25 Construction and Workshop Practice

Eamon Skelton, EI9GQ

There are many reasons why radio enthusiasts might choose to build at least some of their own equipment. Even in a world where commercially made equipment is readily available, there are some advantages to rolling your own. The most obvious advantage is the potential to save a lot of money by building your own rig instead of buying off-the-shelf equipment.

It is possible to build useful radio equipment using components that were recovered from scrap equipment. Surplus components can often be bought at very reasonable prices from specialist suppliers or from Internet auction sites. A well stocked junk-box is a very valuable asset to the radio constructor.

Cost is not the only consideration for the home constructor, there are other advantages to building your own. The home constructor enjoys a great deal of flexibility when he/she builds a new item of equipment. In most cases, the designer, builder and the end-user will be one and the same person. This usually means that the operator will end up using equipment which is built to his specifications rather than generic equipment which may be less suitable for the task.

The home constructor can also decide where to concentrate the available resources. For example: he might decide to use a £10 transistor in the front-end of a homebrew receiver, even if it only gives a small advantage over a £1 transistor. This small advantage may well be worth £9 extra to someone who wants to have the best possible receive performance. This kind of flexibility is not usually available to the commercial manufacturer.

There are many situations where commercially made equipment is not readily available and home made equipment is the only option. Radio equipment for LF (low frequency) and for the higher microwave bands is usually home made.

Homebrew is not always the cheapest option, some specialised components can be difficult to find and expensive to buy. If you value your time, homebrew can be very expensive indeed. It is not unusual for complex projects to take dozens or even hundreds of hours to complete.

So why would anybody want to waste time and money by building amateur radio equipment when second-hand commercially made equipment is quite cheap and readily available? The most compelling reason for building your own equipment is the knowledge that you will gain from building it and the great sense of satisfaction that goes with operating a homebrew station.

Homebrew radio is not the exclusive preserve of a small number of technical gurus. Home construction is accessible to all radio enthusiasts, young and old, the newcomer or the experienced constructor, the able-bodied and the disabled. Anybody with a soldering iron and a handful of components can build a working circuit. Basic components like transistors, diodes, resistors and capacitors are very cheap and easy to find.

The cost of failure is very low. If a circuit doesn't work first time, you can easily modify it or even completely re-design it and try again. You will probably learn more from the projects that fail to work at the first attempt than you will learn from the projects that work perfectly the first time you apply power to them.

Not every homebrew project involves electronic components and solder smoke. A lot of radio equipment and test instruments are software based. If you use a software defined radio (see the chapter dedicated to this subject), you could completely redesign and rebuild it without ever modifying the hardware or even taking the lid off the box!

Your project might be as simple as measuring and cutting a wire dipole or as complex as building a multi-band transceiver. It is a good idea to start with a few simple projects before you progress to more complex projects. It would be possible for the new constructor to build a very complicated circuit as a first project, but it can be very discouraging for the newcomer when a circuit just refuses to work. The experience gained from building and debugging a few simple projects will be very valuable when you start to build more elaborate circuits.

THE WORKSHOP

Only a small minority will be fortunate enough to have a separate radio shack for operating and a dedicated workshop for construction projects. Most of us have to make the best of limited resources. The radio shack usually serves as the workshop and increasingly as a computer workstation. The shack may have to be shared with other family members or be used as a centre for other hobbies and activities.

My first radio shack was a small table in the corner of the bedroom. I still have painful memories of standing on upturned IC pins in my bare feet! Even now, despite the fact that I have a dedicated outdoor radio shack, some of my homebrew activities take place on the kitchen table.

A garden shed or a bench at the back of the garage can be pressed into service as a radio shack. An attic, basement or spare bedroom can make a good radio shack. Caravans, campers or even boats (subject to local planning regulations) can be used as a radio shack. If space is very limited, a small cupboard, a wardrobe or a roll-top desk can be used as the radio shack.

TOOLS AND INSTRUMENTATION

Given unlimited resources, the home constructor can build just about anything that could be built in a factory or a professional machine shop. A few amateurs will have the facilities and the necessary skills for both electronic construction and metalworking. Most of us have to work in less ideal circumstances. A typical

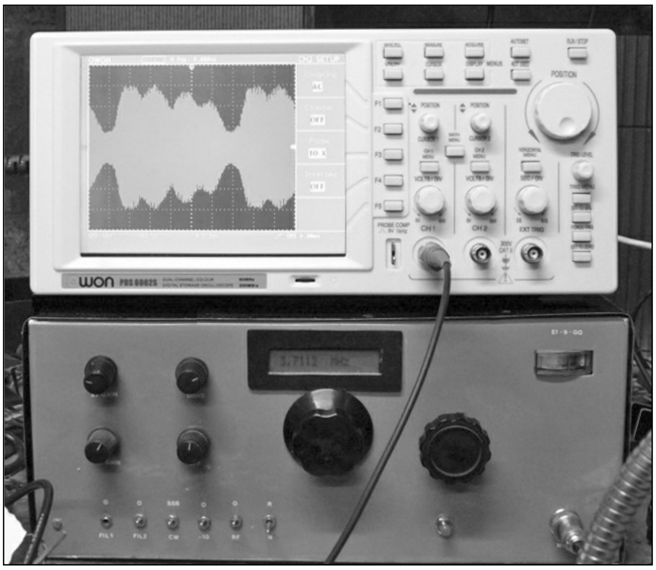


Fig 25.1: Using a digital oscilloscope to monitor the output of a home made SSB transmitter

radio shack will contain a basic electronics tool-kit and a few very basic metalworking tools.

The simplest practical tool-kit will consist of a few basic hand tools like a soldering iron, screwdrivers, pliers, side-cutters etc.

A simple multi-meter is also an essential part of the home constructors tool-kit. I have owned many different multi-meters over the years and have found that the ability of a meter to survive a fall onto a hard floor is inversely proportional to the price of the instrument. Even if you have an expensive multi-meter, it is a good idea to keep a cheap one in the shack for use in risky environments.

The choice between analogue and digital meters is largely a matter of personal preference. Digital meters will offer greater accuracy and resolution; analogue meters tend to be better at tracking slowly changing values of voltage, current or resistance. Very cheap digital meters tend to be almost as accurate as more expensive models, so there is little to be gained by paying extra for a high quality instrument. My favourite digital meter cost less than £4 from a local supermarket. When buying an analogue multi-meter, it is worth spending a little more on a meter with a sensitivity of at least 20kΩ per volt. There is much more on meters in the chapter on Measurements and Test Gear.

Most general purpose soldering irons can be used for electronics. A temperature controlled soldering station is desirable, but not strictly necessary for most construction work. A relatively high powered iron is useful for heavier work and for de-soldering. A lighter iron in the 15-18 watt range is more suitable for delicate electronics work. A 25 watt iron is a good compromise between the two.

More advanced test equipment like oscilloscopes, signal generators, and spectrum analysers are very useful, but they are not absolutely essential (see the Measurements and Test Equipment chapter).

The basic tool-kit described above is enough to get you started. Once you have gained some construction experience, you can buy or even build any other instruments that are required. Surplus and second-hand test equipment from industrial, education and military sources can sometimes be found at radio rallies and Internet auction sites like eBay. Some specialist suppliers offer surplus and used test equipment. Buying from a specialist will be a lot more expensive than taking pot luck at a rally or auction, but the extra expense can often be justified, especially if the equipment comes with a guarantee. A specialist supplier will usually be able to offer a repair and calibration service for the test equipment that they sell.

The more experienced radio constructor’s shack will usually have a few electronic test instruments. An oscilloscope is one of the most useful instruments. Professional laboratory grade oscilloscopes are quite expensive, though surplus and second-hand scopes are quite widely available. Digital storage scopes were once considered to be the state-of-the-art in test equipment. The ready availability of high speed, high precision digital to analogue converter ICs and other recent advances in digital electronics have greatly reduced the cost of digital scopes. **Fig 25.1** show a digital oscilloscope used to monitor the output of a home made SSB transmitter.

Digital oscilloscopes which use a standard PC (personal computer) to process and display sampled signals are now available ready built and in kit form.

Another instrument which is of great value to the amateur constructor is the digital frequency counter. The cost of new frequency counters has been falling steadily for many years. The availability of modern micro-controllers like the PIC micro have made the job of building a home made frequency counter [1] a lot easier (**Fig 25.2**).

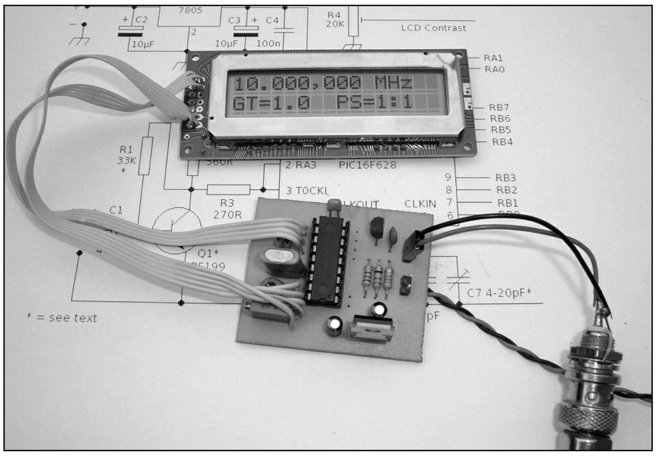


Fig 25.2: Home-made frequency counter

A good signal generator is a useful tool for the home constructor. Again, lab grade signal generators are very expensive, but a simple RF signal generator is a fairly easy construction project. Sophisticated test instruments like spectrum analysers and network analysers are extremely expensive and rarely found in the amateur's shack. This situation is changing gradually as faster, better and cheaper digital instruments are developed.

A standard PC fitted with a sound card and some clever software makes a very useful audio signal generator, oscilloscope and spectrum analyser. Unfortunately, PC sound cards have an upper frequency limit of about 22kHz. A relatively simple RF to AF converter can be used to mix an RF signal down to audio frequencies so that is can be sampled by the PC. Most, if not all, PC sound cards use input coupling capacitors. This means that a sound card based oscilloscope will not be able to display DC signals or very low frequencies below a few Hz. Oscilloscope and spectrum analysis software is available for most types of home computer and operating systems. Much of this software is open-source and/or available free of charge.

METALWORKING

Metalworking or 'chassis bashing' is a very important part of amateur radio construction. Commercially made metal and plastic enclosures are available at reasonable prices from most component suppliers. You may find that such enclosures are not exactly suited to your needs and you will need to modify them. For some projects, a home made enclosure will be the only option.

Surplus or scrap equipment is another good source of high quality metal enclosures, but you may find there is just as much work involved in modifying an ex-equipment enclosure to suit your project as would be required to build a new one from scratch. ‘Murphy's Law’ says that all of the holes and openings on the front and rear panels of a second-hand enclosure will be the wrong size, wrong shape and in all the wrong places.

The successful home-brewer will need to master the art of mechanical engineering, metalworking and fabrication. Most amateur constructors will not have access to machine shop tools like lathes and milling machines. However, a basic tool-kit is all that is needed for most construction projects.

You will need a selection of basic hand tools. The tools required for metalworking are a little different to those in our electronics tool-kit mentioned earlier in this chapter. A minimal tool-kit would include a good hacksaw, a drill, an engineer’s rule (steel), a bench vice and a selection of basic hand tools like screwdrivers, files, spanners etc.

A more complete metalworking kit might include a few more specialised items like sheet metal shears, a metal nibbling tool, a drill stand and an assortment of reaming tools, hole cutters and chassis punches.

An old fashioned hand drill can be useful for delicate jobs where an electric drill would be too fast and difficult to control. A small 12DC drill is useful for drilling small holes in PCBs. Dremel and similar types of rotary multi-purpose tools can be very useful for drilling, grinding and polishing. A bending brake is useful for making nice sharp and straight bends in sheet metal. You can easily make your own bending brake from a couple of lengths of angle steel and a few bits of metal hardware.

Obviously there will be a limit to how much you can spend on your hobby, but you should try to buy reasonably good quality tools. You can cut corners (if you will pardon the pun) in some areas. In most respects, a cheap hammer is just as good as a more expensive one. On the other hand, you certainly shouldn't compromise on the quality of screwdrivers, hacksaws, drill bits or side-cutters.

SAFETY

Safety should always be the first and most important consideration for the home constructor. Some of the dangers facing the home constructor are quite obvious. Every radio amateur will know that you shouldn't work on live mains-powered equipment because of the risk of electric shock. The dangers of working on low voltage equipment might not be so obvious. Never wear jewellery when working on electrical equipment. Even a few volts from a high current source will melt a metal ring or bracelet.

Always wear eye protection when working with power tools. You should always wear a dust mask when cutting or grinding hazardous materials like fibreglass PCB laminate.

Only a fool would climb a tower without a proper safety harness, but much lower platforms such as chairs or tables can be dangerous if they collapse while you are standing on them.

The risk of injury to yourself and others can be greatly reduced if you apply a little bit of common sense. Make sure that live equipment is properly insulated and housed in a safe enclosure. Even if you feel that you know what you are doing and feel comfortable about working on live equipment, there is always the danger that someone else will be electrocuted by it. A visitor to the shack, a young child or a stray animal may not be aware of the danger.

You should also take great care when using and storing chemicals. Trays and tanks containing etching or cleaning chemicals can be very hazardous to children and small animals. Don't leave exposed etching or developing trays unattended. Don't place dangerous chemicals near the edge of the workbench where they can be reached by children or accidentally knocked over. Always wear rubber gloves and eye protection when working with chemicals.

Take particular care to keep PCB developer and other caustic chemicals away from your eyes. Remember that paint, thinner, paint remover and other solvents are usually highly flammable and could pose a serious fire hazard. It is worth noting that many of the things that we keep in our radio shacks would not be allowed in an industrial or commercial environment because of health and safety regulations and conditions imposed by insurers.

Fumes from soldering and chemicals can pose a health hazard. As hand soldering is usually done very close to the operator’s face, it is very important to ensure that your workbench is well ventilated. Batteries should not be charged in a confined space because of the risk of fire or explosion due to the build up of hydrogen gas. Hot soldering irons should be placed in a proper stand to reduce the risk of burns and damage to equipment.

Amateur radio is just a hobby. There is no need to take unnecessary risks to save time or to reduce cost. If you need professional help or expert advice, don't be afraid to ask for it. Experienced radio builders are usually willing to help out if you have a query. If you are a licensed amateur, you can just ask on your local homebrew net frequency. Your local radio club may be able to offer help and advice. There are several homebrew mailing lists, web forums and Usenet groups. If you are concerned about the safety of a particular tool or product, ask the manufacturer for advice or a safety data-sheet.

More safety advice can be found at the end of this chapter.

MATERIALS

The first requirement is an elementary knowledge of the materials useful and normally available to constructors.

Metals

Aluminium and aluminium alloys

Typical uses: cabinets, boxes, panels, masts, beam antennas, heatsinks.

These are good electrical conductors and are the lightest in weight of the normal metals available. They are non-magnetic, of medium to high cost, and available in sheet, rod, tube and other forms. The annealed quality usually bends and machines easily but soldering requires special fluxes and solders, and often the soldered joint is not good electrically. With the unknown quality of aluminium that most amateurs encounter, soldering is best left alone or experimented with before committing the project to this method of fastening.

Long-term corrosion is also a problem with soldered joints in aluminium unless specialist methods are applied to prevent this. Adhesive bonding is very good but such joints are not usually reliable electrically. Aluminium is non-corroding in normal use, but direct contact with brass or copper should be avoided as these react and encourage corrosion with the resultant troubles of poor or non-existent electrical contact (see the section on corrosion.) Nickel-plated or stainless-steel screws, nuts and washers, rather than brass or steel, should be used to reduce the chances of corrosion around this type of fastening.

A metal that looks similar to aluminium, and which may come into the hands of the unwitting, is magnesium alloy. Filings and chippings from this metal are highly flammable. They can burst into a glaring flame with the heat generated by filing and drilling, and trying to put out such a fire with water only makes matters worse. (Factories where this alloy is machined use a chemical fire extinguisher, one type of which goes under the title of DX powder!) It is difficult to tell the difference by just looking, but size-for-size magnesium alloy sheet weighs less than aluminium and usually on bending there is a 'granulating-cum-scrunching' feel.

For reliable bending of sheet aluminium up to 1.5mm thick, it should be possible to bend a sample strip back on itself and hammer the fold flat without breakage. Annealing aluminium is possible, but under specialised conditions. Before any heating is attempted it is essential to degrease the part thoroughly.

A workshop 'dodge' which sometimes works, and is certainly worth a try if bending or forming proves difficult, is to heat the aluminium slowly (the slower the better) to such a temperature that a small pine stick, when rubbed on the heated surface, just chars, but does not ignite (equivalent to approximately 300°C) Care needs to be taken to avoid overheating, for this will destroy the properties of the aluminium. This method of localised annealing has proved effective when bending tube or rod for the folded dipole elements of beams. In this case the area to be annealed is small and the 'dodge' can be applied easily. Another method, suitable for larger areas, is to rub soap on the surface and warm the aluminium until the soap turns black. However, it is essential that it is the aluminium which is warmed by the blow torch or hot air gun and not the soap. This is ensured by applying the heat to the non-soaped face. Any bending or forming should be carried out as soon as the annealed piece has cooled, because most grades of aluminium alloys start to re-harden and may actually improve in strength after such heating. Do not attempt to speed up the cooling process by quenching in water. Allow the part to air cool. Note that during the heating of the component there must be no visible change in the surface appearance of the aluminium, otherwise the aluminium will lose its properties and disintegrate. This fact can be verified, if you are so minded, by attempting to bring a small piece of aluminium sheet to red heat.

Heat treatment of aluminium and its alloys is a specialised process used to improve a particular characteristic of the metal, and is carried out in controlled conditions not usually available in the home workshop. Technically the above techniques are more of a part softening process rather than true annealing.

For interest, the usual method of commercially annealing most aluminium alloys is that the part is 'soaked' for about two hours at 420°C, air cooled, and then soaked for a further two to four hours at 225°C. All this is performed in a furnace or salt bath which has its temperature closely controlled. Most, if not all, of the aluminium used in extruded tube or sections has undergone this type of treatment immediately prior to extrusion, ie in the billet form.

Brass

Typical uses: Morse keys, terminal posts, weatherproof boxes, extension spindles, waveguides and other microwave components.

This is an expensive but good non-magnetic electrical conductor, and is available in sheet, rod, tube and other forms. It can be soldered easily but adhesive bonding can prove difficult. For work involving bending or forming, the most suitable grade is ductile brass. For panels and non-formed parts, the half-hard and engraving brasses are adequate. It can be annealed like copper, though care is necessary as brass is nearing its melting point when heated to bright red. Brass is non-corroding in normal use but reacts with aluminium and zinc.

Copper

Typical uses: heatsinks, coils, antenna wires, PCBs, tuned lines, waveguides, earthing stakes and straps.

This is a very good, though expensive, electrical conductor. It is non-magnetic and is available in sheet, rod, tube and other forms. Before work of a forming or bending nature is attempted, this metal should be annealed by heating as uniformly as possible to a bright red heat and air or water cooled. If considerable bending is required, this annealing should be repeated as soon as the metal begins to resist the bending action. In the annealed state, copper bends very easily. Soft or hard soldering present no problems but adhesive bonding can be troublesome. It is non-corroding in normal use but does react with aluminium and zinc, especially in an out-of-doors environment.

Steels

Typical uses: Masts and tabernacles, screw fixings, guy wire stakes and parts subject to high wear or heavy loads.

These are electrical conductors, which are magnetic except for the expensive stainless types. They are cheap and available in numerous forms and qualities. The common grades are called mild steel or GCQ (good commercial quality) steel. The black quality steels are usually cheaper and used for such things as the stakes for mast guys, or similar 'rough' work. Silver steel is a special grade suitable for making tools, pivot pins and other items which require the parts to be tough or hardened and tempered. Most steel sheet forms commonly available will bend, solder and machine easily, and can be annealed by heating to bright red and allowing to cool slowly in air. Do not quench in water or in any way cool rapidly, for this may cause some steels to harden. Corrosion is a problem unless plated or well painted. For outdoor use the commercial process of galvanising is perhaps the best form of protection. The next best thing is a few coats of paint.

Tin plate

Typical uses: Boxes, screening cans and plates, light-duty brackets, retaining clips, spacers.

This is a good electrical conductor. It is very thin steel coated with an even thinner layer of tin on each side, and is magnetic, cheap and available in sheet form up to 0.5mm thick. It can be easily soldered, bonded, bent and machined, and is non-corroding in normal use. Cut edges should be re-tinned with solder if the full benefits of the non-corroding properties are required. It can be annealed but this will destroy the tin coating. This is a 'friendly' metal to use and is normally readily available in the form of biscuit tins and similar containers, hence the old timers' expression 'an OXO tin special'.

General comments on metals

All of the above metals work-harden and will break if repeatedly flexed at the same point. Annealing removes the effects of work-hardening, providing it is carried out before the part is over-stressed. There are professional standards which classify the above metals and each is given a specific identifier code. Fortunately, amateur constructors do not normally need to enter this maze of professional standards, and metal suppliers usually understand that to most of us steel is steel and brass is brass!

Plastics

Plastics are electrical and thermal insulators, and are not suitable as RF screens unless the plastic is specially metal coated or impregnated. The insulation can normally be considered as excellent for most amateur purposes.

The following is a brief description of the more commonly available and useful plastics, and is a very small selection of the many plastics in use today.

Laminates

Typical uses: PCBs, matrix boards, coil formers, insulating spacers.

Various base fabrics such as paper, cotton, glass, asbestos etc, are bonded together by selected resins, and usually compacted and cured under pressure. The combination of the resin and the base fabrics produce laminates which may be used for many applications. Most are available in sheet, rod and tube form. The cured laminate cannot be formed easily. Normal machining is possible, particularly if attention is paid to the lay of the base material. Drilled or tapped holes should be arranged so that they go through at right-angles to, and not in the same plane as, the laminations.

Where components made from this material are exposed to the elements and expected to insulate, the glass-based laminates should be preferred. This also applies where the dielectric properties are important (VHF converters, RF amplifiers etc). The normal heat generated by valves and similar electronic components will not harm these laminates but the glass or asbestos fabric-based laminates should be used for higher temperatures (100-140°C). Costs range from expensive for the paper and cotton bases to very expensive for the nylon and glass bases. Glassfibre repair kits for cars are a useful laminate for weatherproofing antenna loading coils and making special covers or insulators. The filler putty supplied with these kits may contain metal.

ABS (acrylonitrile butadiene styrene)

Typical uses: antenna insulators, coil formers, handles, equipment enclosures.

This is expensive and is available in natural white coloured sheet. It machines easily and can be formed by heating, similar to Perspex®. Bonding requires proprietary adhesives. A tough plastic and a good insulator.

Acetal copolymer

Typical uses: bearing bushes for rotating beams, Morse key paddles or electrical parts such as insulators or feeder spacers.

This is a medium- to low-cost plastic and is available in white or black rod and sheet. It machines very easily without specialist tools and is a useful plastic to have available in the workshop. It cannot be formed easily. Bonding requires proprietary adhesives and such joints are usually the weakest part of any assembly.

Acrylics (Perspex®)

Typical uses: decorative and protective panels, dials, Morse key paddles, insulated fabricated boxes and covers.

A medium-cost plastic which is available in clear or coloured sheet, rod or tube. It is non-flexible and can shatter or crack under shock or excessive loads, although it is often used for see-through machine guards. The clear sheet is ideal for covering and protecting the front panels of equipment. Perspex® may be formed by heating but not with a flame as this plastic is combustible and gives off unpleasant fumes if burnt. If placed in a pan of water and simmered (or in an oven) at around 95°C, the plastic softens and can be formed or bent very easily. Forming should be stopped and the work re-heated the moment hardening or resistance to bending is felt, otherwise breakage will occur.

Bonding requires proprietary adhesives. A properly made bond is structurally sound and can be transparent. Normal drilling, sawing and filing are straightforward, providing the work is adequately supported. Most sheet forms are supplied covered with protective paper and it should be worked with this left in place. It is not very heat resistant and should not be placed in direct contact with any heat source such as lamps, heatsinks and valves. Where transparency is not required a better plastic to use would be Acetal or, for particularly tough applications, ABS.

Nylon 66

This is a cheap- to medium-cost plastic, and is available in sheet, rod or tube. It is usually supplied in its natural creamy white colour. It can be machined but it does tend to spring away from any cutting edge, making tapping or threading difficult. It is not easily formed. Proprietary adhesives are available which claim to bond nylon successfully. There are other types of nylon but most of these are expensive and intended for special applications, such as bearings, gears etc. For most amateur purposes the Acetal copolymer, mentioned previously, is generally an easier material to use. Note that nylon is susceptible to moisture absorption and ultra-violet rays.

Polyethylene (Ultra High Molecular Weight)

A medium- to high-cost plastic, normally available in sheet and rod. It is usually supplied in its natural white colour and it can be machined easily but not formed. Proprietary adhesives are available for bonding. This is an ideal plastic for outdoor components such as insulators, feeder separators etc as it is virtually rot-proof.

Polycarbonate

This is an expensive material, normally available in transparent sheet form. It can usually be bought at builders' merchants where it is sold as vandal-proof glazing. A very tough plastic, virtually unbreakable (bullet proof!) and, though it can be machined, it will wear out normal tools very quickly. It is ideal for making an insulated base for a vertical mast/antenna and in other areas where impact, high loads and temperature changes would rule out other less-durable plastics. Polycarbonate is particularly sensitive to ammonia and clear sheet will turn white, become brittle and fail if this substance is used for cleaning it. A common trade name for polycarbonate is Lexan.

Polypropylene

This is an expensive plastic, normally available in opaque-coloured rod form. It can be machined and formed but not bonded. It can be sensitive to prolonged frictional contact with metals, particularly copper, and disintegration can occur in these circumstances. Because of its strength and resistance to atmospheric attack, it is usually used by amateurs in its rope form for halyards, mast guys etc.

The twisted strands are normally melt-welded together to prevent fraying, using a soldering iron. A naked flame should not be used as a substitute for the soldering iron as this plastic burns and melts, and burning droplets can go anywhere, even on the hands!

Polystyrene

This is a relative cheap plastic and is available in a variety of types, shapes and colours from black to transparent. It can be formed, machined, painted and bonded very easily - a model maker's delight! Usually used by radio amateurs for coil formers, insulated extender spindles and in other areas requiring insulation. It is a particularly tough flexible material, although some transparent types can be brittle. Some forms are also heat sensitive. A paper/card laminate of this plastic is available and this is very useful for making mock-ups of cabinets, boxes etc. Model-making suppliers usually stock extruded polystyrene sections, some shapes of which can be utilised in making bezels and other cabinet embellishments.

PTFE

This is an expensive material, noted for its excellent dielectric performance and low frictional properties. It is available in sheet, tube and rod, and is normally supplied in its natural off-white colour. PTFE is easy to machine with normal wood or metal turning tools. However it is extremely difficult to grip the PTFE in a normal lathe chuck or a vice as it has a consistency like putty and deforms easily under pressure. Round PTFE rod is normally turned by gripping in a collet chuck. Bondable PTFE in sheet form is supplied with one side burned by liquid sodium that allows normal contact adhesives to be used. This sheet has a white side and a dark brown side. PTFE is usually used for low-friction bearings, insulators (up to UHF), capacitor dielectric and the nozzles of desoldering guns. The fumes from overheated PTFE are very toxic.

PVC

This cheap material is available in many forms including rod, tube and sheet. It is usually grey or black in colour, and can be easily machined, formed and bonded. Proprietary adhesives should be used (although hot-air welding with a filler rod is also possible) but skill is required to produce structurally sound joints. Certain of the building types of PVC encountered seem to have some conducting capabilities which can lead to problems if used in electrical or RF applications.

A suggested test for this is to try to cook a small sample in a microwave oven (alongside a cup half full with water), and if no metal is present the PVC should stay cool to just warm. However, note that microwave ovens operate on the agitation of water molecules so this test is not reliable for some drainage pipes made from PVC which contain carbon black or other filler materials that have poor RF properties even at lowish frequencies. 40m traps wound on PVC can exhibit very low Q factors!

PVC insulating tapes are strong, cheap and normally self-adhesive, and are supplied in a variety of colours and widths intended for wrapped insulation. Some of the poorer-quality tapes do not weather very well and suffer adhesion failure with the first frost.

Adhesives

Many modern adhesives are hazardous and it is essential to follow meticulously the manufacturer's instructions when mixing, using and curing them. Most are insulators and unsuitable for electrical joints. Five general rules should be applied for bonding:

- Degrease the parts thoroughly; even finger marks impair results.
- Roughen the joint faces unless a transparent joint is required.
- Do not place bonded joints under a peeling type of load.
- Ensure that the work is dry and warm.
- Wear protective glasses and gloves and have the necessary first aid chemicals readily to hand.

Epoxides

A group of medium to expensively priced, cold- or heat-setting resins (usually self-generated heat) that can be used for bonding, surface coating, laminating or encapsulation. Air and gas bubbles are the biggest problem with encapsulation (this work is carried out professionally under vacuum). The problem can be minimised by warming the work and the resin to around 40°C and providing a generous shrinkage allowance with a large pouring area which can be cut off from the cured encapsulation. Careful thought should be given to the necessity of encapsulation, for once completed, the encapsulated module cannot be altered or repaired. Encapsulation is usually used when circuits are subjected to harsh vibrational and environmental conditions. (See also 'silicone sealants'). The correct choice of encapsulating epoxides is critical. Incorrect types when cured shrink and can compress components so much that they change shape and often fail, eg electrolytic capacitors.

These are usually two-part adhesives and require careful mixing just prior to use. A structural joint should not be over-clamped during bonding, and a bonding gap of typically around 0.05mm is required for the joint to be made properly. In other words, don't squeeze out all of the glue! Surface coatings can be applied by dipping, spraying or brushing. Flexible resins are usually used for this type of work and are ideal for protecting beams, traps etc.

Cyanoacrylates ('Superglue')

These expensive adhesives are available in various grades, each intended for bonding a particular set of materials. The low- to medium-viscosity grades are suitable for most amateur work. They are scientific marvels of bonding and as such require correct and proper application to ensure success. Releasing or debonding agents are available and it is a wise precaution to keep some of this handy in case of accidental bonding of fingers etc. The household type of superglue is unsuitable for outdoor use as the bond fails when moisture is present. If you accidentally glue your fingers together holding them under a cold water tap and gently easing them apart causes the bond to peel away. These adhesives should be used and cured in well-ventilated conditions. It is advisable to wear protective glasses when using this adhesive.

Toughened acrylics

These are expensive, fast-curing adhesives intended for structural joints. Various types are available and are usually supplied in two parts - the 'glue' and the primer/activator. The glue is applied to one side and the primer to the other. They are suitable for use on most of the materials already mentioned, but some may not be used with certain plastics as they dissolve the material and eventually the joint fails. They are usually easier to use than the cyanoacrylates.

Other Materials

Silicone rubber compounds

These are medium-priced materials, available as paste in squeeze tubes and as a liquid in tins. When cured, they normally set to give a white or translucent silicone rubber finish. They are ideal for encapsulation and the sealing or weatherproofing of antenna connecting boxes and similar out-of-door items. The electrical insulating characteristics are excellent and can be used to prevent parts from vibrating in equipment used for mobile or portable work. One type of this compound emits acetic acid during curing and this may damage some insulators and component connections.

Though not normally sold as such, one type of this compound has been used successfully as a resilient adhesive for structural and pressure-sealing joints on metal, plastic and glass.

Self-amalgamating tapes

These are a form of insulating tape which, when stretched and overlap-wrapped around cables, coaxial plugs etc, will amalgamate or flow together as one. They are reasonably priced and available in widths up to 50mm either with or without a self-adhesive face. Excellent weather-resistant properties. The self-adhesive form of this tape is ideal for waterproofing antenna traps, joints and connectors.

Nickel silver

Typical uses: Boxes, screens, nuts, bolts and washers etc where the non-corroding properties are required.

An expensive non-magnetic electrical conductor which is corrosion resistant. Available in sheet and rod. It is often used by railway modeller's for it is a very 'friendly' metal to use, soldering and bending easily. It has some resistivity.

Wood

Typical uses: Aerial masts, booms, spreaders, storage boxes, and other parts which do not require RFI protection.

A common material, not usually used today for radio work as it has no ability to conduct electricity or to screen against RFI. Its insulating properties are marginal unless dried and treated to prevent ingress of water/moisture. The radios of the 1920s and 1930s were cabinetmaker's delights but this was before the 'enlightened' days of electromagnetic compatibility. Other joinery-type materials such as chip board, MDF, plywood etc are equally unsuitable for RF applications but may be utilised to make mock-ups of proposed designs, especially for the panels and cabinets.

Corrosion

There are two main processes of metal corrosion. The first relies on environmental conditions such as rain or condensation which results in an acidic electrolytic liquid being formed on the surface of the corroding metal. As the metal corrodes, the acidity of the liquid increases until the electrolytic process of corrosion becomes almost self-sustaining. The second occurs due to the

1.	Magnesium alloys	8.	Lead
2.	Zinc	9.	Tin
3.	Aluminium alloys	10.	Nickel
4.	Cadmium	11.	Brass
5.	Mild steel	12.	Copper
6.	Stainless steel 18/8	13.	Silver
7.	Lead-tin solders	14.	Gold

Note: the numbers are item numbers only and are not values!

Table 25.1: Galvanic series

electrolytic action occurring between dissimilar metals in contact, and is referred to as galvanic action. Both processes change the metal into a different form, which in the case of steel or iron we know as 'rust', and often refer to the process as oxidisation.

Table 25.1 shows the galvanic relationship between metals. The numbers are item numbers only to show the position of each metal in the galvanic series. The actual values depend on several factors such as temperature, radiation and acidity etc. This list enables metals to be selected which will have the minimum galvanic corrosion effect on one another. The greater the list separation, the greater will be the possibility of corrosion.

For example, brass and copper are adjacent and would therefore not cause problems if in direct contact with each other. Brass and aluminium alloy are widely separated and corrosion occurs if these metals are in direct contact with each other. This state can be reduced by tinning the brass with lead-tin solder, which falls about halfway between the other two metals (galvanic interleaving). The higher item-numbered metal will normally promote the corrosion of the lower item-numbered one.

Dissimilar-metal galvanic action corrosion is avoided by ensuring that the galvanic series separation is minimal. If widely separated metals do need to contact each other, then a suitable interleaving material should be used to reduce the galvanic separation level.

Corrosion by moisture or rain is more difficult to combat effectively. The commonly accepted anti-corrosion treatment consists of protection by paint and, providing the paint coat remains intact or is renewed regularly, this is a very effective treatment. Steel and iron can also be coated by protective metals such as zinc (galvanising) or nickel (plating), both methods being normally outside the range of the home workshop. Aluminium is slightly different, for this metal forms its own protective oxide barrier which, providing it is not disturbed, will prevent further corrosion. It is this oxide barrier which makes soldering aluminium difficult. Unfortunately, this natural protective oxide layer can be disturbed by stress or galvanic action, and the corrosion process bites deeper into the metal. Anodising is a process on aluminium which forms a controlled layer of oxide on the surface and presents a toughened surface finish which can be coloured by dyeing. Brass and copper can be considered as corrosion resistant for most amateur purposes. However, their surfaces do oxidise and this can impair good electrical contact. Certain platings, such as silver, nickel or chromium, can reduce this. The platings themselves can also corrode but usually at a much slower rate than the parent metal.

If not adequately protected, corroding metal will gradually lose strength and the device from which it is made will fall apart. This is usually seen as collapsed masts, broken antennas and similar expensive disasters.

Corrosion also affects RF and electrical connectors, particularly feeder-to-antenna connections, and causes a gradual decline in the overall performance of the system. Signals become weaker and calls to DX stations which used to be answered are ignored. Most observed lowering of performance in this area is usually (or eventually), discovered to be due to corrosion. This can be practically eliminated by first ensuring that no dissimilar metals are in contact to cause galvanic corrosion, and second that water is excluded from all connections. Copper coaxial-cable inners should never be connected directly to the aluminium elements of a beam but should be tinned first as previously stated. Coaxial plugs and sockets should be fitted with heat-shrink sleeving and wrapped with amalgamating tape to prevent the ingress of water. Connector boxes can be filled with silicone-rubber compound for a like purpose. Stainless steel antenna fittings are the least affected by corrosion but even these would benefit from a coating of protective lacquer, particularly on screwed fittings. The position of stainless steel in the above list is interesting, for sometimes the metal in contact with the stainless steel will corrode due to galvanic action, particularly in wet, smoggy or salt-laden conditions, and some stainless steel fittings are supplied with plastic interleaving spacers and washers to prevent this.

Condensation is also a problem with outdoor enclosed or boxed-in items such as rotators, remote tuners, antenna traps, coaxial cable and the like - where there is condensation, there will be corrosion. The simplest, but not necessarily the easiest, solution is to allow the enclosed area to 'breathe' by introducing suitable weather-resistant holes as drainage vents, while ensuring that these are at the lowest point of the enclosed area and cannot be inlets for insects, rain or the run-off water from the outside of the enclosure. Complete sealing usually makes matters worse, for a fully water and air-tight enclosure still produces internal moisture by condensation. Full hermetic sealing is difficult to apply for it normally requires the ability to pressurise the enclosure with an inert gas, as with some military or maritime equipment. If the items within the enclosure can be protected as if they were exposed to the elements, much of the corrosive effect of condensation is reduced.

In the case of rotators, attempts at filling the voids with grease does not help, for the grease forms small pockets which can hold water. It is better to lacquer or wax oil the moving parts and the inside of the housing, and to use grease for its intended purpose of lubrication. Any electrical items within the rotator should be sprayed with a commercial, non-insulating, waterproofing liquid.

The threads, screws, nuts, bolts etc should be given a light coating of anti-seize compound on assembly and, after assembly, sprayed over with lacquer or wax oil, or coated with a water-repellent grease such as lanolin. This makes for easier maintenance and reduces the possibilities of corrosion.

Corrosion is not limited to outdoor items. Corrosion of connector pins of microphones, plug-in PCBs, computers etc is not uncommon. Careful selection of mating materials to avoid galvanic action, combined with appropriate painting, plating etc and regular maintenance, will reduce the effects of corrosion.

CONSTRUCTION TIPS

Before you start a construction project, you should have a definite plan of action. Changing your mind part way through the project can be expensive in terms of wasted time and materials. When building an enclosure for a project, you should measure the exact dimensions of any printed circuit boards or other types of circuits used and the height of the tallest components mounted on the board to make sure that the size and shape of the enclosure will be suitable for the job. Make allowance for any inter-stage screens or compartments within the main enclosure. You should also consider the possibility that you may want to introduce modifications or add new features at a later time.

ENCLOSURES

At one time, most amateur radio construction projects were enclosed in a home made aluminium box. Aluminium sheet was relatively cheap to buy and it made economic sense to build your own enclosures. In recent years, the price of aluminium sheet in small quantities has risen to the extent that it is almost as cheap to buy a ready-made box as it is to build one. Commercially made boxes are available in a wide range of shapes and sizes. Most of the smaller boxes are made from sheet aluminium. A few of the more expensive types are made from die-cast aluminium.

Die-cast boxes are stronger and more rigid than boxes made from aluminium sheet. The greater rigidity of die-cast boxes makes them very suitable for use as VFO enclosures or for any other project that requires good mechanical stability. Larger boxes are often made from a combination of aluminium and steel. The bottom half of the box which includes the front and back panels is made from aluminium and the top cover is made from vinyl coated or painted steel. The bottom half of the case is easily drilled or machined to make any holes or other openings that are required on the front and rear panels. The superior strength of the steel top cover makes this type of enclosure very strong and durable.

If your project calls for a size or shape of enclosure that is not commercially available or, like me, you just prefer to make your own, a home made box can easily be made from sheet metal. If you have a bending brake or a simple set of bending brackets, it is a very easy job to bend the aluminium sheet into the required shape. Even if you don't have any special metalworking tools, it is possible to make a respectable looking enclosure using only a hacksaw, a straight edge, a sharp carpet knife and a few very basic hand tools (pencil, measuring-tape etc).

The box shown in **Fig 25.3** was made as an enclosure for a VHF transverter. The top cover is a simple U shape made from a rectangular piece of aluminium sheet. The bottom half of the box is also made from the same 1mm aluminium sheet.

Before you start cutting and bending the aluminium, you should make a drawing of the box. The exact dimensions should be marked on the drawing. For more complex projects, I use a computer with drawing or CAD (computer aided design) software to create plans and templates for metal boxes. Suitable software is readily available for most personal computers. I use "Xfig" vector drawing software on a Linux PC. For more about using computer aided design, see the chapter on Computers in the Shack.

For very simple projects, a pencil and ruler is usually faster and easier. When you are drawing the plans, you should remember to account for the thickness of the sheet. The inside measurements

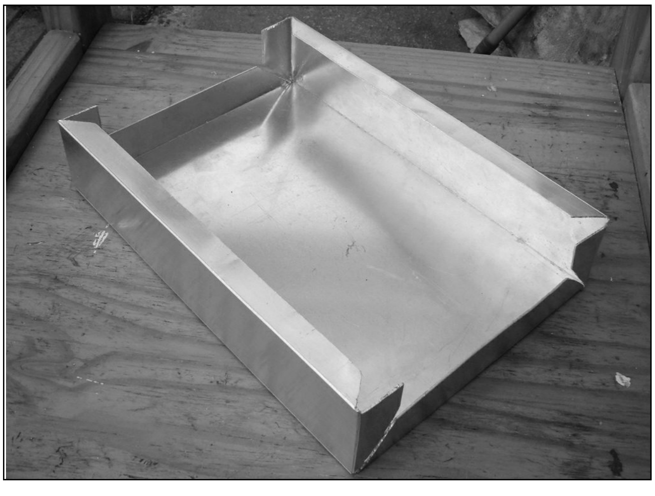


Fig 25.3: Home made aluminium box

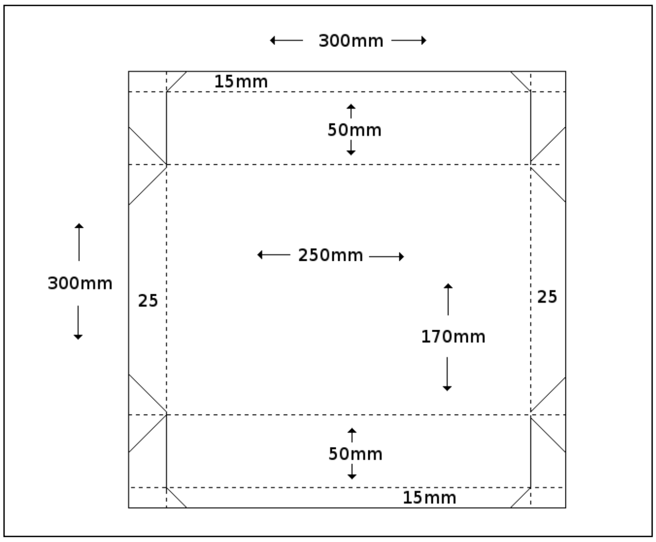


Fig 25.4: It is advisable to make a detailed drawing before making a box

of a box made from 1mm sheet will be at least 2mm less than the outside measurement. The bottom half of the simple project box was made from a 300mm x 300mm square of 1mm thick aluminium sheet. A detailed drawing is shown in **Fig 25.4**.

Since this box was made without the aid of any special tools, it was necessary to find a way of making perfectly straight 90° bends without using a bending brake. This was achieved by cutting a deep score along each line that was to be cut or bent. A very sharp carpet knife (Stanley or similar) was used to score the metal. Be very careful when you are cutting the score. Use a good quality metal-bodied knife and not a cheap plastic one which might break under pressure. A steel engineers rule can be use as a guide for the knife blade.

The 300mm square was cut from a larger sheet of aluminium. As this was literally a 'kitchen table' project, a sheet metal cutting guillotine was not available. It is very difficult to cut a perfectly straight edge with a hacksaw and even more difficult with an electric jigsaw.

The 300mm square was carefully marked out at one corner of the sheet with a felt-tip pen. After double checking my measurements, the carpet knife was used to cut a deep score along lines that were to be cut. The sheet was then carefully bent over a straight edge. Our kitchen table was not suitable for this job because it has a bevelled edge. I used a surplus sheet of veneered panel that was left over from a DiY project as a suitably square edge and to protect the kitchen table or counter-top from damage. A thick sheet of steel or the edge of a metal workbench can also be used. You will find that it is quite easy to bend the sheet along the line of the knife score. With a little practice, you will get a perfectly straight bend or cut first time, every time.

When you want to cut rather than just bend the sheet, you should bend it to 90° and then partially straighten it again. Move the bent section back and forward a little more each time until it is so weak that it breaks off cleanly. The type of soft aluminium sheet that is sold by hardware and DIY stores is quite flexible and it probably won't break until it has been bent to almost 180° several times. Be patient at this stage. Don't apply too much force or you will end up with a slight curve at the edge of the sheet.

Once the sheet has been cut out, you should mark the bends and cuts with a felt-tip pen. A fine laundry marker or a CD/DVD marker pen is ideal for this job. The bends should then be scored with the carpet knife. Since you only want to bend the metal rather that cut it, you should only cut the score deep

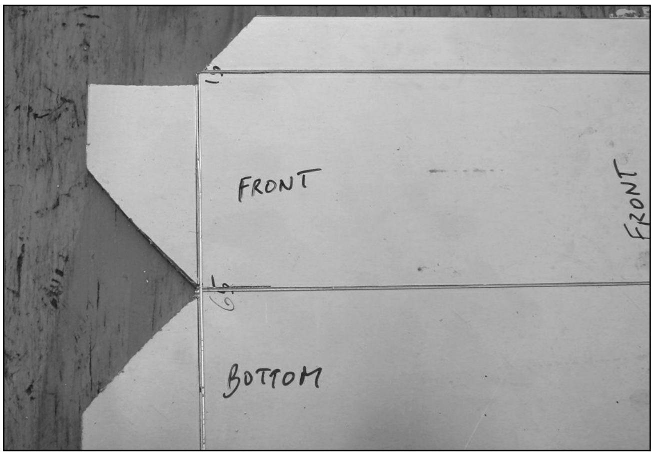


Fig 25.5: The cut sheet ready for bending

Fig 25.6: SWR meter enclosure using the score and bend method

enough so that the metal bends cleanly and easily. If the score it too deep, it will weaken the structure of the box or in the worst case, it might break off altogether! At this stage, you should cut off any unwanted parts at the edge of the sheet. You will usually need to cut out a V shape where there will be a 90° bend. **Fig 25.5** shows the sheet when it is ready for bending.

The bottom half of the box should now be bent into its final shape. This job is not as easy as the first cutting/bending operation because we are now dealing with a three-dimensional structure. The task becomes progressively more difficult as each part of the box is bent into shape. A small bench vice and a few pieces of timber will make this task a lot easier.

The 15mm lip at the very edge of the sheet is particularly tricky. Since this lip will not be visible when the lid is on the box, a small amount of damage in this area can be tolerated. A vice grip or large pliers can be used to tidy up the edges, provided that you take care not to damage the front panel. You can use a fine file or a medium grade of sandpaper to remove any sharp edges. This is not critical for the bottom half of the case because all of the edges will be inside.

The top half of the case is very easy to make. A simple U shaped cover can be made from a rectangular sheet of aluminium or other metal. Only two bends are required.

The most critical part of the project is the front edge of the top half of the box. This is the part of the box that will be most visible to the user. Take great care when you are cutting this edge, or preferably cut this panel from the edge of the original sheet of aluminium so that the guillotine-cut edge is towards the front of the enclosure. The aluminium top cover can be replaced by a steel cover. Steel is much harder to cut than sheet aluminium, but it is more durable and easier to paint. Fancy perforated steel is easier to bend than plain steel sheet. The perforated sheet will allow some air-flow through the enclosure which will help to keep high power circuits cool. This type of construction is often used for power supplies and RF power amplifiers.

The score and bend method of construction has served me well over the years, I have used this method to build cases for many projects of all different sizes. **Fig 25.6** shows a SWR bridge which was built using score and bend.

There are many other ways of making metal cases and each individual builder will have their own favourite method. One approach is to make every panel of the box from separate pieces of sheet metal. The panels are screwed together at the corners. Angle aluminium or angle steel can be used as corner supports. The panels can be drilled and secured to the angle pieces with nuts and bolts. As an alternative to the strip of angled metal, a solid length of square metal bar can be drilled and tapped and used as the corner pieces. Square plastic, fibreglass or even wood can be used for the corners, but metal is to be preferred for screened enclosures.

Painting the Enclosure

Most of my enclosures are painted with aerosol spray paint. Aluminium is notoriously difficult to paint. It helps if the metal is absolutely clean before you apply the primer coat. Special etching primers are available which are used for painting motor caravans or aircraft. You can get away with using ordinary car primer, provided you apply it as several light coats. After it has dried thoroughly, the primer should be lightly rubbed down with very fine wet sandpaper. Be careful, the fresh primer will rub off the aluminium panels very easily.

You may need to apply several coats to fill in any scratches in the metal. Larger scratches or pinholes in the surface of the primer can be covered with a filler/stopper compound before sanding. Once you have a clean and smooth coat of primer, you can paint the enclosure with an aerosol enamel. Aerosol paints use a lot of paint thinner and they tend to run very easily. Do a few tests on a scrap sheet before you tackle the real project. Remember that aerosol paints use thinners and propellants that are highly flammable. Keep well away from sparks and flames. Wear a mask and work in a ventilated area.

Letraset or similar rub-on lettering can be used to label the controls on your equipment. Several coats of clear lacquer should be applied over the finished panel to protect the letters from damage. The first coat of lacquer should be applied very lightly so that it will dry quickly. If you use a heavy coat, the thinners used in the aerosol will tend to melt the plastic lettering. **Fig 25.7** shows the front panel of a home made SSB/CW transceiver. For more information, see the finishing and painting sections near the end of this chapter.

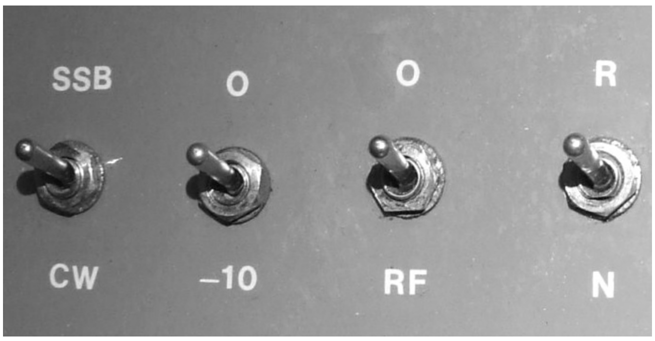


Fig 25.7: Smart lettering on a front panel

Two Useful Home Made Tools

One of the problems most of us have in metalwork and the like is cutting or filing things square and straight. Also, marking out to ensure that parts will fit together correctly is sometimes not as easy as it appears. The few extras described here should simplify things.

Bending bars

See **Fig 25.8**. These are normally home-made. The things to check when obtaining the steel angle are its straightness and squareness. If these are not 'true' the corrective actions needed will require skills which may be beyond those so far acquired. Old bed-frame angles, provided they have not rusted too badly, usually make very good bending bars. One bed frame can be used to make several bending bars of different lengths. The choice of length and distance between the clamping bolts of the bars is related to the maximum width of metal or other sheet purchased and the maximum size of panel worked. Aluminium sheets, for example, usually come in 1.8 x 1.2m size and trying to cut this is rather difficult. It would be better, if buying in these sort of quantities, to have the sheets guillotined by the supplier to widths which will fit between the bending bar clamp bolts. (Suggest 482mm.) The length of the sheet will not matter.

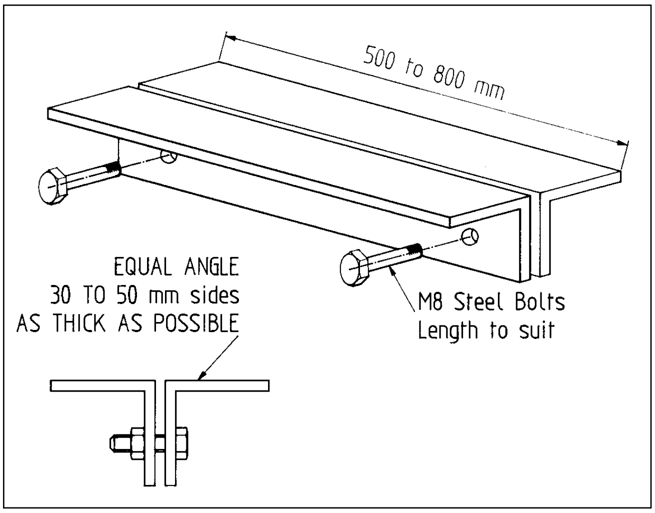


Fig 25.8 Bending bars

Hole embossing tool

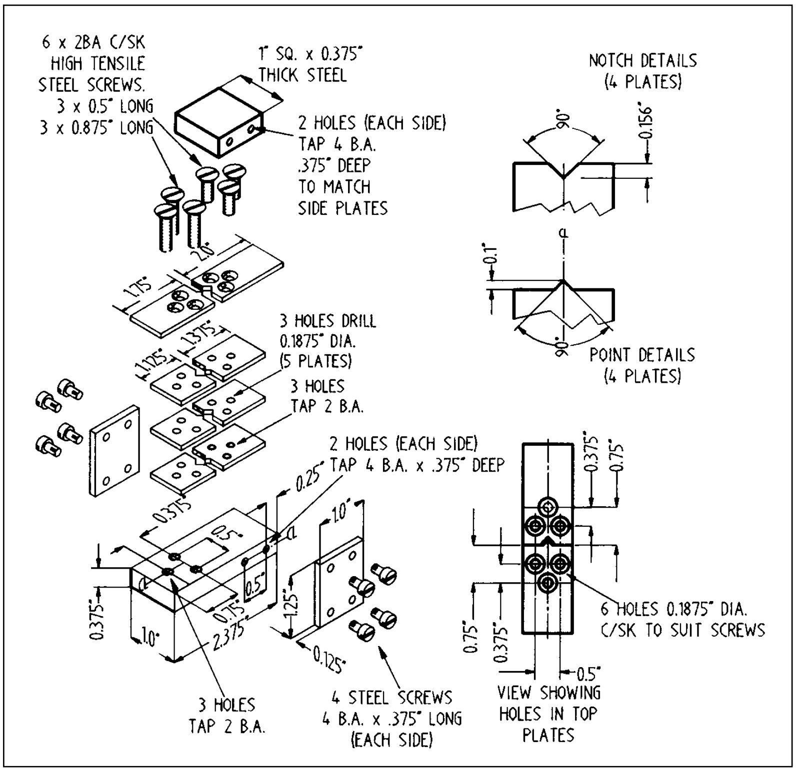
See **Fig 25.9 and 25.10**. This very simple home-made tool forms holes along the edges of metal panels to accept self-tapping screws of about 2.5mm diameter (depending on metal thickness), and with an engagement length of about 12mm. It facilitates the rapid manufacture of cabinets, boxes and screens, eliminating much of the accurate metal bending normally associated with this type of work.

The working parts and the two side plates of the tool are made from 1in wide by 0.125in thick-ground gauge plate, which is a tough but workable steel capable of being hardened and tempered. It is not necessary to harden the finished tool, unless it is intended to form mild steel but, if facilities are available to do this, it will improve the durability. The base and top plates are made from 1in wide by 0.375in thick mild steel.

The vee points must be on the centre-line of the tool. The vee notches are not so critical for they play no part in the shape of the hole produced, but they should allow sufficient clearance for the formed metal to flow into. The point and notch sizes shown are suitable for most metal thicknesses up to 1.6mm (16SWG). The vee points can be made as a set, with the four plates clamped together. The fully shaped points should be polished with an oilstone to obtain square, burr-free, sharp edges along each face of the point. The edges of each point are the working parts of the tool and should be made with care. It is essential that the flat faces on both sides of the notch and point ends of each set of plates are square, parallel and level with each other.

Tapping-size holes should be marked out and drilled on each top plate only, and these are then used as a template to drill the remaining holes on each plate. The holes in each bottom plate should be tapped 2BA and the holes in the remaining plates opened up to 0.1875in diameter - a good fit for the high tensile steel 2BA screws. ISO M5 may be used in place of the 2BA screws and tapped holes. The outside faces of the holes in the top plates should be countersunk deep enough to ensure that the screw heads are slightly under-flush. It is good practice to lightly countersink both sides of every hole to remove burrs and facilitate assembly.

The back edges should also be square and level to ensure that the vice pressure is applied evenly during forming. The 0.125in thick side plates and the 0.375in thick top and bottom plates form the bearing for the sliding part of the tool, which must slide easily with the minimum of play in any direction. Failure to get this right will ruin the action of



25.9: Hole embossing tool

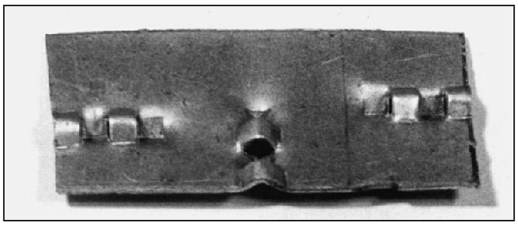


Fig 25.10: Holes made by the embossing tool

the tool, for each point must just slide over the opposing one or two points with the minimum of clearance.

To operate, the tool is placed between the jaws of a 4in vice and supported by the extended top plates. The metal to be holed is rested on the base of the tool and the marked out hole position is aligned with the tip of the visible point. The vice is then closed with the minimum of force until each face of the working ends of the tool just contacts the metal sheet. It is pointless trying to go beyond this, for squeezing the tool and the work by excessive pressure from the vice will only ruin the tool and the work. The vice is then opened and the metal sheet gently prised away from the points. This method of releasing the formed sheet could be improved upon, but the tool would become more complex to make and, as a spot of oil on the working faces eases the problem, this extra complexity is not worthwhile.

The tool performs very well on aluminium, tin plate and annealed brass or copper in thicknesses up to 0.0625in (16SWG). It is not recommended for use on sheet steel unless the points have been hardened and tempered.

Hole Making

Drilling is a straightforward operation providing a few key guidelines are observed. The drill speed should be adjusted to match the size of drill and the material being drilled. As a general rule, the smaller the drill, the higher the speed. A correctly sharpened drill should not require heavy pressure to cut. If it does, either the material is too hard or the drill needs to run slower. There is a tendency for normally sharpened drills not to cut some brass properly. This is due to the incorrect rake angle of the cutting edges of the drill. The problem can be solved by sharpening the helix edge on the face of the flute to give a rake angle of about 15 degrees, ie leaning backwards away from the normal cutting edge (**Fig 25.11**). Drilling some plastics can also present problems, such as chipping of the edges and breakaway of the material as the drill breaks through. Some improvement is possible by sharpening the drill to produce an included point angle of around 80 degrees. The standard included point angle is 118 degrees.

When drilling steel, it is advisable to use a coolant such as soluble cutting oil to keep the drill and the work cool. It also saves having to re-sharpen the drill so often. Paraffin is a good coolant for aluminium and copper. Brass and most plastics do not normally need a coolant, providing the drilling speed is correct, but treat as steel if necessary. These coolants may be applied either by an oilcan, or a brush. (Old liquid soap squeeze bottles make good coolant dispensers and any left-in detergent is not detrimental.) Soapy water is also a good coolant and certainly better than no coolant at all - it is best for most plastics.

Holes should always be centre-popped before drilling and, if a BS1 centre drill is available, this should be used next to provide an accurate location for the drill.

The holes drilled in thin sheet are often anything but round because the drill does not have enough depth of metal to round the hole properly before it breaks through. There are at least two ways round this. The easiest is to drill the hole undersize and bring to size using the taper reamer, which must be allowed to cut without forcing, otherwise another fancy-shaped hole will be produced (another reason for scribing-in the holes beforehand.) Another way is to alter the cutting angle of the drill to an included angle of about 140 degrees and thin the chisel edge of the drill to a point. It is worthwhile keeping a set of drills sharpened in this manner especially for thin sheet drilling. These sheet metal drills normally require a much slower speed and the holes should also be pre-drilled using the centre drill.

Whatever is being drilled should be well supported and clamped to prevent rotation and lifting. Failure in this direction can lead at best to a broken drill and at worst to serious personal injury, for drills have a habit of picking-up just as they are about to break through the hole. (Plastic and copper are particularly susceptible to this.) A panel whirling round on the end of a 10mm diameter drill is a frightening sight!

All drilling will produce burrs around the hole edges and it is good engineering practice to remove these using the hand-held countersink bit referred to previously. A file will only scratch the rest of the surrounding surface and bend the burrs into the hole. The use of a large drill for hole de-burring is not recommended, particularly on the softer metals, unless the 'touch' for this method has been acquired.

It is usual to step drill holes larger than 10mm diameter, that is a hole of smaller diameter (3mm) is drilled first, then another slightly larger (+1mm), and so on until the finished size is reached. Step drilling is unsuitable for sheet material. There are special, though expensive, stepped drills available for drilling holes up to 40mm diameter in sheet material.

Making large round holes can be tackled in at least two ways. The first requires a washer or tank cutter (**Fig 25.12**) and an extra slow speed drill (a joiners' brace is effective). The biggest snag with this method is trying to obtain an even cut around the full circle. By clamping the work to a block of wood and drilling through into the wood for the centre pilot of the tank cutter, a guide is provided which improves things a little. The main thing with this method is not to be in a hurry.

The second method can be applied equally well to non-round holes. Contiguous holes of about 5mm diameter are drilled on the waste side of the hole or cut-out and 1 to 2mm away from the finished size markings. The waste is removed, using tin snips. The hole is then carefully filed to size, using the bending bars for support, and as a guide for any straight portions.

To de-burr large holes, a small half-round needle file can be used in the 'draw' fashion. The file is held at both ends and drawn round the edges to be de-burred in a manner similar to

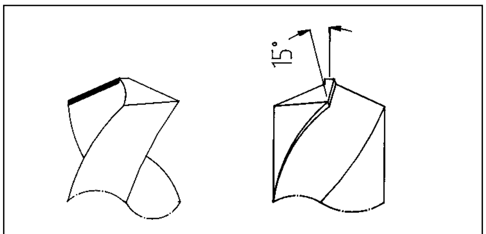


Fig 25.11: Modified drill rake angle to improve the drilling of brass

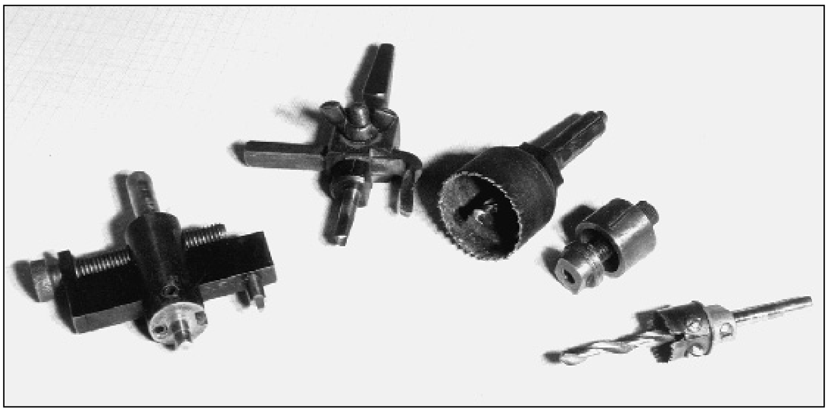


Fig 25.12: Large round hole making tools. Clockwise: deluxe home-made tank cutter, commercial tank cutter, screw-up punch, hole saw

using a spoke shave. The file should be held at an angle to produce a small 45° chamfer around each edge of the hole.

Machine countersinking of holes requires a very slow speed drill (60RPM). They can be produced using the handle-mounted countersink bit referred to previously. Even the multi-toothed rose-countersink bit will chatter and leave a very unsightly surface if too high a speed is used. Countersunk screws will not sit properly on such holes. There are countersink file burrs available which do a similar job and can be used in the DIY drill, providing the work is held securely with the burr square to the work. They do have a tendency to skid over the surface, leaving a trail of deep scratches, if not located properly.

Drilling accurately positioned holes can be a problem and templates can be used with success. Keeping the small holes for multi-pin DIL ICs in-line and at the correct spacing is difficult, but the job is made easy by using a piece of the correct pitch matrix board as a drilling template. One of the holes (pin 1 is suggested) is drilled first in the required position. The template is located from this by passing a drill through the hole and the matrix board.

The matrix board is then aligned with the rest of the marked-out hole positions and clamped using the toolmakers' clamps. The remaining holes are drilled through using the matrix board as a template. It helps to mark each row of holes on the matrix board template. If a considerable amount of such accurate drilling is to be done, it is worth making a set of metal templates for each size of IC using this method. Sometimes the component itself can be used as a template to reduce the risk of error, eg slide switches, coaxial sockets, dial fixing holes etc.

Filing

Accurate filing is a skill which can only be acquired by practice. General rules are:

- Always use a file handle. This eliminates the risk of running a file tang into the wrist and enables the file to be guided properly.
- Use a sharp file. It is normal practice to use new files for brass and, as their sharpness wears off, use them for filing steel or aluminium. New files should be kept separately or otherwise identified.
- Do not force the file to cut. Only a light relaxed pressure is required, which also aids the accuracy of filing.
- Keep the file clean by brushing with a file card or by rubbing a piece of soft brass or copper along the teeth grooves.
- Support the work properly. Trying to file with the work held in one hand and the file in the other is a guarantee of failure.

The height of the vice should be such that when the filer's arm is bent to place the fisted hand under the chin, the point of the elbow should just rest on the top of the vice jaws. An old but effective form of ergonomics!

A strip of emery cloth wrapped along the file can be used to obtain a better surface finish. This method was frowned upon in apprentice schools but it works and saves time. The draw filing technique referred to previously imparts the final touches to the filed edges and removes the filing burrs. The bending bars can be used as a filing guide (Fig 25.13) and a set of bars can be made especially for this purpose, checking them regularly to see that the guiding surfaces are not bowed by too much one-spot filing.

Bending and Forming

The first essential is to ensure that the material can be bent. This sounds obvious but it is better to check first than to find out after all but the bending work has been done. Annealing can be applied as mentioned previously under the section on materials.

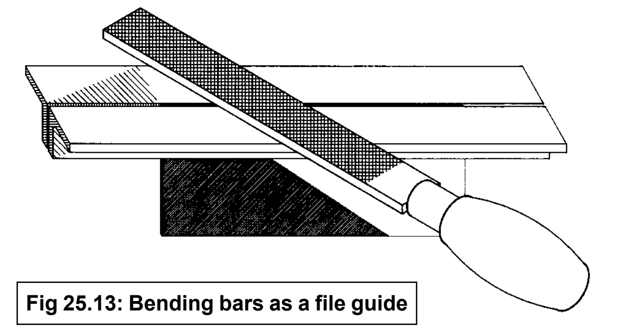


Fig 25.13: Bending bars as a file guide

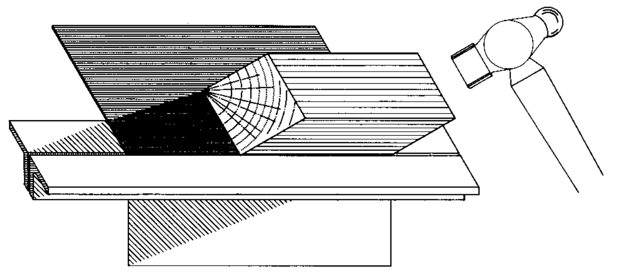


Fig 25.14: Method of bending in the bending bars. Note that the cutter block is held into the point of the bend

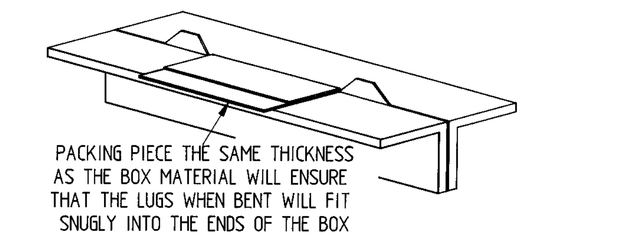


Fig 25.15: Making allowance for bending the end lugs of the box

A metal hammer should never be used directly on the metal when bending or forming. Either use a soft-faced hammer or a block of wood or plastic as a buffer for the hammer blows (Fig 25.14). This prevents all the humps and hollows which would otherwise occur. Do not use any metal as a buffer block as there is a danger of the metal chipping and flying into the face or eyes. The block should be kept into and near the point of bending. It seems easier and quicker to try to use the sheet as a lever and hammer as far away from the point of bending as possible. This will only produce a bend which curves up and back again. Where three sides are to be formed by bending, the point of intersection of the marked-out bend lines should be drilled before bending, with a hole diameter three times the thickness of the metal. This prevents corner bulge. Fig 25.15 shows how to ensure that the end lugs of the box fit snugly inside the box. Fig 25.16 shows one method of bending the ends of a similar box.

Tube bending for beams, tuned lines etc is not difficult and flattening or kinking can be reduced by observing the following:

- Ensure that the tube is suitable for bending. Anneal as necessary and re-anneal as soon as resistance to bending is felt.
- Unless skilled or equipped with specialised tooling, do not attempt to bend to a radius of less than three times the outside diameter of the tube, eg 12mm OD tube should have a 36mm minimum bend radius.
- Always bend round a former shaped to the required radius (Fig 25.17).

- Pack the tube tightly with fine sand (clean birdcage or builders' sand is ideal). Wet the sand and cork both ends. This will minimise the risk of kinking during bending, and the sand can be washed out afterwards. The tube can be re-annealed if necessary with the sand left in place but the corks should be removed to let the hot gases escape.

A low-melting-point, lead-like material is available which can be used in place of sand, and is known as bending metal. It is poured molten into the blocked, ended tube, allowed to set and, when the bending has been completed, is melted out to be used again. This process should be carried out with the tube at an angle to allow the hot air to escape. The molten bending metal should be poured very slowly for it has a tendency to 'blow-back'. Normal lead is not a substitute for bending metal and should not be used, particularly with aluminium tubes, for the melting temperature required for lead may destroy the aluminium's properties.

Modelling wax (also known as American wax) can be used in place of sand or the bending metal. The wax is heated, not boiled, until molten and then poured into the tube in the same manner as the bending metal. The wax should be allowed to set hard before bending. Unfortunately it is not possible to re-anneal with this wax left in place.

Another tube-bending tool is the bending spring. This is used by plumbers and pipe fitters. Each tube size should have a matching bending spring, for it is all too easy to use the wrongly matched spring and have it permanently trapped inside the bent tube!

Cutting

Cutting long strips of metal with tin snips or shears is an expert's job. The cut edges usually produced by non-experts are anything but straight and they require flattening to remove the cutting curl. Tin snips are best used where a one-snip cut will remove the required amount of metal, such as 45-degree corners, or the trimming to length of narrow strips. Snips should not be used on plastics, laminates or copper-clad board (PCBs), for the cutting action can cause de-lamination or shattering of the edges.

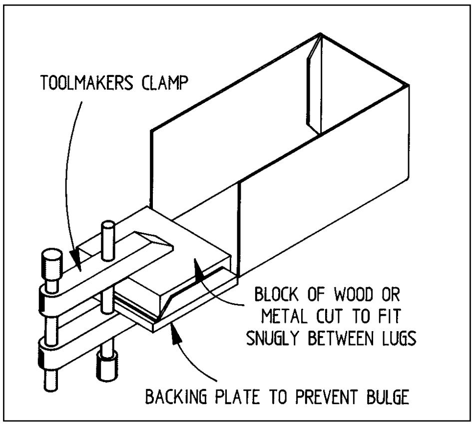


Fig 25.16: Bending the ends of the box using a toolmaker's clamp to secure the forming block

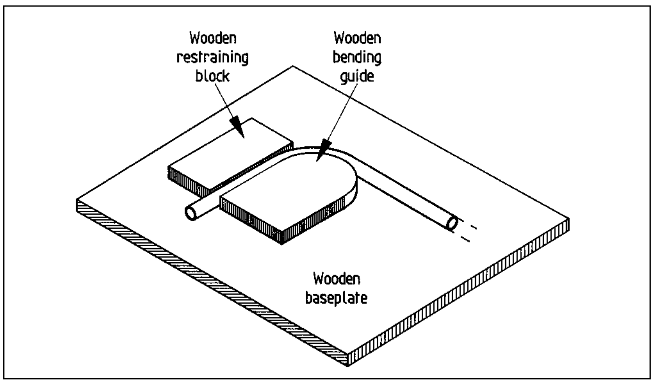


Fig 25.17: Tube or rod bending set-up

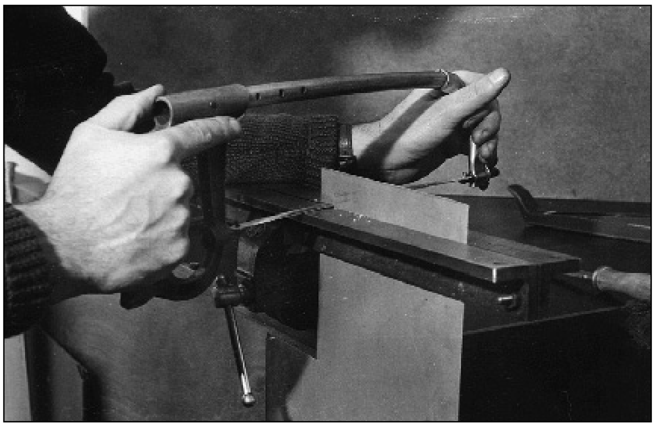


Fig 25.18: Bending bars as a sawing guide

A guillotine is by far the quickest way of cutting most sheet material but it is not a normal home-workshop tool. For the home workshop the cutting of nearly all materials is best done by sawing, using a hacksaw and the bending bars as a guide and support (Fig 25.18). The hacksaw should be fitted with a 24 to 32 teeth per inch blade for most sheet work, and coarser blades for cutting blocks of material. The blades known as ding-dong style, ie with their cutting teeth arranged along a wavy edge, are very good for most work. When sawing copper-clad laminates such as PCBs, it is advisable to score through the copper beforehand using a sharp chisel. This will bevel the edge of the copper and prevent delamination due to sawing. The coolants recommended for drilling can be used if required. A fine-toothed roofers' saw is very useful to cut sheets which are too big to pass inside the hacksaw frame. This is similar to a joiners' saw but shorter with a stiffer fine-toothed blade. A pad saw or fine-toothed machine hacksaw blade which is handle mounted is equally effective in these circumstances. Some constructors use a fretsaw with either a metal-cutting blade or an Abrafiler to overcome the problem, particularly when making large holes.

Hand-operated nibblers are available for cutting most thin sheet materials, including laminates, although they are sometimes difficult to guide properly. The power-operated nibblers are not normally a home-workshop tool. The DiY jig-saw fitted with a fine-toothed, metal-cutting blade is very good but it is essential to support the work very well and even then it is a noisy process. Similarly the DiY router-cutter is extremely useful, especially for large holes, but it can be a dangerous tool to use.

Cutting and sawing are areas where gadgets are forever appearing, each claiming to save time and produce a better job! The hacksaw has not yet been replaced as a good general-purpose cutting tool.

SOLDERING

Whether they are made in the radio amateur's workshop or commercially by a large electronics company, almost all electronic circuits will use soldered electrical connections. The type of soldering that is usually used for assembling electronic circuits is known as soft soldering.

To make a solder joint between two metal conductors, a metal alloy (the solder) which has a relatively low melting point is applied between the two conductors and heat is applied to the joint. The heat causes the solder to melt and flow around the joint. When the joint is allowed to cool, the solder makes a good electrical and mechanical joint between the two conductors.

For a typical hand assembled circuit, the heat is provided by a hand-held soldering iron. The main difference between soldering and welding is the much lower temperatures used. A solder joint is formed at a temperature which is well below the melting point of the metals to be joined. The term soft soldering usually refers to joints that are formed at a temperature of less than 450°C. Higher temperature soldering methods like hard or silver soldering are rarely used for electronic assembly work because the much higher temperatures involved would damage the PCB and electronic components. Until recently, most solders used for electronic work were made from a tin/lead alloy, the most common type used is a mix of about 60% tin and 40% lead. The introduction of the new RoHS (reduction of hazardous substances) directives in Europe have lead to a gradual phasing out of lead solder over the last few years. From 2006, electronic circuits are required to be lead-free. These new regulations have led to the development of lead-free solders with a high percentage of tin and a small amount of copper and/or silver. Lead-free solders generally have a higher melting temperature than tin/lead solder. The 99.3% tin, 0.7% copper alloy that is often used to replace lead solders has a melting point of 227°C and a recommended soldering iron tip temperature of 350°C.

The use of a soldering flux prevents oxidation of the metals to be joined and helps to 'wet' the solder so that it flows freely and makes a better joint. Tins of solder flux are available from most component suppliers, but most solders that are designed for electronics work have a built-in flux core so that no additional flux is required. Traditional solder fluxes are made from rosin which is produced from a naturally occurring resin found in pine trees and some other plants. Some types of flux are very acidic and are not suitable for electronics work. Most types of flux will leave some residue around the soldered joint. This flux is usually removed from mass produced circuit boards by washing them in a solvent.

New regulations that are intended to reduce the amount of VOCs (volatile organic compounds) released into the environment have also led to changes in the formulation of soldering fluxes. Water soluble fluxes can be removed without the use of environmentally harmful solvents.

Rules for Soldering

- All surfaces must be scrupulously clean. This applies equally well to the bit of the soldering iron. These bits are often iron plated to prevent rotting, and this plating should not be removed but wiped clean using a damp cloth or sponge. The bit should be kept 'tinned' with the solder and flux in use. Note that some fluxes will rapidly corrode the iron plated soldering bits. In the case of PCBs, the copper cladding should be cleaned using a fine abrasive eraser block. Component wires and pins etc should be similarly cleaned for, although many brand-new parts come with the wires dipped in a solder-through or flux coating, it is still worthwhile just making certain. There must be no dirty or greasy patches on any of the surfaces to be soldered and corrosion spots must be removed. Even finger marks can spoil the soldering, so "clean it and when you think it's clean, clean it again!"
- Ensure that there is just sufficient heat to make the joint. Too much heat is harmful to electronic work and, in the case of structural work, it causes distortion. Not enough and the solder will not bond correctly.
- Heat and apply solder to the greater conductor of heat first, which in the case of PCBs is usually the copper pad and in the case of switches etc is the solder tag, and then finally the wire, (the lesser conductor of heat) in one continuous process.

Soldering Technique

After cleaning the parts, the following technique should be used to make a sound mechanical and electrical joint on PCBs.

Apply the clean, tinned hot iron to the pad adjacent to the component wire to be soldered, and tin the copper surface by applying just sufficient multi-cored solder to the point where the tip of the iron is in contact with the copper. When the solder and flux starts to wet the copper surface, slide the iron into contact with the wire and apply a little more solder. As soon as both the wire and the copper surface are blended together in a small pool of molten solder, draw the iron up the wire away from the work.

This whole process should take seconds, not minutes. The less time any component is under this form of heat stress the better. In the case of particularly heat-sensitive components, it is advisable to heatsink the component's leads during soldering. A good soldering heatsink can be made from a crocodile clip, on to each jaw of which is soldered a small strip of copper. A small set of metal forceps will also serve as a clamp-on heatsink. The clamp-on action is required so that both hands are left free to do the soldering. Commercial soldering heatsink clips are available.

This method of soldering applies equally well to the joining of wires to components such as switches, potentiometers etc. In this instance, good practice demands that the wires should be wrapped and secured mechanically before soldering, but bear in mind that the joint may need to be unsoldered later.

Good soft-soldered joints should look neat and smooth, with no draw-off points, and be continuous around or along the joint. Large blobs of solder do not ensure that a joint is well-made. The cored solder should always be applied to the work and the iron at the same time, not carried on the iron to the work.

The results of incorrect soldering procedures are 'dry joints' which are either open-circuit or high-resistance points and will prevent the circuit from functioning correctly. Often the effects of such soldering will appear after several hours of operation, and this really leads to frustration especially to the novice constructor. Commercial equipment has exactly the same problems, if this is any consolation! In most instances it is difficult to locate a dry joint visually. Sometimes a perfectly sound looking joint turns out to be 'dry' immediately at the contact point, which is hidden under the solder. As a rough guide most 'good' joints appear shiny and devoid of any 'craters' or occlusions, whereas some 'dry' joints appear dull and pitted. There is no easy way to identify 'dry' joints but there is an easy way of avoiding them - follow the rules and guidelines set out above.

To join structural parts together by soft soldering, each mating face should be pre-tinned, ie given a thin coat of solder using the appropriate flux to ensure proper 'wetting'. The tinned faces must be clean and shiny with no burnt or blackened flux spots.

The two tinned faces are then painted with flux, brought together, heated to just melt the solder, and clamped together until the solder has cooled. There is no need to apply any more solder. If the job has been done properly, a neat, clean solder fillet will be seen along the mating edges. The flux residue must be washed away immediately after soldering to prevent long-term corrosion. It is not good practice to have long soldered joints on PCB laminates. The expansion caused by the necessary heat will de-laminate the copper. In these circumstances it is better to tack solder along the required structural joint, placing 2mm long solder-tacks at about 10mm intervals. Any such joints on copper laminates will be as mechanically strong as the laminate-to-copper bond.

It is better practice to secure mechanical parts such as angle brackets, shaft housings etc by screws or clips which pass right through the laminate, and to use solder only to make a sound electrical connection.

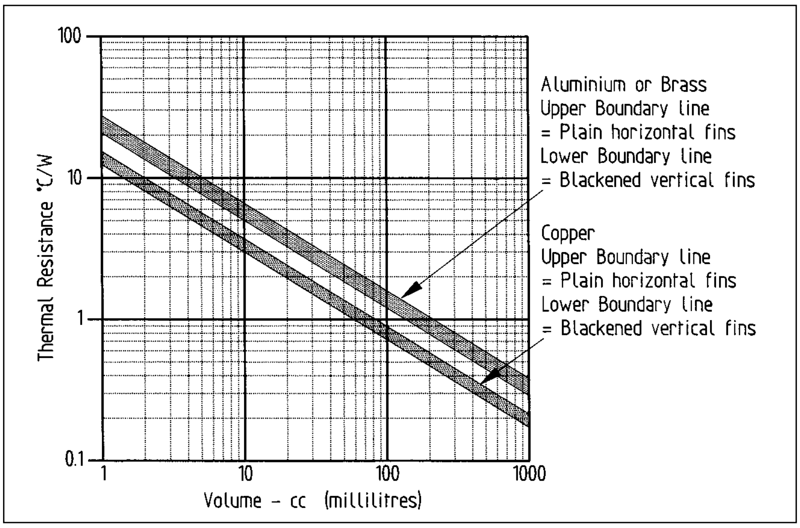


Fig 25.19: Heatsink thermal resistance / size chart

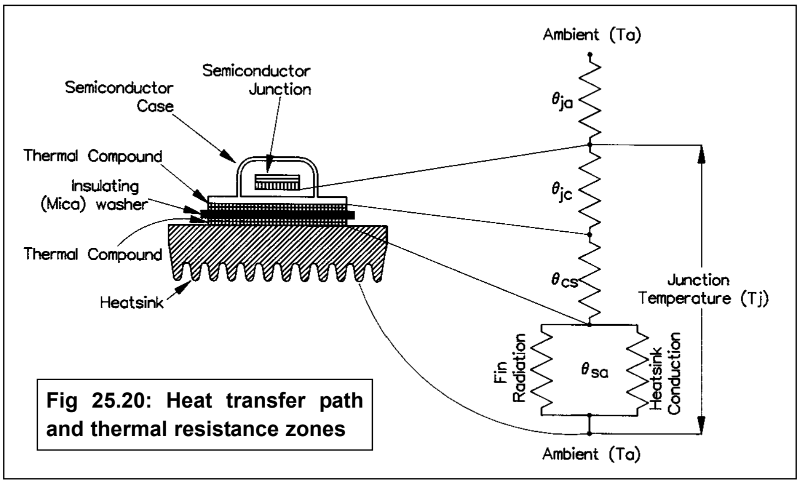


Fig 25.20: Heat transfer path and thermal resistance zones

HEATSINKS

Heat levels for semiconductors are usually defined as 'not to be exceeded' junction temperatures. The power dissipation at these junction temperatures is specified in watts, and can range from milliwatts to several hundred watts. Whenever the device is operating, heat is being generated. Unless some means of heat dissipation is used, the recommended operational limits will be exceeded very rapidly and the device will fail. A semiconductor junction takes microseconds to reach its operating temperature and only a few more microseconds to destroy itself if heat is not dissipated. Electronic protection circuits can help safeguard the device but will not eliminate the requirement for heat dissipation. Semiconductor devices generate heat very rapidly and it is essential to ensure proper cooling.

The heat must be dissipated at the same rate as it is being generated in order to ensure that the device remains within its specified temperature operating limits. The usual system consists of a heat-conducting material which takes the heat from the device and transfers it to the atmosphere. The materials used for heatsinks should have high thermal conductivity to conduct the heat away from the heat source, and high emissivity to radiate this heat to the atmosphere. Emissivity depends on the surface finish and texture of the radiating elements, rather than the type of material, but thermal conductivity depends on the type of material. Most commercial heatsinks are made from aluminium, for it is lighter in weight, cheaper and can be extruded or cast into the complex shapes required for compact, efficient heatsinks. Aluminium can also be anodised and dyed black to increase its emissivity. Heatsinks should be made and positioned such that the fins radiate to atmosphere. Fan cooling enables heatsinks to be smaller for the same thermal resistance, and as a guide, a substantial airflow over the fins can reduce the existing thermal resistance by up to 40%, depending on the number of fins receiving air.

Painting a natural metallic finished heatsink with a very thin coating of matt black paint will also produce a reduction in overall thermal resistance by increasing the emissivity. The actual amount of reduction depends on the mattness of the black paint and the thickness of the coating but a 10 to 25% improvement could be expected.

Fig 25.19 shows the thermal resistance possible at various volumes for different metals, finishes and fin positioning. The bands indicate the effects of a matt black finish and the vertical or horizontal positioning of the fins. The top edge of each band represents the thermal resistance of natural-finished horizontal fins. The bottom edge represents matt-black-finished vertical fins. The band for aluminium or brass is an approximation, for the thermal conductivity of either metal varies widely depending on the alloying metals, but it can be considered reasonably accurate for most amateur purposes.

The heat transfer path is considered as flowing from the semiconductor (heat source) to ambient air, with each junction or transitional point treated as a thermal resistance (Fig 25.20). The thermal resistances of fins and heatsink to ambient are considered in parallel to give the total heatsink value. Similarly the thermal resistance of the device junction to air is considered to be in parallel with the total thermal resistance.

Heatsink formulas:

$$P_D = \frac{T_j - T_a}{\theta_{jc} + \theta_{cs} + \theta_{sa}} \qquad (1)$$

and

Total thermal resistance x Power dissipation = Temperature rise in °C above ambient of the transistor junction (2)

from which:

$$\theta_{sa} = \frac{T_j - T_a}{P_D} - (\theta_{jc} + \theta_{cs}) \qquad (3)$$

also:

$$T_j - T_a = P_D \times \theta_{ja} \qquad (4)$$

where:

T_j = Maximum allowable junction temperature (°C)
T_a = Ambient temperature (°C)
θ_{ja} = Thermal resistance, junction to air (°C/W)
θ_{jc} = Thermal resistance, junction to case (°C/W)
θ_{cs} = Thermal resistance, case to heatsink, plus any insulating washer and heat-conducting compound (°C/W). (Can be assumed to be between 0.05 and 0.2°C/W).
θ_{sa} = Thermal resistance, heatsink to ambient air (°C/W).
P_D = Power dissipation (W)

Readily available semiconductor data do not always specify all of the above details but approximations may be used to design a heatsink suitable for most amateur applications. The usual data includes P_{tot} in watts (P_D) at a case temperature of usually 25°C. This can be used to derive the junction temperature (T_j) for substitution in formula (1). Because θ_{ja} = 25°C/P_{tot} and T_j = 25°C/θ_{ja} then T_j = P_{tot}°C.

Example: A 2N3055 is to be used as the pass transistor in a 13.8V DC regulated PSU. The available data shows P_{tot} as 115W at 25°C, thus T_j = 115°C and θ_{jc} = 0.2°C/W (25°C/P_{tot}). It is estimated that, for most applications, θ_{cs} = 0.1°C/W, assuming thermally conducting compound and a mica insulating washer are used. (If no insulating washer is used, θ_{cs} = 0.05°C/W but the heatsink may be 'live'.) Substituting in formula (3):

$$\theta_{sa} = \frac{115 - 25}{115} - (0.2 + 0.1)$$

$$= 0.3°C/W$$

which is the heatsink thermal resistance required to ensure correct thermal operation.

The above example assumes that the 2N3055 is working at its limit, and this should seldom be the case. In this example it would be usual to supply the pass transistor with 18V DC to allow adequate regulation, and the transistor would be run at 5A, or half the rated amperage. The difference between the supply voltage and the output voltage is 18 - 13.8 = 4.2V DC, hence the power to be dissipated will be 4.2 x 5 = 21W. Substituting this in formula (3) gives:

$$\theta_{sa} = \frac{115 - 25}{21} - (0.2 + 0.1)$$

$$= 4.0°C/W$$

This is the thermal resistance required for this power supply. If necessary a further allowance could be made for the inefficiency of the semiconductor and the calculated thermal resistance decreased to allow for this. In this example this factor has been ignored.

From Fig 25.19 it can be seen that about 6cm^3 of blackened or 9cm^3 of natural finished copper (vertical fins) will be a suitable minimum size. **Fig 25.21** shows a typical home-made heatsink based on the above calculations. Ideally the two bent plates should be soldered together to minimise any thermal resistance at this junction. Thermal conducting compound should be used if this joint is not soldered. The copper should be annealed before bending.

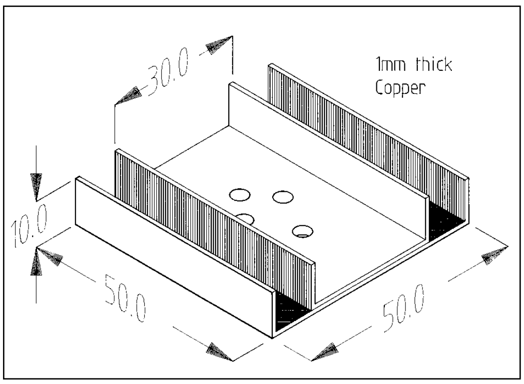


Fig 25.21: Heatsink based on the calculations of the example in the text

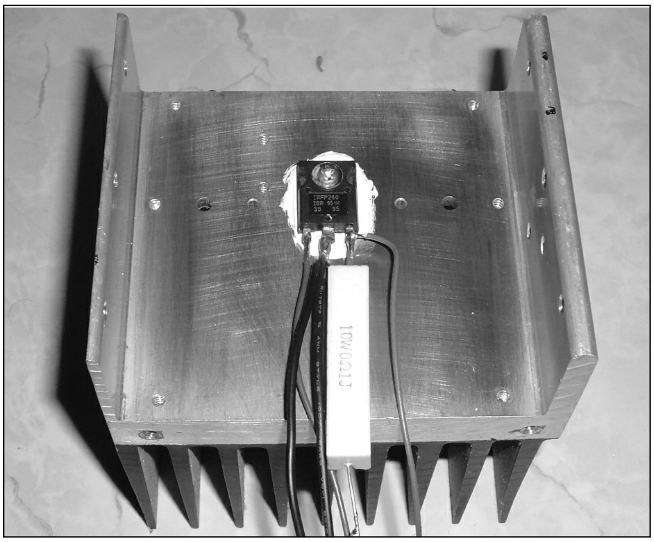


Fig 25.22: Power MOSFET on a substantial heatsink

Using a heatsink with a lower-than-calculated thermal resistance will not affect the operation of the cooled semiconductor. It would be unwise to use a heatsink with a higher-than-calculated thermal resistance. In other words, a larger heatsink is better than one too small (**Fig 25.22**)!

In use the heatsink will become warm to the touch but it should not become untouchable. If it does, then the semiconductor providing the heat is probably being overworked. It would be worthwhile checking the calculations and ensuring that each mating joint is made properly. Some semiconductor devices are designed to run at high temperatures and a hot heatsink would be expected. This heat represents wasted energy, and the necessity of large cooling systems, though seeming to be the norm, indicate inefficiency.

Professional designers use very sophisticated computer programs to design their heatsinks. Even then they rely on trials and measurements to tune-up their results! The above formulas and chart are accurate for most amateur purposes, eliminating the need for complex equations or computers.

Heat can also be dissipated from the connecting pins, together with the case, of semiconductor devices, and this can be useful if trying to keep the finished unit compact. A heatsink can be attached to the pin side of the board by a layer of thermally conducting, electrically insulating, elastomer and mechanically fastened to the board with screws. This method can provide about a 5 to 10% reduction in the required overall size of the main heatsink. Large soldering pads for the pin connections also act as heatsinks, and these can be used in conjunction with a suitable heatsink for such devices as audio amplifiers, power regulators, rectifiers etc.

The duty cycle of the device also affects the size of heatsink required. A 50% duty cycle can allow about a 20% reduction in the size of heatsink, depending on the heatsink design. During the OFF cycle, the heatsink must be able to dissipate all of the heat generated during the ON cycle, and it is usual in these circumstance to provide forced-air (or water) cooling to ensure that this is achieved. Many solid-state transceivers specify ON and OFF times for continuous full-power carrier (FSK, AM etc) operation. The small handheld transceivers start to get very warm if they are transmitting for longer than they are receiving. In the case of handheld transceivers, the batteries usually limit the transmit time but, if such transceivers were operated from a mains PSU, it would be essential to ensure that the heatsink arrangements were adequate.

CIRCUIT CONSTRUCTION

Every electronic circuit consists of a number of point-to-point electrical connections. Modern mass produced electronic circuits are usually built on some form of printed circuit board. Older production methods use a variety of point to point connection methods, usually involving tagstrip or stand-off terminals and insulated wire.

Many amateur circuits need to be functional rather than reproducible. If the project is to be housed in a screened enclosure, it won't really matter whether it is visually ugly or attractive. For portable operation, weight or size can be the most important factor. There are many different construction methods used by home constructors. Before you start building your project, you should consider the various options available and try to choose a method that best suits your needs.

The following list suggests some of the factors worth considering before starting on any circuit design and making:

1 Permanent or experimental?
2 How will the circuit/s be housed and mounted?
3 What power supplies are required (battery, mains, internal, external etc)?
4 What types of inputs/outputs are required?
5 What controls are required?
6 What safeguards are required (eg accidental switch-on, wrong polarity, over-voltage etc)?
7 Methods of construction. (Can you make it?)

In the rush to make the circuit, it is all too easy to discover afterwards that some of the above factors were important! Many solid-state devices are susceptible to damage by static discharge. Handling and soldering such components requires care. It is safer to assume that all semiconductor devices are prone to static discharge damage and treat accordingly. Assembly and repair should be carried out using the normal anti-static precaution of connecting the soldering iron, PCB and operator to a common point. This equalises the static level and is further improved if the common point is properly grounded.

Commercial equipment is available, consisting of an anti-static mat, wrist strap and connecting leads for the other tools. Semiconductors which are very sensitive to static are usually supplied mounted in a metal strip or wire clip (or a carbon-conductive foam in the case of ICs) which shorts together every pin of the device. This anti-static protection should be left in place until the device is plugged or soldered in position, and then carefully removed. A suitable size 'Bulldog' clip connected to the ground mat by a length of flexible wire makes an excellent anti-static connector for ICs.

Experimental, Temporary or Prototyping Circuits

One of the most important features in an experimental or prototype circuit is the ability to make quick and easy changes to the circuit. The effect of the changes is then observed and further changes are easily made if necessary. If you take a wrong turn in the development of your circuit and the modifications are unsuccessful, the circuit can easily be restored to it's original configuration.

The word 'breadboard' is often used to describe an experimental circuit. At one time, it was common practice to use a wooden board and conductive nails or pins to keep the components in place. The components and interconnecting wires could be soldered to the nails or simply wrapped around them. Since the days of the original wooden breadboard, both amateur and professional constructors have devised many ingenious methods of circuit prototyping. We will examine some of the most popular methods in detail.

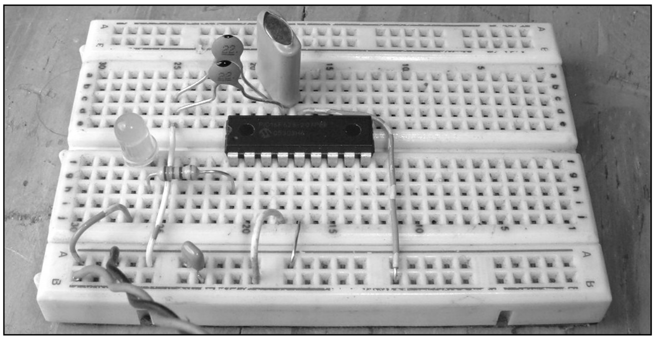


Fig 25.23: Modern breadboarding system

The modern equivalent of the old fashioned breadboard is still known by the same name as it's predecessor. Breadboard prototyping systems are available from most component suppliers. The most common type of breadboard is shown in **Fig 25.23**.

This type of prototyping system uses a 0.1in matrix of holes for mounting the components. This is ideal for mounting DIP ICs and other components with a 0.1in lead spacing. Each row of holes is numbered and connected internally to the other holes in the same row. There is usually a gap at the centre of the breadboard to facilitate the mounting of ICs. Such breadboards are available in a variety of sizes and some of the more expensive types come with a built in power supply.

One of the simplest and most popular construction methods uses a sheet of copper-clad PCB laminate or other conductive material as a base or ground plane for the other components. Any components that require a connection to ground can be soldered directly to the metal ground plane. Other components can be soldered together using copper wire or mounted on insulating stand-off terminals. If PTFE or ceramic stand-off terminals are not available, high value resistors can be used instead. A 10MΩ resistor is not likely to have any significant effect on the operation of a typical circuit.

This method is widely known as 'dead bug construction', probably because of the resemblance of an inverted IC to an insect lying on it's back. This is one of the most flexible of all prototyping systems. Apart from a soldering iron, no special tools are required for this construction method. Changes or additions to the circuit are trivially easy and this technique probably offers the fastest and simplest practical method of getting your circuit from a paper schematic to an assembled circuit. **Fig 25.24** shows a dead bug circuit of an audio preamp and power amplifier.

There are several types of point-to-point construction which are based on a collection of isolated pads or 'islands'. One such method uses a grid of square or rectangular pads that are cut or

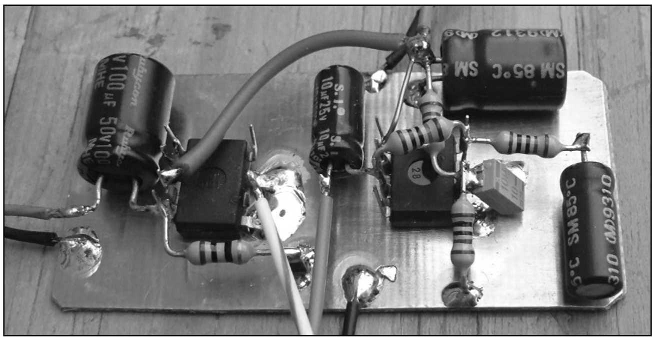


Fig 25.24: 'Dead bug' construction

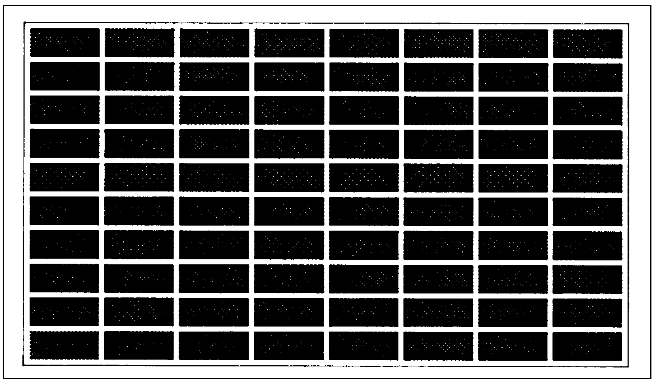


Fig 25.25: Grid pattern PCB for experimental circuits

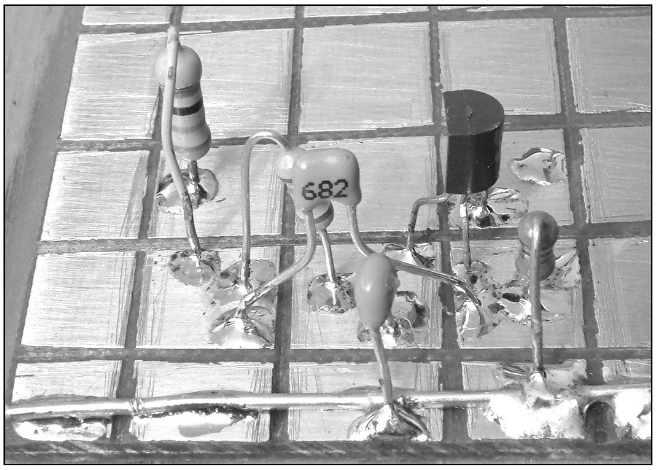


Fig 25.26: 'Pad' or 'island' construction

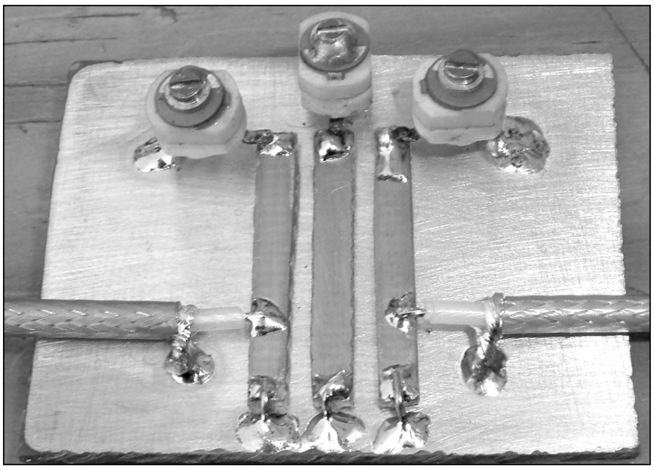


Fig 25.27: Microwave filter built using glue-on copper laminate

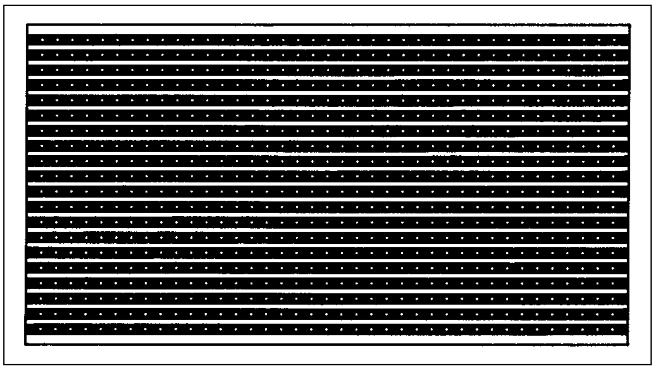


Fig 25.28: Strip board PCB suitable for experimental or permanent circuits

etched on the copper side of copper-clad PCB laminate (Fig 25.25). The component leads are soldered directly to the copper pads. Connections to adjacent pads can be made with a blob of solder. Connections to more distant parts of the circuit can be made using insulated wire. Fig 25.26 shows an active low pass filter built on a square grid which was cut into the face of a copper-clad board using a hacksaw blade. The components can be surface mounted on the copper side of the board or if the board is drilled, the components can be mounted on the opposite side of the board. This is probably the easiest way of making a double-sided circuit board.

A few variations of the pad/island method have evolved over the years. Some builders use small square or circular pads made from PCB laminate which are glued to a copper ground plane. One example of this type of construction is known as 'Manhattan Construction', presumably because it resembles an aerial view of city blocks. Each pad is glued to the copper plane using super glue. Each component can be mounted on a collection of small generic pads or alternatively, special dedicated pads can be made for each type of component. Special pads for IC, transistors, etc can be cut from PCB laminate. At least one company sells ready made pads for this type of construction. As with dead bug style construction, ground connections are soldered directly to the copper plane.

Dead bug and Manhattan style construction are excellent ways of making UHF and microwave circuits. Resonant sections of transmission line can be formed by glueing rectangular strips of single-sided PCB to a ground plane. Microstripline circuits made by this method are much more forgiving of errors in construction or design than conventional etched PCBs. If a line is found to be slightly off resonance, it can be lifted off the board and re-trimmed or replaced by an new one. A similar problem with an etched PCB would usually require the replacement of the entire board. Glue-on microstripline circuits are very easily interfaced with dead bug and other similar methods of construction. Fig 25.27 shows a UHF band-pass filter made from three narrow strips of copper laminate.

Strip board offers many of the advantages of conventional PCB construction. A typical strip board is made from SRBP (synthetic resin bonded paper) or other composite material. The board has a 0.1in x 0.1in matrix of holes and parallel copper tracks with a 0.1in spacing on one side of the board (Fig 25.28). The components are mounted on the top side of the board and the component leads are soldered to the tracks on the bottom of the board. A special cutting tool is used to cut any required gaps in the tracks. A sharp knife or drill bit makes a good substitute for this tool. Strip board is also available as 0.1 matrix board without any copper tracks. This type of matrix board is used as a framework for building circuits. The component leads and connecting wires are soldered together at the back of the board. Both types of strip board are often used in combination with wire wrap pins and tools. Wire wrap is a method of assembling circuits which uses special pins, wire and a wire wrap tool (rather than solder) to make electrical connections. Fig 25.29 shows a circuit built on stripboard.

One method we have not considered so far is virtual construction. Instead of actually building a prototype, it is possible to create a mathematical model of the circuit and use electronic simulation software to test the circuit. Suitable software is available for most computer platforms and operating systems.

Simulation can be a very valuable tool for the circuit designer. It can sometimes allow you to test circuits that would otherwise

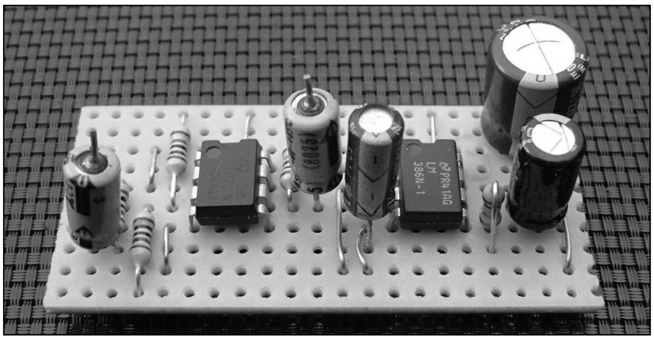


Fig 25.29: Stripboard construction

be difficult to design and build. Testing on a simulator can give the designer the opportunity to experiment with component values when fine-tuning a design. Changes can easily be made to a circuit in a matter of seconds. All but the most complex simulations will take no more than a few seconds to run on a modern computer.

Electronic simulation software can give very accurate results, but it is important that the user has a good understanding of the software and it's limitations. A simulator is only as good as the mathematical algorithms used by the programmer that created it. Even if the simulator is very accurate, it is only going to produce accurate results if it is given the right data. Even a seemingly small error in creating the model can have a dramatic effect on the simulation results. A capital 'M' instead of a lowercase 'm' or vice versa could turn millivolts into megavolts or megohms into milliohms. Computer software often lacks any kind of sanity check so it might allow to to put 100 megavolts across a 10 ohm resistor without asking if this is really what you intended to do.

Fig 25.30 shows a simulation of a VHF power divider and **Fig 25.31** shows the predicted frequency response of the circuit. The simulation software used is QUCS (quite universal circuit simulator) [2] which is free and open source software for Windows, Linux and other Unix based systems like BSD and Mac OSX.

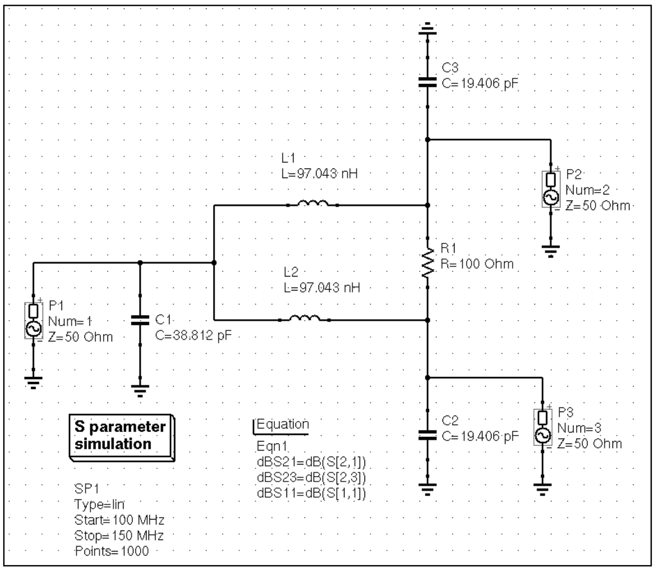


Fig 25.30: Computer simulation of a VHF power divider

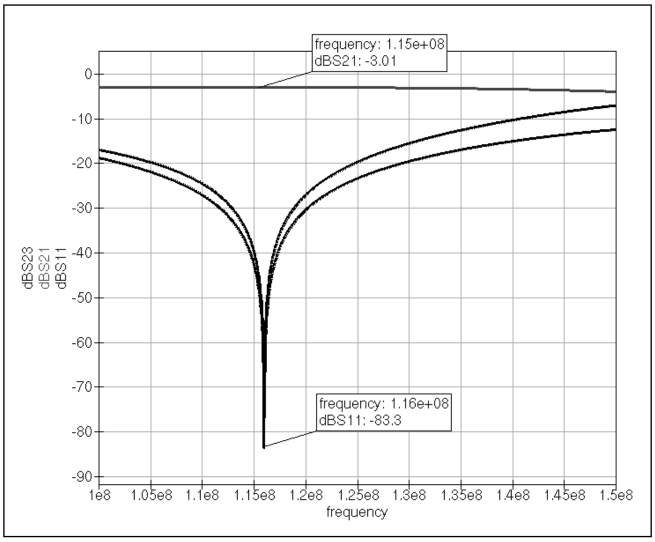


Fig 25.31: Computer-predicted frequency response of simulated circuit from Fig 25.30

Permanent Circuits

Although they are described as experimental or prototyping circuits, all of the circuit construction methods mentioned so far are quite suitable for use in permanent circuits. Some of my dead-bug circuits have given more than 20 years of reliable service and will hopefully continue to work for many more years.

Printed Circuit Boards

Almost all modern electronic circuits are built on some form of printed circuit board. The most common type type of PCB consists of one or more layers of copper foil which is bonded to an insulating substrate. The substrate is usually made from a composite material like fibreglass/epoxy or SRBP. Some specialised PCBs use a ceramic substrate for use at high temperatures or as part of a hybrid module.

Most of the PCBs made by home constructors will be either single-sided with a single conducting layer on one side of the board, or double-sided with a copper layer on both sides of the board. Multi-layer boards with more than two layers are commonplace in commercially made equipment, but they are rarely seen in home made equipment.

Fibreglass PCB is probably the best choice for home constructors. Fibreglass is very durable and it has very predictable electrical characteristics. The great strength and durability of fibreglass PCB makes it quite difficult to drill and cut. Always wear a mask and work in a well ventilated area when cutting, drilling or grinding fibreglass. SRBP board is cheap and readily available. It is easier to drill than fibreglass, but it can also be difficult to cut with a saw because it cracks very easily.

Before we can make a PCB, we must first design a suitable track layout for the circuit. The rules for designing a PCB layout are much the same as for any other type of circuit construction. The designer must ensure that adequate grounding, decoupling and screening are used at critical points in the circuit. The input and output of high gain amplifiers must be well isolated from one another. Ground connections should be short and direct. One approach that is recommended for radio frequency or high-speed digital PCBs it to use double-sided PCB and use most of, or even all of one side of the board as a ground plane.

For a simple one-off PCB, the track pattern can simply be drawn directly on the copper PCB with an etch-resist pen. Some types of permanent waterproof pens can be used as etch-resist pens. I have had good results with fine laundry markers and CD/DVD marking pens. The board is then immersed in an etchant which will dissolve the exposed copper, leaving only the tracks that are covered by the etch-resist ink. Etch-resist pens and suitable etchant are available from many component suppliers. After the etch resist is removed from the board, it can be drilled and the components mounted on the board and soldered into place.

General PCB guidelines

1. Keep mains and high-voltage circuits well away from other circuits, preferably by using separate boards. In the interests of safety, any tracks etc carrying high voltages should be covered by an insulating material to prevent accidental contact.
2. Avoid high current density points. Rapid changes of line width should be avoided, particularly in circuits carrying several amperes. The recommended current density I in amps is:

   $$I = 3(\sqrt[3]{w^2})$$

   where w is the width of the copper track in millimetres. Circuit board tracks make excellent fuses if incorrectly designed!
3. The capacitance C in picofarads between each side of a double-sided board is:

   $$C = 0.0885 \times K \times (A/h).$$

   where A is the area (sq cm) of the smallest side; h is the thickness (cm) of the laminate excluding copper, and K is the dielectric constant (relative permittivity) of the laminate material, which for glassfibre is about 4.5.

Creating the PCB artwork

There are many different ways of creating the track pattern for PCBs. Instead of drawing the pattern directly onto the copper board as described above, the pattern can be drawn by hand on white paper and then photocopied onto a clear acetate overhead projection film which can be used to produce a PCB by photographic methods. Some of my first PCB layouts were created using rub on dry transfers for the component pads and rub-on tape for the tracks.

Such techniques are rapidly becoming obsolete due to the ready availability of cheap personal computers and PCB CAD (computer aided design) software.

The design of simple PCB layouts is not very demanding in terms of computer resources. A simple PCB CAD program will run on a very humble PC. One of the computers that I use for PCB design has an 850MHz CPU and 384MB of RAM. This computer was rescued while it was on its way to the recycling centre. Such a computer would be considered by many as little more than a doorstop and could be acquired at little or no cost. The Linux operating system and PCB [3] CAD software are both free and open source, so the total expenditure on hardware and software was somewhere in the region of zero.

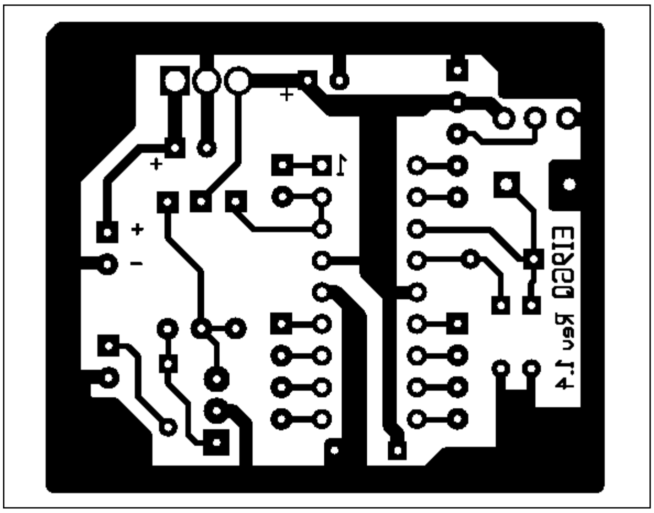


Fig 25.32: PCB drawn using the industry standard Gerber format for CAD

If you don't have a PC in the shack, you really should consider getting one. Even a very basic model can be a very useful design tool. See the chapter on computers in the shack for many other amateur radio applications for a computer.

The size and complexity of PCB design software varies from the very simplest CAD program which can be used to draw single or double sided track layouts up to very large and complex EDA (electronic design automation) packages costing thousands of pounds. If you don't require advanced features like schematic capture, auto routing and circuit simulation, an ordinary vector drawing program can be used to draw the track layout. However, it is probably worth spending a little time and effort to learn how to use proper PCB layout software.

You should beware of "Free!" CAD software because it often comes with strings attached. Some commercial PCB houses provide free CAD software for download. This free software often uses a proprietary file format that can only be used by the same PCB manufacturer that gave you the 'free' software. Unless you want to be tied to a single PCB supplier, you should make sure that your PCB CAD software can produce output files in the industry standard Gerber format. **Fig 25.32** shows a PCB layout drawn using PCB CAD software.

Once the PCB artwork has been created, we must find a way of getting the track pattern from the computer screen onto the copper-clad board. Commercial PCB makers use a special high resolution plotter to print the layout onto a clear film which can be used as a master artwork for the PCB production process. Alternatively, the artwork is printed on paper and then photographed with a special camera. It is possible to make PCBs at home using similar methods.

Before we consider these methods, we will look at an even easier way of making prototype PCBs [4] The plastic toner used in ordinary laser printers and photocopiers is quite waterproof and makes a very good etch-resist. Special PCB transfer films are available from specialist PCB suppliers and some electronic component suppliers. The film is available in A4 size sheets. The process for making the PCB is quite simple. First, a mirror image of the PCB track pattern is printed onto the transfer film using a laser printer or photocopier. After the copper PCB has been carefully cleaned, the transfer sheet is placed toner side down in contact with the copper PCB and the toner is ironed onto the copper using a standard clothes iron. The transfer sheet is then carefully peeled away from the copper board, leaving the toner behind. At this stage, any flaws or gaps in the track pattern can be carefully touched up with an etch resist pen. After the board has cooled down, it can be etched in the same manner as a hand drawn or photographically produced board.

Many home constructors have found that the type of glossy photo paper that is normally used in inkjet printers makes a good substitute for the special toner transfer film. I have had particularly good results with Epson branded Glossy Photo Paper. Before we progress to the printing stage, a few words of warning are in order.

Putting anything other than the manufacturers recommended paper or film through your laser printer or photocopier could damage it. Using the printer in this manner will probably invalidate the manufacturers warranty. The printer/copier fuser unit is a high temperature roller which is used to melt the toner into the surface of photocopy paper. Some types of plastic OHP film or plastic coated paper could make a horrible mess of it. If in doubt, you can check the heat resistance of your paper by running a hot clothes iron over it. If you have any reservations about putting inkjet paper through your laser printer, then you should buy the 'proper' transfer film instead. You have been warned! I have had good results with TEC200 film in the past. The most popular type of film currently on the market is called Press-N-Peel.

The track pattern should be printed onto the transfer film using an ordinary black laser printer or photocopier. Since most PCBs will be much smaller than an A4 sheet, you should be able to make several PCBs for each sheet of film. If you use transfer film, you should pay close attention to the manufacturers instructions. If you are using glossy photo paper, the track pattern should be printed on the glossy side of the paper using a laser printer and not an inkjet printer.

Remember that the process of ironing the toner onto the surface of the board will reverse the image. The image printed on the film must be a mirror image of the track pattern that will appear on the finished PCB. use the print mirror image function of your PCB CAD software. If you are using ordinary drawing software, you will find that most if not all drawing software will have a 'flip horizontal' function that will produce the desired effect. Before you commit your track pattern to an expensive sheet of transfer film, you should first print a copy on ordinary A4 paper. Hold the paper up to the light and make sure that the track pattern is correct. Check the size of the image carefully. If the pattern contains pads for an IC, you can use a spare IC or an IC socket to check that the pin spacings are correct. Once you are satisfied that the track pattern is ok, the image can be printed on the film. When the film has been printed, you should avoid handling it. Fingerprints on the film will lead to poor adhesion of the toner to the PCB.

The edges of the film or glossy paper should be trimmed with a scissors so that the remaining film is just slightly bigger than the track pattern. The PCB should be cleaned with very fine steel wool, rinsed thoroughly and dried with a lint-free cloth. The transfer film should then be placed toner side down on the copper PCB. The film can be secured to the PCB using a single length of paper masking tape along one edge only. Vinyl tape is not suitable for this job. The PCB and transfer film should be placed on a heat resistant platform before ironing. I use the back of a telephone directory for this purpose. The instructions below are for glossy photo paper. For special toner transfer films, you should follow that manufacturer's instructions instead.

The toner is then ironed onto the surface of the board using a hot clothes iron. The amount of time and pressure required will depend on the temperature of the iron and the type of toner used in the printer or copier. I use a hot iron at close to maximum setting. The iron is placed on the back of the photo paper and pressed firmly down for about 10 seconds while the toner sticks to the PCB surface. The iron is then moved slowly back and forth for a further 15 to 20 seconds. The amount of pressure required will depend to some extent on the complexity and size of the PCB pattern. Heavy pressure on the iron will tend to flatten the toner and tends to make lines slightly thicker and pads slightly larger. Do not attempt to peel the photo paper off the PCB at this stage. The board should be placed in warm soapy water which will soften the photo paper. You should then unplug the iron and leave the room immediately. All homebrew radio enthusiasts have a natural curiosity and a tendency to poke at things, even when we know we shouldn't.

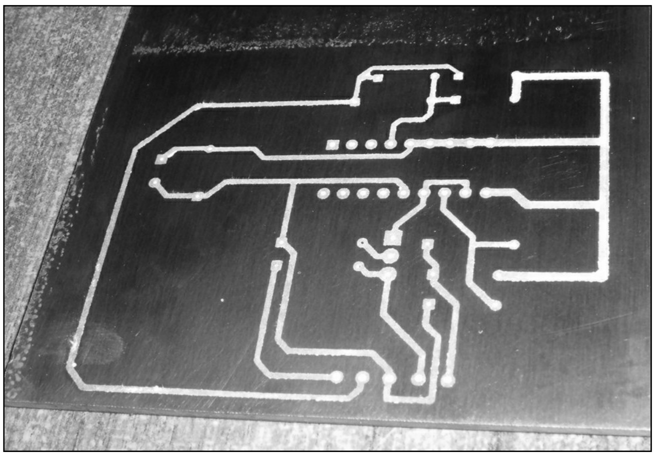


Fig 25.33: Home made PCB after ironing

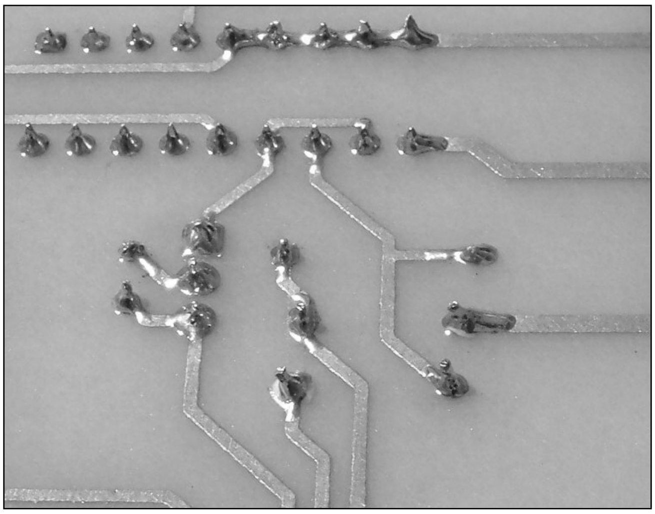


Fig 25.34: Home produced PCB

Leave the board in the warm water for at least twenty minutes before you try to remove the photo paper. By this time, the paper will be very wet and should be easy to remove. Any remaining paper residue can be removed by brushing it off with your finger tip while holding the board under running water. Very stubborn residue can be removed with a coarse sponge. Don't be too afraid of damaging the track pattern, the toner is usually quite durable. Any small gaps in the toner can be repaired with an etch-resist pen. **Fig 25.33** shows the board after ironing. The board is now ready to be etched.

The toner transfer method is a good way of making one-off PCBs. The resulting boards are not quite as good as photographically produced boards from commercial PCB makers, but as you will see from **Fig 25.34** they can come quite close and are certainly good enough for most amateur projects. Most PCB CAD software can be used to generate one or more silk-screen layers. The silk-screen layer can be used to show the outline of components, component values and other text like PCB revision information.

If you want to give your home made PCBs the professional look, you can use the toner transfer method to print the silk-screen layer on the top of the PCB. Simply iron a mirror image of the silk-screen layer onto the top of the PCB. The PCB should be drilled first so that the component holes can be used as guide for positioning the transfer film. **Fig 25.35** shows the silk-screen layer of a home made PCB. The term silk-screen is usually used to describe this layer, even when a printing technology other than silk-screen printing is used.

Photographic PCB production

The best way to make very high quality PCBs is to use the same photographic methods that are used by commercial PCB makers. One of the biggest advantages of the photographic method is the ability to make many identical PCBs from a single master artwork.

Most photo sensitive PCBs are coated with a positive photoresist. This means that we will need to use a positive artwork in which the copper areas are opaque and the copper-free areas are transparent. Note that some PCB production systems require a negative master artwork. The PCB artwork is created using a CAD system as used for the toner transfer method above. The artwork is then printed onto a clear film. A laser printer or photocopier can be used to print the artwork onto a clear acetate overhead projection film. You must use the type of overhead projection film that is designed for use in photocopiers and laser printers. I have had very good results with Staedtler brand OHP laser film.

Photo sensitive PCB is coated with a special UV light sensitive photo-resist. To transfer the track pattern to the PCB, the positive artwork is placed in close contact with the photo-resist and the board is exposed to a source of UV light. After exposure the board is developed in a bath of developing solution. The photo-resist that was exposed to UV at the exposure stage of the process is washed away by the developer, leaving only the photo-resist that was protected by the opaque parts of the master artwork.

Exposure

The first, and probably the most critical step, in the process is the exposure of the photo-sensitive PCB laminate to a UV source. A high quality master artwork is the key to success. The printed image from most laser printers and copiers is not quite opaque enough to make a good master artwork. You should adjust the printer settings for maximum toner density for best results. I have found that it is often necessary to stack two identical copies of the printed artwork to obtain the required opacity. The two sheets must be very carefully aligned. Most PCB CAD software will produce alignment marks which will help you to achieve precise alignment. When you cut out the two copies of the master artwork from the A4 sheet, one copy should be cut slightly larger than the other. The larger copy should be placed on a flat surface and the smaller copy placed on top of it. When the two sheets have been perfectly aligned, A strip of Sellotape can be placed along one edge of the upper sheet to secure it in position. Hold the artwork up to the light and check it for any flaws or alignment errors. Note that the toner should be printed on the side of the OHP film which will be closest to the PCB. If the toner is printed on the wrong side of the film, some UV light will be able to creep around the edges. This can lead to thinning and fuzzy edges of the PCB tracks.

Some home constructors have reported that they can make good master artwork using an inkjet printer and the type of OHP film that is specially designed for use in inkjets. My earlier attempts to make PCB artwork using a standard low cost inkjet printer were far from successful. Even though the printed artwork looked quite good, the black ink from my old inkjet printer seemed to be almost completely transparent to UV light. My more recent efforts have been much more successful.

I now use a photo quality HP D5160 inkjet and Stephens brand inkjet film from a local newsagent. This produces a very high quality positive artwork which is at least as good as the best results I have seen from the laser printer. With a B&W print resolution of 1200dpi and even higher resolution in photo mode, this type of printer can produce artwork which is good enough for the very fine tracks and pads used for surface-mount components. You should check that the type of film you are using is compatible with your inkjet printer. However, inkjets are much less prone to damage than laser printers because they don't use a high temperature pressure roller (toner fuser). Even in the worst case scenario where the print head is destroyed, HP and several other brands of inkjet have the print head incorporated into the ink cartridge. If you do manage to damage the print head, a new cartridge will probably cure the problem. As with the artwork produced by the laser printer, I often use two stacked sheets of film to achieve the best possible results.

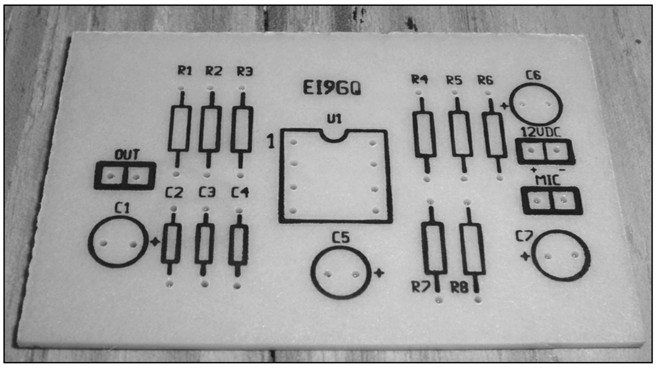


Fig 25.35: Silk screen layer on home made PCB

The PCB can be exposed using a commercial or home made UV exposure unit. A typical unit will consist of a lightproof box containing UV fluorescent tubes, a sheet of glass for mounting the PCB and a timer. UV light can be very harmful. A home made exposure unit would have to be built in a totally enclosed box and the lid should be fitted with a safety switch to ensure that the tubes can never be illuminated while the box is open. A simple timer mechanism should also be included. Exposure to high intensity UV can lead to eye damage or even total blindness. The type of UV tubes used in EPROM erasers and the germicidal tubes used in sterilising units produce short wavelength UV which is considered particularly dangerous.

Ultra violet fluorescent tubes of the kind used for disco lighting (black light), banknote scanners, suntan lamps or insect zappers can be used for PCB exposure.

Ordinary fluorescent tubes can also be used, but exposure times will be much longer. A single Philips 30 Watt TLD 30W/33 tube gave good results with an exposure time of one hour at a distance of 10cm. Many constructors use photographic equipment to expose PCBs. A photographic contact frame can be used to hold the PCB and artwork and a photoflood bulb used as the UV light source. Optimum exposure times will vary depending on the power of the bulb and the distance of the PCB.

One very obvious source of UV light is the sun. The PCB exposure frame is simply exposed to direct sunlight for a few minutes. This method can sometimes produce excellent PCBs, but the unpredictability of the weather makes it difficult to get consistent results. I have found that the optimum exposure time to direct sunlight is somewhere between two and five minutes. My tests were performed under a clear blue sky in the early afternoon.

After many years of experimenting with different types of UV lamps, I have found that the best UV light source can be purchased for no more than a few pounds from any DIY or electrical supplier. Quartz-halogen lamps of the type used for garden lights and security lighting produce enough UV light to expose a PCB in a few minutes. I use a 500W halogen lamp and a very simple exposure frame made from a small sheet of glass. The glass from a small picture frame is ideal for this purpose. The OHP film master artwork is secured to the glass with a strip of Sellotape and the PCB is held in place by a small clamp made from a strip of aluminium. The 500W halogen lamp produces a lot of heat so the artwork should not be placed too close to it. A 28-30cm spacing between the lamp and the exposure frame ensures that the PCB doesn't get too hot and gives an exposure time of about 10 minutes. The lamp should not be left unattended because of the risk of fire or heat damage. **Fig 25.36** shows the halogen lamp and exposure frame.

Fig 25.36: Simple exposure unit using 500W security lamp

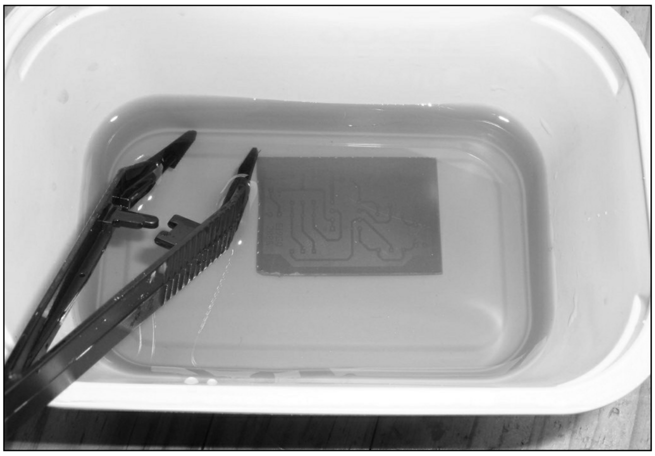


Fig 25.38: PCB in the developing solution

A strip of the photosensitised board should be used to assess the exposure time required. Fig 25.37 shows a suggested exposure testing strip, which should be made of the same materials as those used for the master transparency. Each section of the test strip is progressively uncovered until the complete strip has been exposed. For example, if the timer is set to, say, 20s then each step would be 20s greater than the previous one. This is normal photographic practice. The fully exposed strip is now developed. The image of the test strip should be visible, at least in sections, and at this stage the board is washed thoroughly in running water, dried and then etched. It is possible to assess the correct exposure without etching but etching is more reliable and much easier to see.

The developing is usually carried out by using sodium hydroxide dissolved in water in the ratio of between 7 and 16g per litre, depending on the resist type. Cooled boiled water is preferred. During mixing, the sodium hydroxide tends to produce a boiling effect and great care should be taken to prevent contact with the eyes etc. The usual safety precautions should be observed: glasses, gloves, no children or pets etc. The mixed solution has a limited life and a fresh solution should be made for each batch of developing.

Developing

The two most commonly used PCB developers are sodium hydroxide and sodium metasilicate. Sodium hydroxide is widely available from most hardware stores as caustic soda or drain cleaner. Both types of developer can give excellent results. I have found that caustic soda is a bit more difficult to use. To get good results, the strength and temperature of the caustic solution must be carefully controlled. Sodium metasilicate based developer is much less critical and will deliver consistent results over a wider temperature range.

PCB developer is a caustic solution which should be kept away from the eyes and skin. You should wear eye protection and rubber gloves when working with the developer. One teaspoon (5gm) of sodium hydroxide crystals (caustic soda) dissolved in 500ml of water make an excellent PCB developing solution. Wait until the crystals have fully dissolved before you use the developer. Sodium metasilicate developer is available in liquid or crystal form from most electronic component suppliers. You should follow the suppliers instructions for mixing and safe handling.

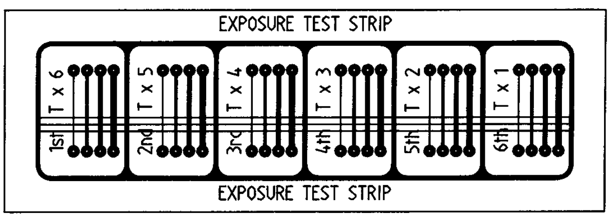


Fig 25.37: Exposure test strip for photosensitised PCB

I use a plastic margarine or ice-cream tub as a PCB developing tray. The PCB should be placed resist side up in the developer. Make sure that the PCB is not exposed to direct sunlight or strong fluorescent lights before it is developed. The developing tray should be rocked gently from side to side to while the etch resist is washed away. The developing time will vary depending on the UV exposure time, the strength of the developing solution and the thickness of the photo-resist layer.

After about one minute in the developer, you will see the outline of the PCB tracks appearing on the board. After two minutes, you should start to see bare copper. At three to four minutes, the board should be almost completely developed. The board should be left in the developer for a further minute to ensure that all of the exposed etch resist is washed away. The board should now be removed from the developer and thoroughly rinsed with running water. If your developing time is much longer than 4-5 minutes, you probably need to increase the UV exposure time or slightly increase the strength of the developing solution.

The remaining photo resist should be an exact replica of the master artwork. The board should be handled very carefully at this stage. The resist must not be scratched before the board is etched. Fig 25.38 shows a PCB in the developing solution.

Etching

The two most common types of PCB etchant are ferric chloride and sodium persulphate.

The optimum temperature for PCB etching solutions is higher than room temperature. Commercial PCB etching tanks are fitted with heating elements and mechanical agitation systems to speed up the etching process. Sodium persulphate doesn't work very well at room temperature, so ferric chloride is probably the best etchant for home PCB construction unless you have a heated etching tank. Ferric chloride is corrosive and quite toxic in concentrated form. However, it is probably no more dangerous than some household chemicals like bleach or drain cleaner. Wear rubber gloves and eye protection. Wear old clothes, ferric chloride stains won't wash out easily. Etching time depends on the temperature, strength and condition of the etchant, the thickness of the copper foil and the area of copper to be removed. Boards that are upside down (with the copper facing downwards) will etch faster than boards that are lying copper

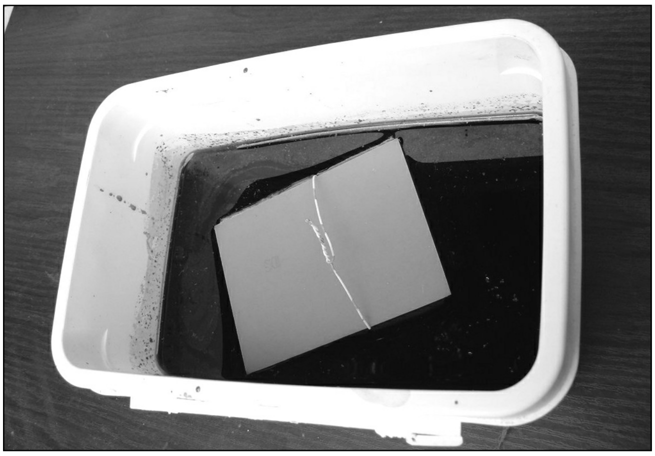


Fig 25.39: PCB in the etchant

side up. Double sided boards can be etched standing on edge if the etchant is deep enough. Typical etching times range from five minutes for a heated/agitated tank up to about an hour for an unheated etching tank on a cold day. A few tests with off-cuts of scrap PCB will help you to determine the optimum etching time, temperature and etchant strength.

Fig 25.39 shows a PCB in ferric chloride etchant. Note that the PCB is floating face-down on top of the etchant. Also note the length of insulated wire that allows for handling of the board without coming into contact with the etchant. The board should be lifted and wiggled from side to side several times during the etching process. This board took about 35 minutes to etch at a room temperature of 22°C.

I would recommend that you use a deep plastic tub rather than the shallow trays that are supplied with most etching kits. Metal tubs should be avoided for obvious reasons! Once the board is etched it should be rinsed with clean water. If you find that there are still a few traces of unetched copper, you can put the board back in the etchant for a few minutes before you rinse it again. After etching, the resist can be removed from the board with paint/varnish remover or steel wool.

Double-sided boards

There are a few different ways of making double sided boards using photographic methods. It is possible to process both sides of the board at the same time by using a double sided exposure frame and a pair of UV light sources. This requires extremely accurate alignment of the artwork on both sides.

I find it is a lot easier to do one side at a time. The first side of the board is exposed, developed, etched and drilled as it would be for a single sided board. Use a sharp new drill to reduce the risk of damaging the photo-resist and protective coating on the opposite side. When the first side is finished, it can be covered with strips of PCV tape to protect it while you are etching the second side. Now you can remove the plastic protective film from the second side of the board. The artwork for the second side should be aligned carefully with the holes in the board. This ensures that both sides of the board are properly aligned. The second side of the board can now be exposed, developed and etched. Be very careful to protect the first side from the etchant while you are etching the second side. After etching, the protective tape and photo-resist can be removed, hopefully to reveal a perfect double sided PCB.

Double sided boards can also be made by etching a pair of single sided boards and sticking them together with epoxy resin. If you use 0.8mm laminate, the finished product won't be much thicker than standard 1.6mm board.

SMT (surface mount technology) boards are made using exactly the same methods as boards for leaded components. PCBs with leaded components are often referred to as PTH (plated through hole) technology. This is a bit of a misnomer in the case of home made boards because most amateurs will not have the equipment to make boards with electroplated holes. I have used the photographic methods described here to make boards using surface-mount ICs and 0603 type chip capacitors and resistors. The main differences between boards for leaded components and boards for SMT are the methods used for mounting and soldering the components. It is not too difficult to solder most SMT components provided that you have a good magnifying glass and a small tip for your soldering iron. Some of the newer SMT packages like the BGA (ball grid array) package have all of the connections underneath the chip so that no pins are visible. Such technologies will provide a new and difficult challenge for the home constructor.

Drilling the PCB

Hand cranked, egg-beater style drills are just about usable for hand drawn PCBs. To drill the tiny holes required by CAD produced PCB layouts, a 12V miniature PCB drill or a mains powered Dremel or similar type of rotary tool will do a much better job.

After many years of breaking drill bits, I have at last discovered a reliable way of drilling holes in PCBs. I use a 12V PCB drill with a reduced supply voltage of around 6 volts. Using a lower supply voltage reduces the kick caused by the torque reaction when the drill starts turning. The lower voltage also reduces the speed of the drill which results in longer life from HSS drill bits. I use a push-to-make foot switch to operate the drill. This leaves both hands free to hold the drill and PCB. You should wear safety goggles when using the drill.

To avoid drilling holes in the table, drill the board on on old telephone book or paperback. The exact procedure I use is as follows: Place the tip of the drill in the centre of the pad to be drilled. I use a 100W light so that I can see what I'm doing. Taking care to keep the drill absolutely vertical, press the foot switch. Don't hold the drill too firmly. The hole in the centre of the pad will act as a guide for the drill. Even if you are a little off centre, the drill will tend to centre itself with the hole in the copper pad. When the hole has been drilled, withdraw the drill vertically while it is still turning and release the foot switch. I use a drill size of 0.7mm to 1.0mm for most PCB work.

Commercial PCB makers use tungsten carbide drills for drilling fibreglass PCB. I have found that these drills are easily broken if they are used in a hand drill. My own preference is for HSS (high speed steel) drills which are relatively cheap and readily available. Most constructors recommend using a drill stand for PCB work. I prefer to use a hand-held drill and the above technique. I have drilled thousands of holes since the last

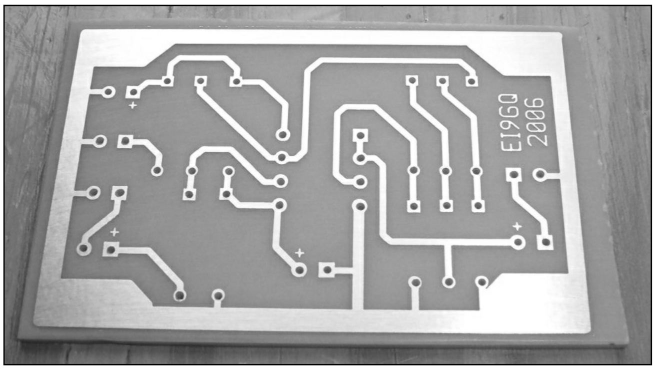


Fig 25.40: Completed home made PCB

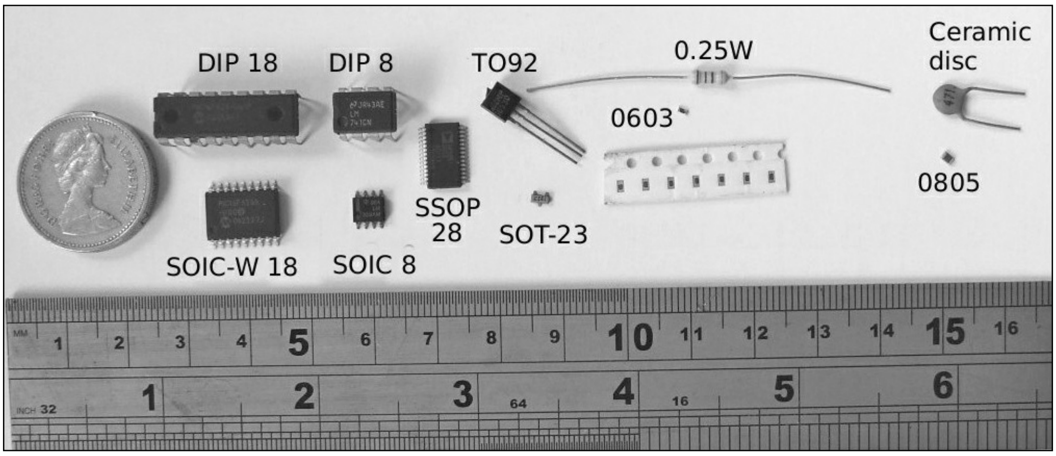


Fig 24.41: A selection of some of the most widely used SM package types showing their size

time I broke a drill-bit. Once the PCB has been etched and drilled, the components can be mounted on the board and the leads soldered to the component pads. Commercial PCBs usually have a protective layer of solder resist applied to the board. This is not strictly necessary for home produced boards, but if you want to preserve the shiny new appearance of your PCB, it can be coated with a clear lacquer after soldering. Special lacquers are available which will allow re-soldering of the board for future modifications or repairs. **Fig 25.40** shows the track side of a home made PCB after etching and drilling.

SMT

Boards for surface-mount components are made in much the same way as boards for through-hole components, but there are a few obvious differences. SMT boards are usually much smaller than their through-hole counterparts. As well as occupying a smaller area of PCB, SM components have a much lower profile than other component types. This makes them particularly well suited to double sided or multi-layer PCBs where components are mounted on both sides of the board. SMT layouts tend to use narrower track widths and pad sizes. SM devices usually use much smaller lead spacings than through-hole components. For example: SOIC (small outline integrated circuit) ICs have a pin-to-pin spacing of 0.05in (1.27mm) or 20 pins to the inch. This is exactly half the pin spacing of a standard DIP package. Smaller SM ICs use 40 or even 50 pins to the inch. **Fig 25.41** shows a selection of some of the most widely used SM package types.

Running tracks between IC pins is common practice for DIP ICs. This is not so easy with SOIC packages and just about impossible for smaller packages. This restriction greatly reduces the routing options available to the PCB designer. Regardless of whether the track routes are chosen by human brain power or auto-routing PCB CAD software, at some point you will find you are completely surrounded by tracks and there is no clear path to the next pad you want to reach. In this situation, the only way to avoid an ugly wire link is to use a double sided or multi-layer PCB.

Inter-layer connections are made by small plated holes called vias. Because vias are not soldered to the copper tracks, they are almost flush with the surface of the PCB. In many circuits, the vias can actually be hidden underneath surface mounted components. This is not easy to achieve with home made PCBs. Most home made PCBs use either a single sided or double sided layout. Vias are typically made by drilling through the PCB and soldering short lengths of wire between the top and bottom layer. Multi-layer (more than two copper layers) boards with electro-plated vias are not easily made in the amateur workshop.

The two main differences between SMT and through-hole PCBs are the way that the components are mounted on the board and the way the components are soldered to the copper pads. Surface mount components are as the name suggests, mounted on one surface of the PCB. Unlike single layer through-hole PCBs which have a 'component side' and a 'track side', single layer SMT PCBs have the components and the copper pads and tracks on the same side of the board.

The most common type of board used for home construction uses a double-sided board. Most of, or quite often all of the components are mounted on one side only. Hand soldered vias are used to make connections to the tracks on the other side of the board. Such boards are reasonably easy to make and offer a reasonable range of routing options to the layout designer. By convention, the component side is considered to be the 'top' copper layer during the design process. Commercially produced PCBs will usually have a solder mask layer and silk screen layer above the top copper later.

SMT Soldering

The basic principles of soldering for SMT boards is just the same as for any other type of construction. The solder joints are made by applying flux and solder to the metals to be joined and heating the joint to a temperature above the melting point of the solder. The methods employed for most SMT boards are often quite different to the methods used for through-hole boards. Solder and flux can be applied to the PCB pads in a paste form.

There are several different ways of applying the solder paste. One method used by commercial board makers is to apply the solder paste to the PCB through a screen. This is similar to silk screen printing except that a solder and flux paste is used instead of ink. The other method is to apply the paste to each PCB pad individually via a small nozzle. For one-off prototyping, reworking and repairs, the solder paste can be applied by hand using a tool similar to a syringe which has a metal nozzle at one end and a plunger at the other end. It is also possible to use a conventional soldering iron to solder SM devices.

Fig 25.42 shows a six pole SSB filter which uses HC49/SMT crystals and 0805 chip capacitors. This board was soldered by hand using a pointed tip in the soldering iron.

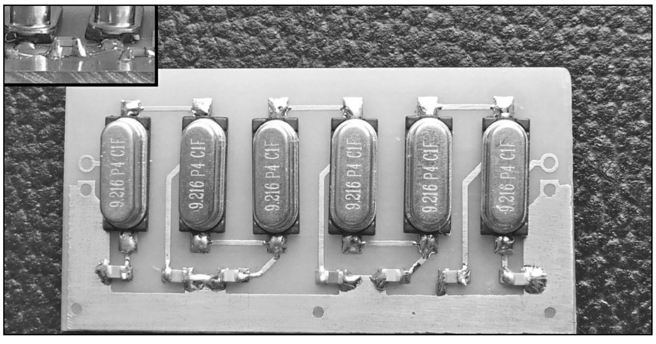


Fig 25.42: Hand-soldered SMT components in a filter

Once the solder/flux paste has been applied to the PCB, the components can be mounted in position using a computer-controlled pick-and-place machine. The components are fed to the pick-and-place machine in much the same way that bullets are fed to a machine gun using an ammunition belt. SM components are supplied on tape reels which have perforations along one edge of the tape. These perforations are used as a guide for the feed system of the pick-and-place machine. The components may also be glued to the board using a small dot of adhesive to keep them in place during soldering. This is particularly important if the components are to be wave soldered. For prototyping, reworking and repair, a simple vacuum operated hand tool can be used to place the components on the PCB.

There are several different methods of soldering the components to the PCB. Wave soldering is sometimes used for SMT boards or for boards that use a mix of SM and through-hole components. Wave soldering equipment works by running a wave of molten solder along the surface of the PCB. This method is ideal for through-hole PCBs because there are no components on the track side of the board. Without any components to get in the way, it is possible to solder the entire board in a single operation using a shallow wave of solder. Wave soldering is not particularly well suited to SMT soldering. The most common soldering method for SMT boards is known as reflow soldering. There are several methods of applying heat to the board. Infra-red, hot gas, and laser heating are all in common use. Each component of the board will have it's own specification for maximum soldering temperature and duration. Most modern components are able to survive the high temperatures encountered in lead-free reflow soldering. In a typical system, the entire board is pre-heated to a temperature of about 150°C at a controlled rate of about 2°C per second. After this 'preheat zone', the board is held at this elevated temperature for the second stage of the process. During this 'heat soak zone' the solder flux is activated. The next stage is the 'reflow zone' where the temperature is increased to above the melting point of the solder. This is typically 200-250°C.

The solder must remain liquid for long enough to flow properly around each joint. Surface tension will make the solder flow onto the solder pads and 'wick' along component leads. Temperatures much above 200°C are close to the maximum temperature limits for most semiconductors. Many components like plastic capacitors, switches, plastic connectors and even the PCB laminate can only survive such high temperatures for a short period of time. The final stage is the 'cooling zone' where the PCB is cooled to room temperature.

So how do we go about bringing all this space-age technology into the amateur's shack? How much expensive equipment will we need? You might be surprised to discover that SMT reflow soldering techniques are well within the capabilities of the amateur constructor. In some respects, reflow soldering is easier, cheaper and more consistent than conventional soldering of through-hole PCBs using a soldering iron.

A humble pair of tweezers is a good alternative to the expensive pick-and-place machine. Metal tweezers are suitable for most PCB work. Plastic or bamboo tweezers are also available for the more delicate jobs. The tweezers might be a bit slower than its industrial counterpart, but it will get the job done eventually. Component placement is not quite as critical as you might imagine. Surface tension during soldering tends to pull the components into line with the PCB pads, even when they have been slightly misplaced before soldering.

Specialised soldering equipment like vapour-phase ovens are staggeringly expensive. However basic infra-red ovens are not too expensive, although they are probably still beyond the reach of all but the most enthusiastic home constructor. The most popular home reflow oven is a domestic toaster oven of the type that might normally be used to make toasted sandwiches. Because the normal operating temperature of such an oven is far above the thermal soak zone temperature and the maximum setting is well above the reflow temperature of the solder, this approach requires a great deal of skill on the part of the operator. Some constructors have built microcontroller based systems to control the 230V AC current to the heating element so that the oven temperature can be accurately controlled. The control system can be programmed to provide the correct thermal profile for the reflow soldering process. A homebrew oven of this type can give excellent results.

Another method of reflow soldering uses a stream of hot gas to heat the solder. Simple systems use hot air, some more expensive systems use a stream of hot nitrogen. The simplest form of hot gas soldering equipment is the hot air pencils that are used for SMT rework and repairs. For this type of work, the entire board is often placed on a hot plate or over a hot air 'bath' so that the entire board is at an elevated temperature. The hot air pencil is then used to reflow solder individual components on the top side of the board. Hot air rework stations are available at reasonably low prices. Basic units cost less than £100. Ordinary hot air guns of the type sold at DIY stores are not ideal for SMT repair and reworking because the air temperature it too high and the large nozzle makes it impossible to heat a small area of the board without reflowing the entire board. However, this type of hot air gun can be used to reflow solder a complete new PCB. Most guns have a high and low setting. The low setting is typically about 300°C which is still a bit too hot. The board temperature can be controlled by keeping the gun at a suitable distance from the board. There are no absolute rules to determine the correct distance. Different guns will have different temperatures, flow rates, nozzle velocity etc. Some practice with a suitable thermometer and a few scrap SMT boards will help you to find the ideal settings.

It is just about possible to draw a track layout for some of the larger SM components by hand. For more complicated layouts using SM IC's, computer CAD software is the only practical way of producing the track layout. All modern PCB CAD software has libraries of SM component footprints. In fact, most CAD software has better support for SM components than through-hole components because SMT has been the most widely used construction method over the last couple of decades. **Fig 25.43** shows a simple PCB layout for a microphone amplifier and active LPF for use in a home made SSB transmitter.

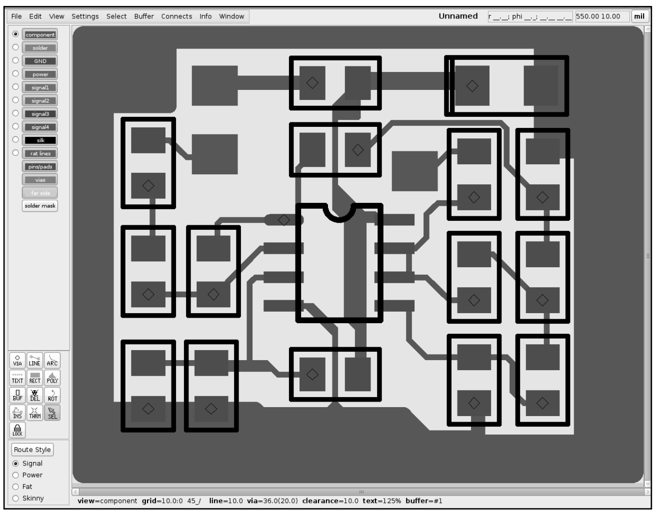


Fig 25.43: A simple PCB layout for parts of a home made SSB transmitter produced using a CAD program

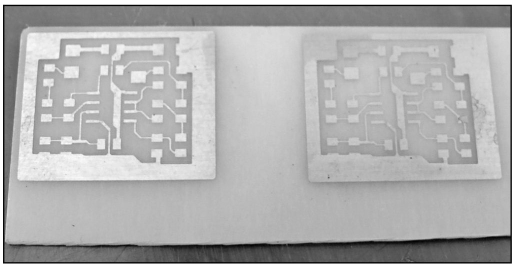


Fig 25.44: The printed circuit after etching

I will build two of these amplifier modules to demonstrate some of the reflow soldering techniques described above.

The layout was drawn using PCB [3] running on a Linux PC. The artwork was printed onto a sheet of Stephens transparency film using my HP D5160 inkjet printer. Two identical copies of the artwork were printed on a single A4 sheet. The sheet was then cut in half so that the two track patterns could be stacked to make a more opaque master artwork. This was used as the master artwork for the photographic PCB process which is described in detail earlier in this chapter. The artwork and pre-coated photo-sensitive PCB were clamped in my exposure frame and exposed to the halogen lamp for exactly ten minutes. After exposure, the PCB was developed in sodium metasilicate photo-resist developer, rinsed with clean water and then etched in a ferric chloride bath. Fig 25.44 shows the PCB immediately after etching, before the photo-resist was removed.

After etching, the two separate PCB patterns were carefully cut out using a hacksaw. Next the photo-resist was removed using fine steel wool. The signal track width used in this board is only 10 mils (0.254mm) so don't scrub too hard with the steel wool or you might lose some of the tracks. The use of powerful solvents like paint stripper to remove the resist is best avoided because the solvent may undermine the thin copper tracks and destroy the adhesive bond between the copper foil and the board substrate. An alternative way of removing the photo-resist is to re-expose the board without using the artwork as a mask and then place the board back in the developer tray for a few minutes.

There are several ways of applying the solder/flux paste to the PCB pads. Unless special equipment is available, the best way is to use a small pointed tool to apply a small amount of paste to each component pad. Alternatively, you can use one of the special syringes that are sold for this purpose. Solder paste supplied in syringes is usually water based and needs to be stored in a refrigerator. This is rather inconvenient for the occasional constructor who keeps solder in storage for long periods of time. This type of paste is relatively expensive and it needs to be shipped on-ice from the supplier using overnight delivery. In this part of the world, 'overnight' usually means several days. The other thing that this paste has in common with a choc-ice, is that it has an expiry date printed on the container. Solder paste is also sold in small plastic tubs. This type of paste doesn't usually have an expiry date or any special storage requirements. However, this paste is too thick to use with a syringe type of dispenser. Both lead free and tin/lead pastes are readily available.

I use Lodestar type L309 paste. This paste can be applied to the PCB using a small pointed tool like a toothpick or cocktail stick. Fig 25.45 shows how the solder paste was applied to one of the microphone amplifier PCBs using a cocktail stick. The amount of paste required for each pad is very small. A 50g tub of paste will last for a very long time.

After the paste was applied, I used small metal tweezers to place the components on the board and another wooden cocktail stick to push the components down into the paste and poke them into correct alignment with the pads. A good illuminated magnifier is very useful for this kind of work (Fig 25.46).

For those who only have experience of through-hole PCB soldering, the next stage is really the fun part. The soldering process only takes a couple of minutes. The whole board is soldered in a single operation and you won't have to worry about dirty oxidised iron tips, dry joints, solder bridges etc.

The board was mounted in a metal 'helping hands' type of holder and placed at the edge of the bench so that I would have free access to the top and bottom of the board. I am a great advocate of 'kitchen table technology', but this is a job for a rough wooden or metal bench.

The hot air from the heat gun is likely to damage a more delicate surface. I used a standard DIY hot air gun on the low heat setting. The high setting of 500°C - 480L/min would just toast the board and blow the components away! The low setting gives an air temperature of 300°C and a flow rate of 280L/min. This is still too hot, so don't allow the nozzle to get too close to the PCB.

My exact procedure is as follows:

1 Heat the underside of the board with the gun at a distance of 0.3 to 0.4m until the board reaches a temperature of 150°C. I used a heat resistant thermocouple and digital multimeter as a thermometer. Even at this distance, it takes less than one minute to reach the desired temperature. This is a bit faster than the recommended temperature ramp rate, but it works well in practice.

Fig 25.45: Applying the solder paste to one of the PCBs using a cocktail stick

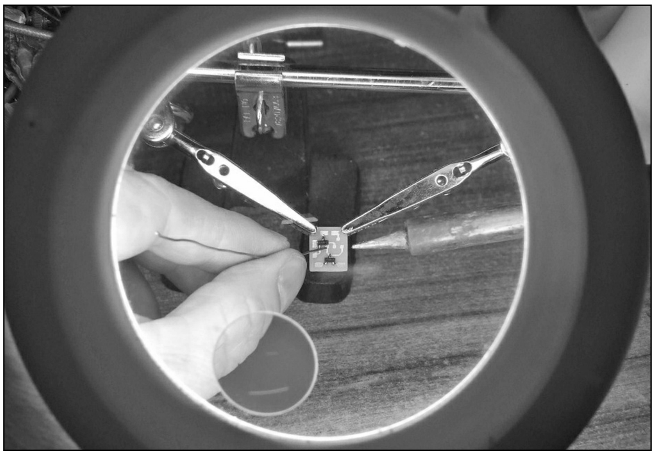


Fig 25.46: The job is made much easier by using a large illuminated magnifier and a jig to hold the work

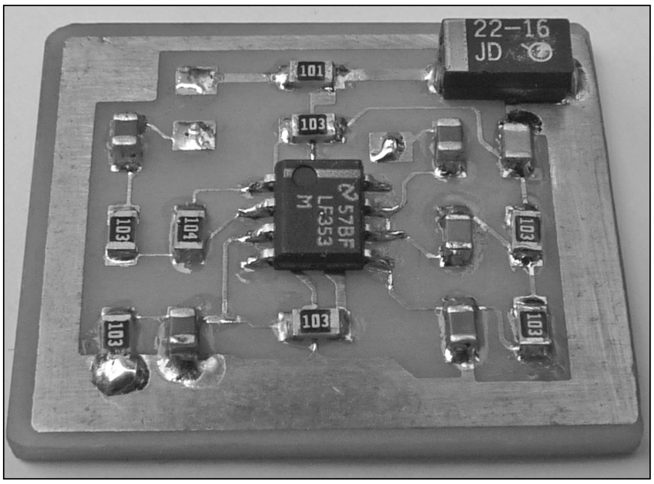


Fig 25.47: (above) Close-up view of one of the boards

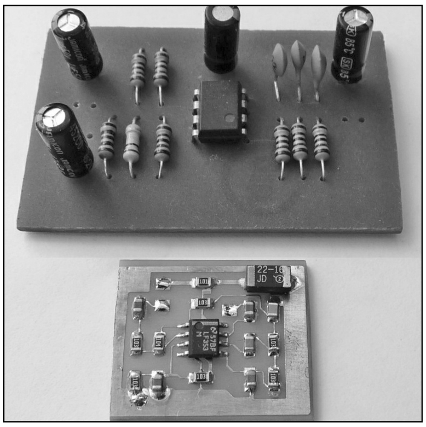


Fig 25.48: Finished SMT board compared with a DIP version

2 Quickly move the gun to the top side of the board and continue to heat the board until the temperature reaches about 200°C. This is the critical phase of the operation. Too much air will tend to dislocate the components when the solder flows. Too much heat will damage the components or the laminate. Move the gun closer to the board (about 10-20cm). You should find that the solder paste begins to flow after a few seconds. Keep the heat at this level until all of the solder paste has melted.
3 A few seconds after full reflow, withdraw the heat gun and allow the board to cool.
4 Check the board carefully under a strong light using a magnifier. I have found that this technique rarely produces solder bridges. If you do find any bridges, they can be removed using a hot iron and desoldering braid. Any dry or 'pasty' joints can be reflowed using a fine pointed soldering tip.

Both of the amplifier boards turned out quite well. No solder bridges were found. Both ICs were perfectly centred on the pads with good solder connections at the pins. One of the boards had a couple of 'pasty' looking joints which indicates that I didn't heat the entire board to a uniform temperature or the reflow zone temperature was a little too low. It only took a few seconds to rework these joints using a soldering iron. Fig 25.47 shows a close-up view of one of the boards. Fig 25.48 shows a finished board compared to a through-hole version of the same circuit using a DIP package.

The cosmetic appearance of the boards is reasonably good. A close examination of the board shows that there is a small amount of visible flux residue around the joints. As this type of paste uses an non-corrosive 'no-clean' flux, this is of no great consequence and can be left on the board.

Hand soldering SMT ICs.

It is also possible to solder SM ICs using an ordinary soldering iron. There are several techniques in common use. You can solder the pins one-by-one using a fine tip and very fine cored solder. This is reasonably easy to do for SOIC packages, but it is extremely difficult for smaller devices with very close pin spacing. One approach that is popular with radio amateurs is to tack a few corner pins to keep the IC in place, then solder all of the remaining pins without worrying too much about solder bridges. The basic idea is to lash on excessive amounts of solder and flux during the initial soldering stage. The excess solder and any solder bridges are then removed using desoldering braid. This technique can produce excellent results, even with high pin density packages.

A similar technique called 'drag soldering' can be used to solder high pin density ICs. The board is cleaned and then fluxed using SMT flux from a syringe or a flux pen. I use a CircuitWorks CW8100 no-clean flux dispensing pen (Maplin N63AA). As with the previous method, a few corner pins of the IC are tacked with solder to keep the IC in place. Make sure that all of the IC pins are sitting properly on the PCB pads before you start the final soldering. Using a relatively large and flat iron tip and copious amounts of flux, solder the IC pins, several at a time while gradually dragging the tip along the IC pins using cored solder. This is not quite as difficult as it sounds because surface tension will tend to draw the solder to the pads and IC pins. The hot iron will easily flow the solder from pin-to-pin so that excess solder tends to disappear and solder bridges are avoided.

Once soldering is complete, do a visual check and remove any bridges using desoldering braid. The drag soldering stage of the operation is easier if the board is mounted vertically during soldering. Start soldering at the top of each row of pins so that gravity will assist the flow of solder as you move the iron across the pins. This is known as 'vertical drag soldering'.

For quad packages with pins on four sides, solder the two opposing vertical sides first and then rotate the board 90 degrees before soldering the remaining two sides. YouTube viewers can get a close up view of the process by searching for "SMT soldering" or "drag soldering". Be warned! Watching the experts at drag soldering is a lot like watching the TV Chef cutting a tomato into ten thin slices in a couple of seconds. Trying to repeat this performance with a blunt knife in the EI9GQ kitchen tends to result in a messy pile of tomato purée. However, with a little bit of practice, drag soldering techniques are not too difficult to master.

Fig 25.49 shows a drag soldered IC on a commercially made PCB. The IC was removed from the board using a hot air pen. The PCB pads were cleaned up using a hot iron and desoldering braid. The IC was then re-soldered to the PCB using only a flux pen, soldering iron and cored solder. The following statement might seem counter-intuitive but a fine pointed soldering iron tip is not the correct tool for drag soldering. I normally use

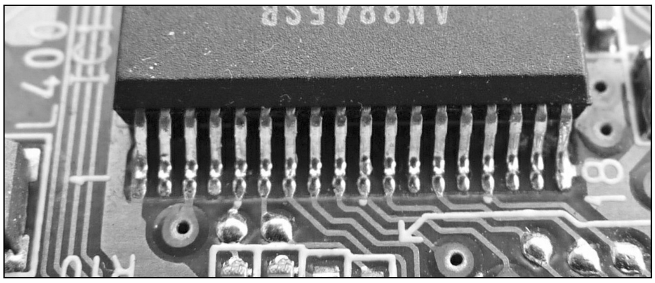


Fig 25.49: A drag soldered IC

a temperature-controlled iron with a pointed tip for most soldering work. My preferred tool for drag soldering SM ICs is an ordinary Antex 25W iron with a standard flat tip. During drag soldering, several IC pins are soldered at once. The flat iron tip lies across the top of the pins and should never intrude into the gap between pins. Use of a pointed tip will slow down the soldering process and is likely to lead to solder bridges. The key to success is to use plenty of flux. Only a small amount of solder is required. If excessive solder is used, it will tend to gather at the end of each row of pins so that it is very easy to remove with desoldering braid or some ordinary stranded copper wire. The flux residue can be removed using PCB cleaner or most alcohol based cleaners.

Rework and repairs

The ability to remove and replace SM components is of great importance to the the amateur constructor. A few simple SMT reworking tools allows the amateur to repair or modify existing home made or commercial equipment. The usual goal is to remove and replace a single SM device without damaging or disturbing the surrounding components. There are several methods in common use. Small chip capacitors and resistors are relatively easy to remove and replace. There are special iron tips for desoldering such devices. Even with a standard type of tip, anyone with a keen eye and a quick hand can flow the solder at both ends of the device and slide it off it's PCB pads. Removing SM ICs is a little more difficult. There are special desoldering tools which will fit over the pins of most types of IC package, but you will need a different head for each package type you are likely to encounter. Hot air reflow is the most popular method of SMT reworking and repair. The general purpose heat gun as used for our earlier experiment is a rather blunt instrument for this kind of work. A small temperature controlled heat pencil is a much more suitable tool. A heat pencil has a small nozzle which can focus the hot air on one individual component rather than heating the entire board. It is a good idea to pre-heat the entire board from below using an air-bath or hot plate heater. This reduces the time that it takes for the air pencil to reflow the top of the board. It also reduces the potential for stress and damage that could result from heating one small area of a cold board. Localised expansion and contraction can lead to cracking of the very hard ceramics used for chip devices. In extreme cases, it could even result in a warped PCB.

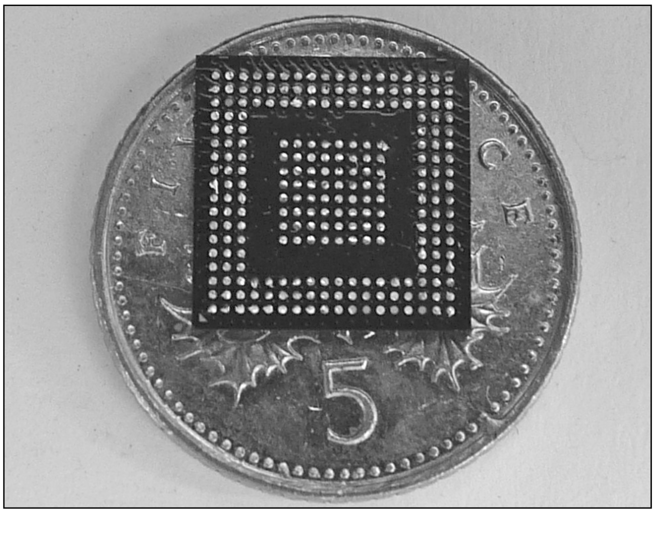


Fig 25.50: The bottom of a BGA style IC from a modern mobile phone. It has 244 connections, including the array of 64 pins in the centre used for conducting heat

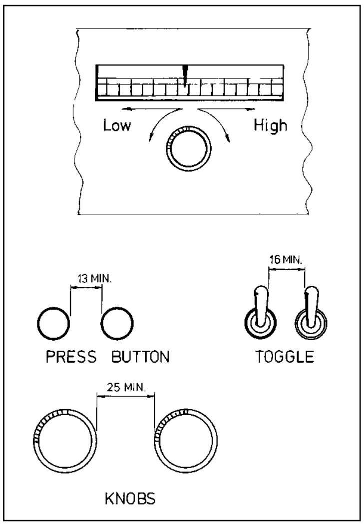


Fig 25.51: Suggested ergonomic strategy for some controls

Some of the newer IC package types will present an interesting challenge for the home constructor. Many of these devices have no connections at the edge of the package. Electrical connections are made via an array of pads on the underside of the package. These pads are reflow soldered to their corresponding pads on the PCB during PCB assembly. One such package type is the BGA or Ball Grid Array. BGA devices have an array of solder balls on the bottom of the chip carrier package. The solder balls flow onto a matching array of PCB pads during reflow soldering. This type of chip can achieve a much higher pin density than conventional IC packages. New BGA devices are supplied pre-balled and ready for soldering.

Reworking or repairing BGA circuits without using new replacement devices is tricky because of the need to re-ball the chip before it can replaced on the board. Fig 25.50 shows the bottom of a BGA style IC which was removed from a modern mobile phone. This is a 1cm square package with 244 connections hidden under the chip carrier. The array of 64 pins in the centre is used for conducting heat away from the chip core, leaving a mere 180 pins for normal electrical connections. A quad style chip with pins on four sides would need to have more than 150 pins to the inch (6pins/mm) to achieve the same pin density.

DESIGN THOUGHTS

Home construction allows individual ideas to be designed into the project being made. With commercial equipment 'you pays your money' but you don't always get exactly what you would like. Some controls you find unnecessary and others which you want are not there or are hidden inside the case. These factors are often overlooked until you make something for yourself and then aspects of commercial designs take on a different meaning. The quality or lack of it becomes apparent and it is very difficult to acquire this appraisal ability without having made, or having tried to make, something for yourself.

Over the years there has become an awareness of the value of ergonomics (see Fig 25.51). Most of us now know that controls should be positioned according to their purpose and that this purpose should be self-evident from either the type of control or the area in which it is placed. For example, most transceiver tuning knobs appear adjacent to the tuning display and, when rotated clockwise, the frequency increases. Usually, it is evident by looking at most transceivers which is the tuning knob.

Unfortunately the same cannot be said about press buttons and other controls. The use of press buttons needs great care, especially where accidental operation could cause damage. If correctly designed, it should be impossible to accidentally press two or more buttons at the same time. Also, if such an action is done accidentally or deliberately, it should not destroy the device, or send it into an unusual operating or locked-up state. Ideally, any button-type control should be accompanied by an indicator which shows what mode the button is in. Power supply switches are particularly important in this respect, especially

Safety recommendations for the amateur radio workshop

1. All equipment should be controlled by one master switch, the position of which should be well known to others in the house or club.

2. All equipment should be properly connected to a good and permanent earth (but see the box about PME in the chapter on Practical HF Antennas, and Note A).

3. Wiring should be adequately insulated, especially where voltages greater than 500V are used. Terminals should be suitably protected.

4. Transformers operating at more than 100V RMS should be fitted with an earthed screen between the primary and secondary windings or have them in separate slots in the bobbin.

5. Capacitors of more than 0.01μF capacitance operating in power packs etc (other than for RF bypass or coupling) should have a bleeder resistor connected directly across their terminals. The value of the bleeder resistor should be low enough to ensure rapid discharge. A value of 1/C megohms (where C is in microfarads) is recommended. The use of earthed probe leads for discharging capacitors in case the bleeder resistor is defective is also recommended. (Note B). Low-leakage capacitors, such as paper and oil-filled types, should be stored with their terminals short-circuited to prevent static charging.

6. Indicator lamps should be installed showing that the equipment is live. These should be clearly visible at the operating and test position. Faulty indicator lamps should be replaced immediately. Gas-filled (neon) lamps and LEDs are more reliable than filament types.

7. Double-pole switches should be used for breaking mains circuits on equipment. Fuses of correct rating should be connected to the equipment side of each switch in the live lead only. (Note C.) Always switch off before changing a fuse.

8. In metal-enclosed equipment install primary circuit breakers, such as micro-switches, which operate when the door or lid is opened. Check their operation frequently.

9. Test prods and test lamps should be of the insulated pattern.

10. A rubber mat should be used when the equipment is installed on a floor that is likely to become damp.

11. Switch off before making any adjustments. If adjustments must be made while the equipment is live, use one hand only and keep the other in your pocket. Never attempt two-handed work without switching off first. Use good-quality insulated tools for adjustments.

12. Do not wear headphones while making internal adjustments on live equipment.

13. Ensure that the metal cases of microphones, Morse keys etc are properly connected to the chassis.

14. Do not use meters with metal zero-adjusting screws in high-voltage circuits. Beware of live shafts projecting through panels, particularly when metal grub screws are used in control knobs.

15. Certain chemicals occur in electronic devices which are harmful. Notable amongst these are the polychlorinated biphenyls (PCBs) which have been used in the past to fill transformers and high-voltage capacitors and beryllium oxide (BeO) which is used as an insulator inside the case of some high-power semiconductors. In the case of PCBs, the names to look out for on capacitors are: ARACLOR, PYROCHLOR, PYRANOL, ASBESTOL, NO-FLAMOL, SAF-T-KUL and others [3]. If one of these is present in a device, it must be disposed of carefully. The local Health and Safety Authority will advise. In the case of beryllium oxide, the simple rule is DON'T OPEN ANY DEVICE THAT MAY CONTAIN IT.

Note A. - Owing to the common use of plastic water main and sections of plastic pipe in effecting repairs, it is no longer safe to assume that a mains water pipe is effectively connected to earth. Steps must be taken, therefore, to ensure that the earth connection is of sufficiently low resistance to provide safety in the event of a fault. Checks should be made whenever repairs are made to the mains water system in the building.

Note B. - A 'wandering earth lead' or an 'insulated earthed probe lead' is an insulated lead permanently connected via a high-power 1kΩ resistor or a 15W 250V lamp at one end to the chassis of the equipment; at the other end a suitable length of bare wire with an insulated handle is provided for touch contacting the high-potential terminals to be discharged.

Note C. - Where necessary, surge-proof fuses can be used.

with portable equipment, where it is all too easy to switch it on and not be aware of this fact.

Rotary switches are less susceptible to accidental switching but they must be positioned and have suitable style knobs to facilitate switching. Most rotary switches occupy more space than push buttons and it is this which usually restricts their use on the compact rigs of today. Rotary switches are ideal for such controls as mode and band selection. One look at the switch position shows immediately what mode or band is selected. It is impossible to attempt to select two modes or two bands and there is no need for any other form of indicator: a power-saving factor also.

Toggle and slide switches are an equally effective form of self-indicating switches but toggle switches are prone to accidental operation. Potentiometric rotary controls, such as AF volume, Morse speed and RF gain etc should all be self-indicating. Also, clockwise rotation should increase the function. Rotary concentric controls should give each control a related and easily identified function. Knob styles can also affect the quality of presentation of the finished work. Unfortunately, knobs are expensive but ex-equipment knobs can often be used as a cheaper alternative. The size of knob should relate to the accuracy required from the function it controls. For example, a tuning knob would not feel right if it was less than say 40mm diameter, but the same diameter for a volume control would be unsuitable and usually unnecessary. Slider controls are very good, especially when simultaneous operation is required on such as faders and mixers, but this facility is not usually required on radio equipment. These controls also take up space if they are to be accessed easily. Some commercial amateur radio equipment uses the smaller type of this control for the less-utilised or preset controls.

Many more factors can be found, and it is discovering these which adds to the pleasure of home construction

REFERENCES

[1] 'Homebrew', *RadCom*, October 2006
[2] http://qucs.sourceforge.net/
[3] http://pcb.gpleda.org/
[4] 'Homebrew', *RadCom*, July and August 2006

About the Author

***Eamon Skelton** has been licensed as EI9GQ since 1987. He has never owned a commercially made rig and has been active on most amateur bands from 1.8MHz to 10GHz using only home made equipment. Radcom 'Homebrew' columnist since January 2006. He has had articles published in various other publications including* Elektor, Radio, JARL. *Author (webmaster) of the EI9GQ Homebrew Radio Page, http://homepage.eircom.net/~ei9gq/*