be drilled to the right of each line.

Drilling

7. Cut the PCB to the required size, allowing sufficient overlap for fixing holes etc.
8. Clamp piece of veroboard to the uncoppered side of the PCB and square it up carefully.
9. Locate and drill (0.8 mm) two of the holes at each end. Thread two lengths of 20 SWG through them and twist the ends. Remove the clamps and trim the twisted wires to about 1/4 inch on top of the board.
10. Place the masking tape so that line "a" only is exposed. Number against every fifth hole on the tape.
11. Drill all holes on line "a" as indicated on the drawing. Move the tape down to expose line "b", keeping the numbering in place, and drill all holes in line "b". Continue until all holes in all lines are drilled.
12. Remove the two clamping wires. You should now have an accurately drilled, but plain PCB.

Manufacture

13. Scrub the copper side with wire wool until it shines. Remove any burrs remaining on the holes by lightly twisting a larger drill into the offending hole with your fingers.
14. Cut a 3 inch length of PVC insulating tape into 2 mm wide strips longways for the tracks. Do this very carefully with a scalpel or razor blade or Stanley knife up against the ruler, with the insulating tape laid on ex-address label backing to preserve the tape's stickiness. Drawing a 2 mm scale each end will help. Do more if required.
15. If required, circular pads can be made by sharpening the end of a 2 mm brass shim tube (obtainable from model shops) with a file, and cutting them from a strip of insulating tape.
16. Cover and join the holes on the copper side of the PCB with the tape strips as per the carbon side of your drawing. Apply the pads at the ends over the holes if required. This pattern is the circuit that will be left after etching. Press down overlaps and edges so that no etching solution can get underneath the taped areas. The rounded handle end of a scalpel is good for this. For a satisfying finish, make sure the tape strips are tidy and parallel or at right angles to each other. If there is room, "letraset" your call sign and project title, date etc, on what would otherwise be a blank area. Double check your work.
17. Find a shallow non metallic dish just large enough to take the PCB (saucer or meat dish etc.). Pour in about 3/8 inch of ferric chloride etch solution (see next paragraph) and carefully place the PCB in it. If the board is made of SRBP, which is a brownish colour, the board will float and it should be placed in the solution copper side down so that the solution is in contact with the copper. If the PCB is made of fibreglass, usually a dull greenish colour, then the board will sink and it should be placed in it copper side up so that you can see how the etching is going. Every 5 mins or so gently rock the dish to move the solution around; after about 15 to 20 mins the copper will begin to vanish, usually at the board edges first, and you should then examine it every few minutes so that you can remove it from the solution just as soon as all the unwanted copper has been removed. Use plastic or non metallic tongs to pick up the board and it is a good idea to wear rubber gloves.

18. Ferric chloride etching solution is usually made up from crystals, largish irregular yellow globules, which can be obtained from electrical suppliers such as Cirkit, Maplin etc. and maybe some chemist shops. Follow the instructions, which usually come with the crystals, on how to make up the solution. If there are none then you should use them at the rate of 250 grams of crystals dissolved into 0.5 litres of water. First asses how much etching solution is needed by filling the etching dish with water to a sufficient depth to cover the board and then pour this into a measuring jug. 0.5 litres will be more than enough for most boards and the surplus can be kept for the next board in a labelled sealed non metallic container safely stowed away from food and children etc.. For making up 0.5 litres of etchant, first place 0.25 litres of cold water into a non metallic dissolving vessel, then weigh out 250 grams of crystals in something like an old ice cream tub which can be later thrown away. Gingerly add the crystals to the cold water, NOT the other way round since heat is generated as the crystals dissolve. After a few minutes carefully add a further 0.25 litres of hot water to the dissolving vessel and stir with a plastic rod till all the crystals are dissolved. Transfer sufficient to the etching vessel and store the remainder. The solution will be warm which speeds up the etching. If you have many boards to do at once, the continued application of a little heat from something like a 60 Watt light bulb will keep the temperature up. Obviously this needs the etching vessel to be safely mounted over the heat source. If you do suffer from a splash of ferric chloride wash immediately with lots of cold water. 0.5 litres of solution will etch up to about eight boards 100 x 160 mm before the strength runs out but the last few will take longer. Generally it is best to avoid keeping part used solution.

19. When the board has been fully etched wash it thoroughly with water. Do NOT pour the spent etching solution down the drain and dont get it on stainless steel sinks, dilute it further with water and pour into a hole in waste land in your garden. Remove the PVC tape strips and pads then scrub once more with wire wool. If you can, spray the board all over with PCB lacquer (available from electrical suppliers) at this stage to stop it tarnishing. You can solder straight on to the lacquer.

20. If the project requires a ground plane of copper all over one side, then use double sided copper clad board and fully cover the wanted ground plane copper side with wide PVC tape or selotape to prevent the etching solution coming into contact with the wanted copper. Be careful to avoid air bubbles and make sure any strips of tape overlap with no curled up edges. Use the scalpel to cut carefully round the edge of the PCB to remove the excess tape, with the board laid ground plane side down on a flat piece of wood. Press the edges down carefully. You will also need to countersink the non-earthy lead holes to prevent those leads shorting to the ground plane; this is done after etching . You will also have to give careful thought about how all earthy component connections are made to the ground plane, since it is not practical to solder the earthy leads of all the different types of parts to the ground plane underneath the part. It can however usually be done with resistors and ceramic capacitors and these make good points for soldering both sides of the board to complete the earth path.

Fitting components

21. Just a few points regarding the fitting of components. If using integrated circuits, it is a big advantage to use IC sockets or holders - note they include a pin 1 locating mark. Fit resistors so that their values can be read left to right or bottom to top. Fit capacitors so that their value can be read easily. Care in soldering components "square" and at a constant height etc will give a pleasing result. If the board has a ground plane be careful not to push in shouldered leads, such as integrated circuits, so far that their shouldered pins short to the ground plane. Derek Alexander G4GVM

RF voltmeter

None of you have submitted anything for the Question Corner so her is a space filler! It is a high impedance peak reading rf voltmeter, which when connected to a digital voltmeter with 10 MOhms input impedance will show the RMS value of the voltage providing it is sinusoidal and sufficiently large for the approx. 0.1 volt drop in the diode to be ignored. It will provide some indication for rf voltages down to tens of millivolts but the actual value shown will be way out and it can only be used as a rough indicator. It is good to VHF if the component leads are short. Tim Walford G3PCJ