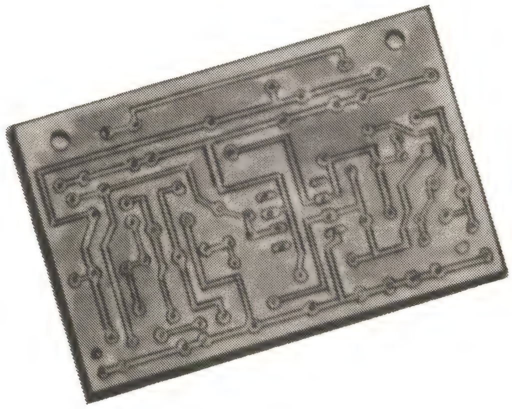
Special Feature

MAKING YOUR OWN P.C.B.s



ANDY FLIND

Printed Circuit Board Making on a Budget

Unmasking some of the pitfalls that may be encountered when producing your own p.c.b.s.

SOONER or later, most serious electronics enthusiasts will want to make a printed circuit board, "p.c.b." for short, for themselves. Either a track pattern will be available but not the ready-made board, or it might be that a circuit has been designed from scratch and is considered worthy of a "proper" board rather than stripboard or tagstrip construction.

There are many ways to make a p.c.b. and over the years the author has tried most of them. As there are pitfalls for the unwary and often little advice on how to go about the task without expensive equipment, it seems worthwhile to pass on some of the experience gained.

Be Sensitive

Undoubtedly the best method for producing a p.c.b. is photographic, using a board coated with a photosensitive "resist" material. This is exposed to ultraviolet light shining through a track pattern on a transparent sheet, then developed and placed in etching fluid to dissolve away unwanted copper to leave the desired tracks. The resist is then removed and the board is drilled and trimmed to size ready for component assembly.

This method produces neat results, extra boards can be produced easily and, where necessary, minor layout alterations are usually simple to make.

At this point, readers may be wondering about the cost of a UV Lightbox and tanks for the chemicals, but in fact a lightbox can be built for a quite modest outlay and the chemistry can be carried out at the kitchen sink. There is definitely no need to shell out for a full production lab to produce the odd board for hobby use.

Making a Master

The first task in making a p.c.b. is to produce the transparent master. To do this by hand, a good method is to begin by drawing up a layout on 0·1 inch squared graph paper. This paper is difficult to find in pads and often expensive, but large sheets can be bought cheaply in W.H.Smith and cut into conveniently sized pieces.

A soft 0·5mm pencil and pencil-shaped eraser are recommended for initial layout work as these make alterations as simple as they would be on a computer. This part can be done in comfort in front of the TV!

Once the layout is finalised, a transparent "master" is required. Stationers and photocopying shops can provide clear acetate sheets, and one of these may be taped in place over the layout and the track pattern laid out using either transfers or adhesive tapes and symbols. Using adhesive tapes is much preferred where this method has to be used.

Computer Aided

Where a computer is available, life becomes much easier. Many professional printed circuit packages are available (*see advertisements in this issue*) offering various extras such as schematic capture, autorouting, design rule checking and so forth. Often these features are more suitable for rapid prototyping than really good finished designs but the basic layout routines will always produce *clean artwork* and this can be done rapidly from a layout drawn up on paper.

Computer Aided Design "CAD" type drawing packages can also produce track patterns. For this work the program should have "layers" and "symbol libraries", and, of course, must be able to draw lines of widely varying width.

Layers are useful for generating different views of the board, with components on one layer, pads on another, tracks on a third and so on. These views can be turned on or off as necessary for layout and printing purposes. Symbols can be designed to represent the components used, various sizes and shapes of pad, and patterns of pads for i.c.s etc.

If any readers have Acorn Archimedes computers, the "Vector" art package sold by "4-mation" is highly recommended. This is similar to Acorn's "Draw", but has many extra features including layers and symbol libraries. Prior to the acquisition of a PC the author used this software for several years.

Computer generated artwork will need conversion into a transparent master. It might be possible to print direct onto transparency at 1:1 scale, but the author's attempts to do this with an ink-jet have so far proved unsuccessful. A good quality printout is not an absolute essential, inexpensive inkjets can do an excellent job and even an 8-pin dot matrix may be used if the output is scaled up and then correspondingly reduced on a photocopier.

The printed copy can be reduced and transferred to acetate transparency in one go, usually at a cost of around 50p a sheet. Most copiers can reduce to 70 per cent in one go, so the author's usual practice is to print at the inverse of this, 1.49:1, and then ask for a 70 per cent reduction straight onto acetate at the copyshop. This easily handles detail such as tracks between i.c. pins.

Shortcomings

This is a good time to discuss some photocopier shortcomings. To start with, photocopies are not usually dense enough for direct use in a UV exposure process. When held up to a light the black areas usually appear slightly "porous".

This is often worse in large black areas, which is a good reason for avoiding these in the layout. However, two copies carefully taped together nearly always achieve adequate density.

Another photocopier problem is that they don't always achieve the exact print size expected. Usually, at least for small boards, this is not a problem, but it's worth taking along a 1:1 printout for checking. A similar problem is acetate copies turning out at slightly different sizes, the difference being tiny but sufficient to prevent the holes in the pads lining up when they are taped together.

The cause of this is probably expansion of the acetate sheet; by the time the second sheet passes through the copier will have warmed up so it will receive the image after being heated, and expanding, more than the first. A recent experiment was to ask for three copies instead of the usual two and, sure enough, two were a greatly improved match.

Right Exposure

Having produced the transparency, the next stage in the process is exposure. Pre-coated photosensitive board is available from most electronic suppliers and is easy to use. The instructions supplied by a major retailer used to suggest that exposure could be carried out with incandescent light.

Don't bother trying this! The author once did, using a 500-watt bulb and an exposure of about fifteen minutes. The results were very fuzzy, certainly not good enough to warrant any further use of this method.

Ultraviolet is essential for the exposure. However, it is certainly not necessary to shell out seventy quid or so for a lightbox as this can be constructed for little more than the cost of the tubes, starters and a choke. Currently the cheapest source of tubes is probably Electromail, who offer a kit consisting of two 8-watt replacement tubes with starters for their small lightbox at a very reasonable cost.

Constructors can design their own exposure unit using these, the essential dimensions are shown in Fig. 1. Connections for the components are shown in Fig. 2, where it will be seen that only one choke is required to supply both tubes. Glass can usually be cut very accurately to size for very little cost by a local glazier.

The exposure time required varies with different makes of board but is usually somewhere between two and six minutes. Since the board is fairly expensive, it's worth trying some small test pieces to determine the optimum time before committing a larger and more expensive bit.

Chemical Works

Once the board has been exposed, it's time to open the chemical works! This consists of developing the photosensitive coating so that the exposed portion dissolves away and the remainder is "fixed", remaining to protect the copper beneath it during etching.

It's possible to develop using dissolved caustic soda crystals, but this appears to be very temperature dependant so a proprietary developer is preferable for the job. This is cheap and re-useable, and gives much more consistent results. It's important to allow enough time for all the unwanted resist to dissolve, otherwise a thin film may be left which can hinder the etching.

Commercial tanks are available for developing and etching, especially the latter where the solution is often heated and agitated by bubbling air past the immersed board. Again, this is not essential, the processes can be carried out in literally any glass or plastic container that will hold the chemicals and the board. The lack of bubble agitation simply means the process will take slightly longer – but a cheap aquarium air pump could be used for the task.

As with developing, ferric crystals are available for making up etching fluid, but the preferred and most convenient method is to use ready-mixed fluid. This lasts for ages before having to be discarded, so the

UV LIGHTBOX CONSTRUCTION

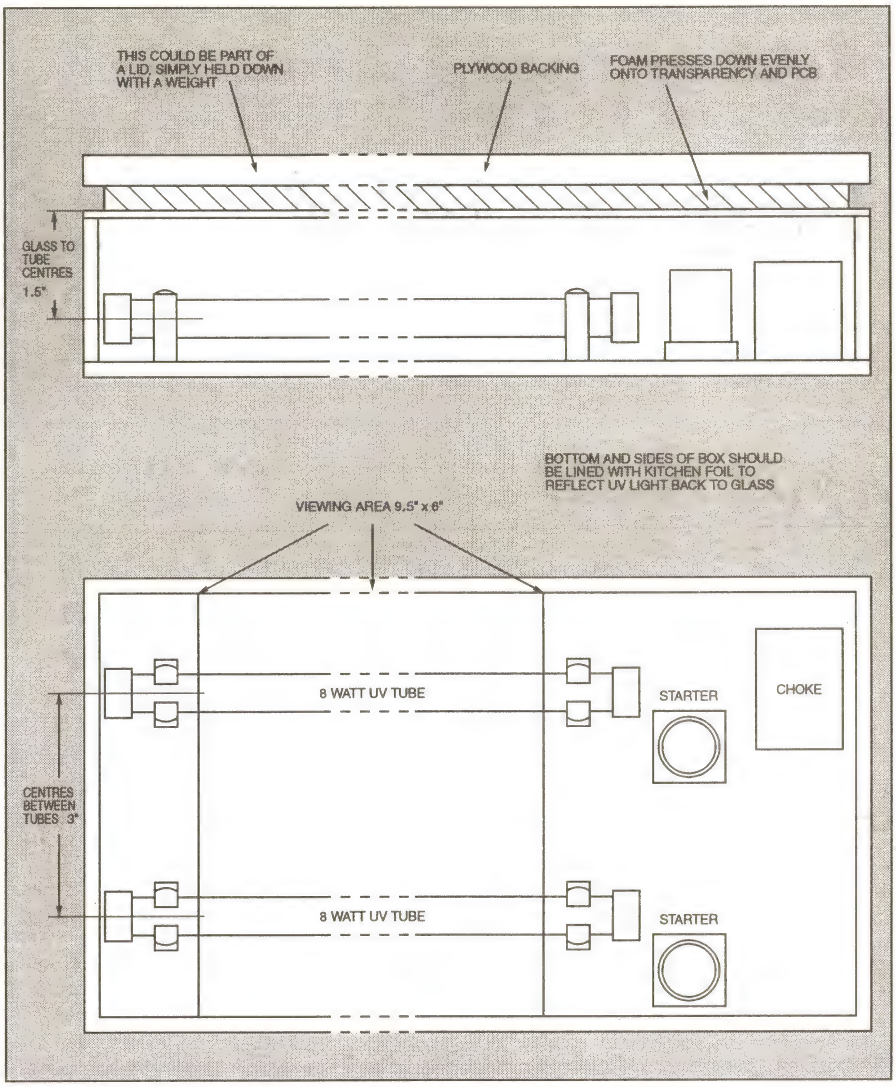


Fig. 1. Dimensions and layout suggestion for a two-tube UV Lightbox.

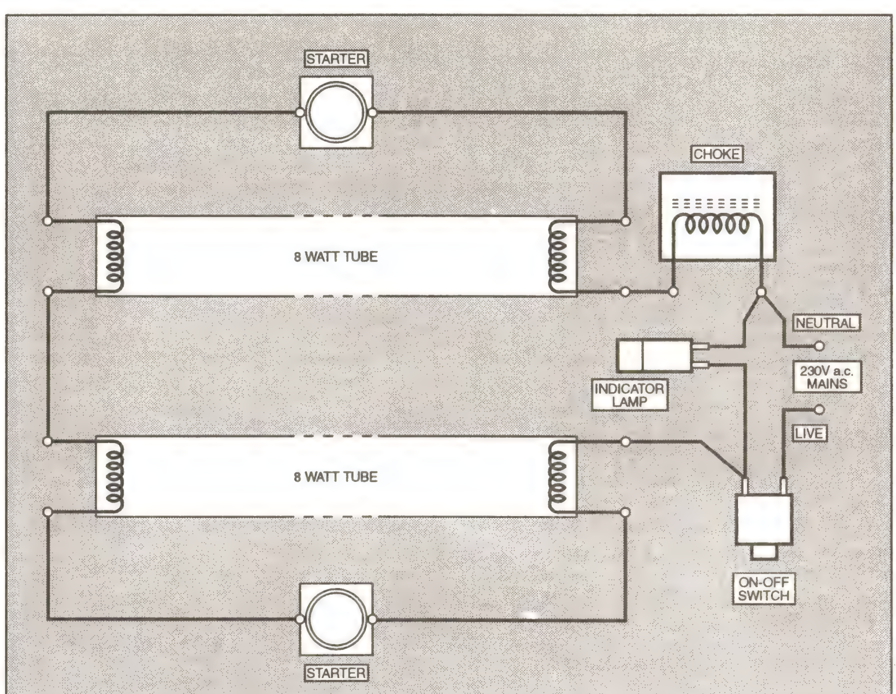


Fig. 2. Interwiring between tubes, starters, choke, mains switch and indicator lamp. ***Note that the unit should be wired with a switch that only allows the tubes to be turned on when the lid is closed. UV can damage eyes.***

small extra cost is not an important consideration.

In Suspension

During etching, the solution becomes heavier as copper dissolves into it so the board should be arranged to allow the fluid to fall away from it as this happens. Small boards can be stood on edge in a jar of fluid, larger ones can be supported face down in a flat tray.

The author's latest method is to make a support by twisting single-core plastic covered wire round the board as shown in Fig. 3, which holds it in the correct position and provides a convenient "handle" for removal. An extension of this would be to support the board vertically with a wire loop that would hook over the edge of a jar, keeping it just clear of the gunge that inevitably forms in the bottom of such a jar.

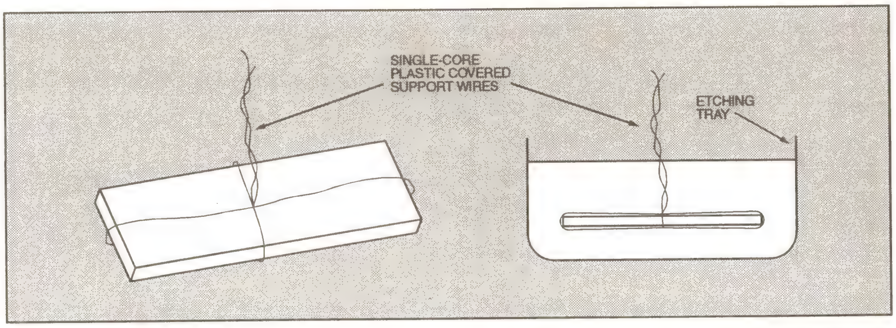


Fig. 3. Supporting a printed circuit board for etching.

Many domestic chemicals such as disinfectants, cleaners etc are now supplied in plastic bottles that are oval rather than round in section, the bottom half of one of these would make an ideal small etching tank.

Etching time depends on the state of the fluid and on temperature. Heat speeds the process considerably so it is worth placing the tray in a sink of hot water to warm it. Don't make the mistake once made by the author of placing a glass dish over a low gas flame though – the heat cracked the dish and the etching fluid did the cooker no good at all!

Polished Job

Following etching, the resist must be removed from the remaining copper tracks. Special chemicals are sold for this, but a wire-wool pad does the job just as well and imparts a wonderful polish to the copper beneath, leaving it in ideal condition for soldering. Thorough washing and rinsing is essential to remove every trace of etching fluid.

The chemicals used are not so much dangerous as unpleasant. The developer is caustic; if it comes into contact with the skin it will give a nasty "soapy" feel that persists even after thorough washing. ***Eye contact should be avoided at all cost.***

The etching fluid will permanently stain almost anything it comes into contact with and corrode just about any type of metal. It even appears capable of producing "rust" spots on stainless steel.

However, contact with the fingers seems to produce nasty staining rather than instant burns. The rule for handling both must be to wear rubber gloves and handle with great care to avoid splashes and spillage. Eye protection in case of splashes is also advisable.

Right Drill

Finally, some notes on drilling the board. In general it is very difficult to drill a p.c.b. using a hand-held drill. It certainly can't be done with the Black and Decker, though one of the mini-drills used for fine modelling work might do.

The ideal is a mini-drill with a suitable stand. They usually require an amp or two of d.c. supply, but check the junkbox for a suitable transformer before buying a custom power supply. If one is to hand, all that is needed with it is a bridge rectifier so again useful savings can be made.

Hole size for component leads is usually 0·8mm or 1mm. Occasionally larger components, or items such as presets, will require 1·3mm. The author usually drills all the holes to 1mm and opens up any that prove too small during construction, using the mini-drill.

If the board is Paxolin there will be no problem with drill bit type but glass-fibre board, preferred by many constructors, blunts ordinary steel drills rapidly. Solid carbide drills last much longer but are very fragile. They are also far more expensive so it is very frustrating when one breaks.

However, don't let anyone tell you they are for machine-controlled drills only. With suitable care a 1mm solid carbide bit can be used with a mini-drill in a stand, and will remain sharp enough to cut clean, accurate holes for far longer.

Have A Go

One final tip concerns preservation of the board after construction. Commercially-produced boards are normally roller-tinned. This prevents oxidation indefinitely, so the board always solders well and looks clean. The bare copper of a home-produced board does not have this advantage and will oxidise quite rapidly.

For easy soldering construction should be carried out fairly soon after cleaning, within say, a day or two. A coating with a spray-on p.c.b. varnish will then keep it looking good almost indefinitely, and will help to reduce other problems such as insulation breakdown between tracks.

Hopefully, these hints and tips will encourage many more enthusiasts to "have a go" at making their own p.c.b.s to near professional standards. Who knows, in time we might even see some of the results in the pages of *EPE*. □

Fault Finding

Here is a fault-finding guide to help locate any problems that might arise when producing boards using the ultra-violet technique.

Develops O.K. but excessive exposure time required.
Artwork drafting film is too opaque – use appropriate clear/translucent film.
Aerosol resist coating too thick: – increase exposure time or use spray more sparingly.
UV tubes need replacing.

Very long developing time, or resist does not develop fully or at all.
Resist is underexposed to UV light: – increase exposure period according (only areas which have been thoroughly exposed to UV light will be developed).
Resist may also be too thick or artwork film too opaque, and has not been thoroughly activated by the UV light – see above.
Developer cold, weak or exhausted. Renew.

Image is developed but then disappears.
PCB has been over-exposed. Reduce exposure time. (Applies especially to good quality sensitive pre-coated boards.)
Artwork layout is not opaque to UV light, permitting UV to pass through transfers etc.
Board has been left in developer for too long.

All of the etch-resist disappears when developing.
Developer could be too hot or concentrated.
The whole board has previously been exposed to UV light. (e.g. sunlight).

Unwanted copper is left on the p.c.b. when etching.
Unwanted etch-resist left on board, due to under-exposure or under-development. Try removing excess resist with a cotton-bud dipped in acetone, then re-immerse in etchant.

Excessive etching time.
Etchant may be exhausted: – dispose of safely and renew.

Narrower tracks are etched away.
Over-exposure to UV may affect very narrow etch-resist tracks which will be removed/reduced in width by etchant.
Developer or etchant may be too strong, undermining narrow sections.

Copper pads lift off when soldering.
Excessive heat applied during soldering.
Diameter of pads too small.
Diameter of the drilled hole too large!
If conductor is intact, repair by applying Super Glue Gel to the affected area, to encapsulate the lifted pad in resin.

Breaks in copper track.
Flaws in etch-resist coating or artwork.
Extremely thin conductors may be undermined by excessive etching time, resulting in the tracks being etched through.
Repair with silver-loaded paint or solder in a jumper wire.