MAKING YOUR OWN PRINTED CIRCUIT BOARDS

Ian Thomas

The cheapest and most simple printed circuit board is the one you buy with or for your ETI project — ready made. With this procedure and a little practice you will find it easy to make boards for your own circuits.

IT'S ONE THING to read about all those 'you beaut' projects but quite another to actually build one up. Or perhaps you have ideas about what seems an excellent project but never find anything like it in the magazines (we do our best but . . .). The clever thing to do is to make it yourself. Making a complete printed circuit right from the original idea to the final unit isn't as complicated as it seems and all the necessary ingredients are readily available.

Breadboards

The place to start the whole business is the original diagram. If you haven't got a complete and detailed circuit diagram you aren't really ready to start making hardware and more development is needed. If you're starting from a magazine article then no problem, but if you're designing your own circuit then you must first try out your circuit design with what is politely called a "breadboard" (for reasons I've never really understood) or, less politely but more descriptively, a rat's nest.

Plug-in type breadboards can be bought from almost any supply house and vary considerably in both price and quality. A good hint if you're going to buy one is first to try all different component lead sizes in the breadboard holes to make sure they're held firmly by the sockets. The cheap and nasty ones tend to not hold the finer leads and will drive you crazy by falling out while you use them.

Once you've got your circuit working, draw the circuit from the breadboard, tracing out each component as you draw it. You'd be amazed how often things aren't exactly as you think they are! Even at this stage it may be desirable to do a little more work in refining the design.

When you're tinkering to get things working you may inadvertently do things that are inelegant (or downright wrong — you'd be amazed how easy it is to get things working only to raise a fierce blister on a transistor because you didn't notice it was dissipating a watt or so!).

After you're satisfied that the circuit diagram is what you've got and what you've got is what you want the next thing to do is redraw the diagram as neatly as possible. Make the lines between components as short as you can get them and have as few lines crossing over each other as possible. This isn't just good documentation; it forms the basis of the printed circuit layout.

If you start the board layout following a neat diagram the layout tends to go pretty easily but if you start with a mess the board will turn out worse! It's a lot easier to have lines crossing each other on paper than copper tracks on fibreglass.

The next step is to decide exactly where you want to mount the board so that you can see how much space you have. Now that you've got the circuit diagram you can make a list of all components and tally up the board area necessary to fit them. This list will normally consist of so many resistors (value doesn't matter), so many transistors, so many capacitors this size, so many that size and so on. Every component must be used **including the power supply bypass capacitors you left out.** That must be one of the most common sources of trouble in getting circuits to work. The final list will have every component in sections according to size.

How big a pc board?

Next you have to work out how much board area is needed for each type of component. A good rule of thumb is to allow 0.05 to 0.1 inch greater than the maximum size of the component. For example, your ordinary common or garden resistor has a body about 0.1″ in diameter and mounts on 0.4″ centre holes. The board area to allow for is therefore 0.15 x 0.5 or 0.075 square inches.

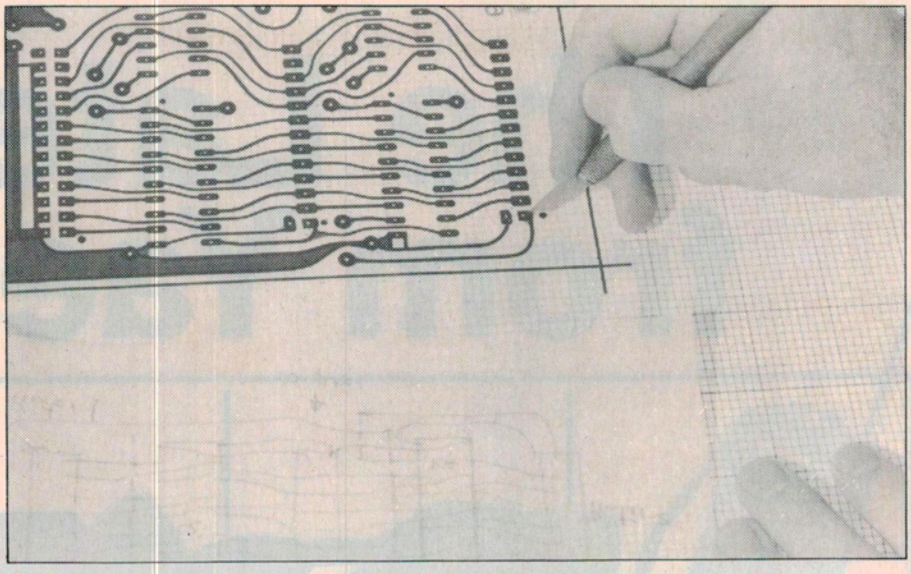
It isn't the object at this stage to allow for the space needed for interconnection but only to find the absolute minimum board area needed.

On your list tally up the areas of all the components and this is the board area you'll need if you could achieve a perfect layout. At this stage you must also consider how you intend to mount the board as mounting holes also use board area — make the mounting holes another item on your list.

Next compare the needed area with the area you have available in your box or whatever. A very good guide to the difficulty of the layout you will have to do is called the packing density of the board. The packing density is the ratio of the board area needed to the total available area usually given as a percentage. For example if your list says you need 4.82 square inches and the total available area is 3″ x 2″ then the packing density needed is

$$\frac{4.82}{3 \times 2} = 0.803 \text{ or } 80\%$$

This packing density could be done but would be rather difficult. As a general



Twice full size artwork being prepared. Note the tenth-inch grid and pencil rough.

rule less than 50% is pretty easy, less than 75% needs a little care, less than 90% needs a *lot* of fiddling and rearranging and greater than 90% means good luck!! (forget it).

If the packing density works out then you're in a position to start blocking out the board layout. Buy some tenth-inch grid graph paper (I know we've all supposed to have gone metric but *all* pin spacing is given in tenths of an inch and 0.1″ = 2.54 mm and the .04 mm can add up horribly) and draw in the outline of the final board you want.

If you're only going to make one board for yourself and you don't have too high a packing density the outline can be the same size as the final board. However if you want a really neat job then the outline should be exactly twice the size of the final board.

Artwork size

Doing the layout at twice full size means that any inaccuracies in the layout and the printed circuit artwork that comes from the layout are reduced to half size when the twice full size artwork is photographically reduced to normal size. This doesn't excuse rough work; it just makes finer detail work possible. Almost all commercial artworks are taped at at least two to one and sometimes four to one if special accuracy is required.

For the hobbyist there are two disadvantages in generating artwork at twice full size. The first is that the artwork has to be photographically reduced to final size after taping is completed. This means you have to take your artwork around to an industrial photographer (there are plenty around) who will photograph your masterpiece so the negative out of the camera is exactly the right final size.

If you're going to use precoated printed board material (more of this later) then this is all you need but if you want to put your own resist on the board then you must ask him to make a contact positive of the reduced negative.

Both these cost money but not as much as you may think. To give an example, the electronic scales that appeared in June and July ETI last year cost me $18 for reduction *and* positives. There was no way the artwork could have been done at full size so I had no choice. Even if they could have I would probably still have gone to twice full size originals.

Probably the greatest disadvantage to using twice full size is that you have to find time to go out and have the photography done. If you're beavering away building your technological masterpiece at 2 am on Sunday morning you probably won't want to stop for the weekend to get reductions done. If the artwork is full size you can carry on and the whole process can be done at home.

A second disadvantage with twice full size artwork is that you can't actually place the components on the layout to see if they'll fit. But if you haven't done a lot of artwork then single size is an invaluable way to make sure you aren't asking the impossible. Stencils can be bought with the outlines of most components twice full size on them to act as layout aids (Bishop Graphics Cat # EZ3367 & EZ3368) but it's not quite the same as the actual IC or transistor.

Whatever size you decide to use there are a few essential purchases you must make before you start. The first is the plastic draughting film that is used to make the artwork and the other is the artwork stick-on pads and tape to actually generate the artwork.

If you intend to use single size artwork then you'll need some 0.1″ diameter pads, a few 0.2″ pads for terminations, some 16 pin IC pads (get the 16 pin ones only and cut them down for 14 or 8 pin IC's) and a few different thickness tapes; say .040″ and 0.075″ for starters. The tape is specially made for artwork generation and is completely opaque to light. It can also be bent (it's sort of like black masking tape) to form curved tracks on the artwork but more of this later.

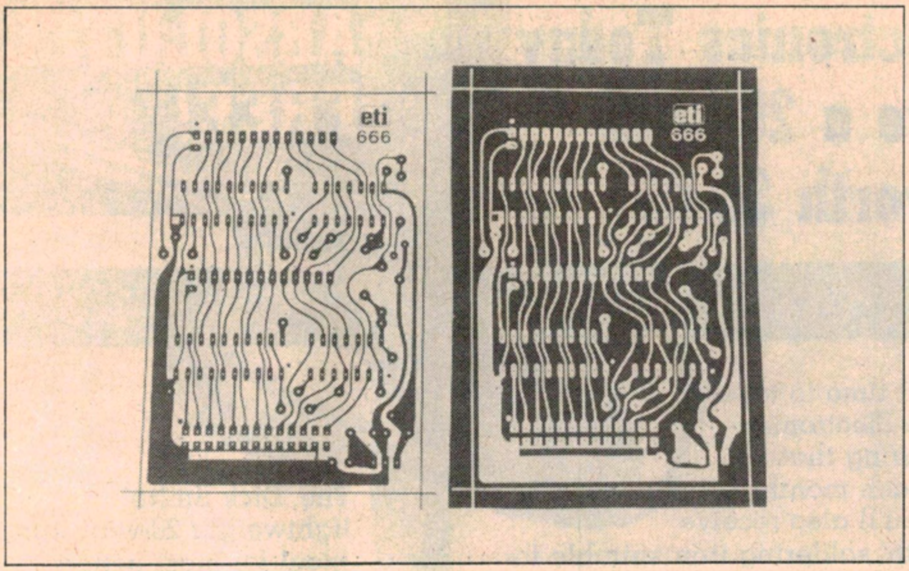
Get yourself a pencil, rubber (a good one as you'll be making a lot of changes) and all the components you intend to finally use and you're ready to start on the real layout. Even if you're doing twice full size artwork it's still essential to hold components on the graph paper to see how they fit sometimes.

Start by drawing in the terminal pads so all the leads come off the board in a nice neat group. Then start drawing in the components in more or less the same pattern as they appear in your neat circuit diagram. This means the components that connect to terminal pads lie near them.

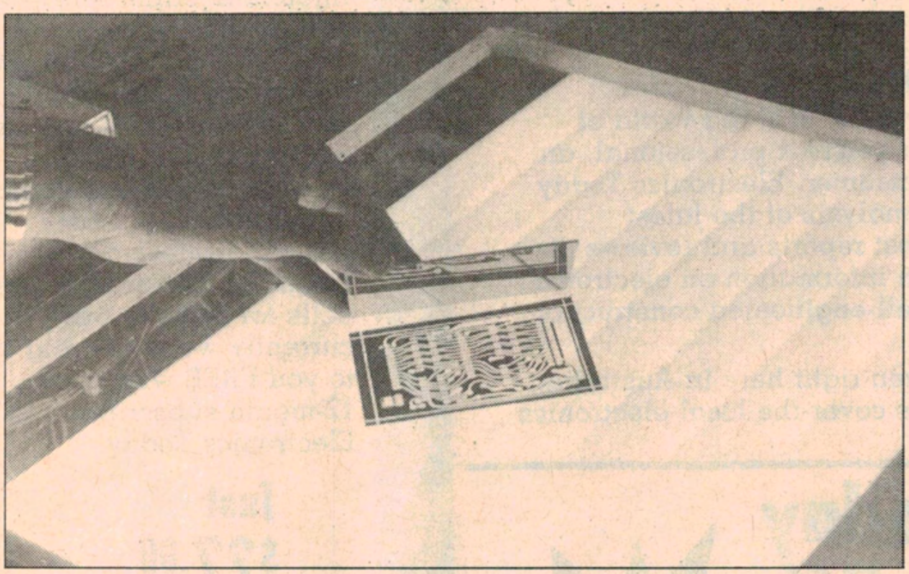
In order to make things look neat and professional there are a few golden rules in placing components. They are:

- ☐ All ICs must be oriented the same way with pin 1 pointing to the same corner of the board. If you turn some of the ICs 180° then it's a cert you (or someone else) will put the IC in back to front and destroy it! Never ever put ICs in at random 'convenient' angles — it looks like a mess and writes across your layout in words of fire "this was done by an *amateur*".
- ☐ Resistors should also be put in only parallel to the sides of the board for much the same reasons as above. Also if you orient them at odd angles then you're bound to waste space on the board. The same applies to capacitors and diodes.
- ☐ Transistors and vertically mounted capacitors can be oriented any way so long as their pins fall on the 0.1″ grid of the graph paper.

A way of getting things started is to mentally divide the circuit up into small blocks and then proceed to lay out the circuit block by block so the interconnections between the blocks fall next to each other. ▶



Positive or negative? You can tell that the one on the left is positive because the black on the film looks like the copper tracks on the board.



Exposing the resist. With precoated board, simply unwrap in dimly lit artificial light to prepare the surface for exposure.

When a block contains say four or five resistors it's often easy to draw them in side by side with the minimum 0.15″ spacing between them then connect them together. Usually the whole group can be connected without moving any component at all or at most the spacing between two resistors may have to be increased to 0.2″ to allow two tracks between the pads. The 0.15″ spacing only allows clearance for one track between the pads.

The transistor of IC that's associated with the group of resistors can be placed right next to them and connected easily. A strong word of warning must be given — always draw in the power supply lines as you go along and make sure that power supply bypass capacitors go in next to the ICs or transistors that need them. It's no good to try and bypass an IC supply line from half way across the board. A good rule is to put a 10 nF capacitor right next to *every* IC on the board or at the very least every three ICs. This probably adds a dollar or so to the cost of the board but is *much* cheaper than spending hours trying to get things to work when you have half a dozen little oscillators whirring away.

Another most important point is to make your earth track *big and short.* Even if your circuit is audio or only dc the transistors and ICs don't know this. They still have gain up to many megahertz and if your earthing and/or bypassing is no good they may oscillate.

In your neat circuit diagram you probably have one earth line drawn along the bottom of the page. Try and preserve this as one earth about 0.2″ wide on the board with all parts of the circuit that need it tied back to this track. All bypass capacitors should also connect directly to this track without long straggly connecting tracks. If you've sufficient space it's a good idea to have a border of earth about ½″ wide all around the board but that's often a luxury physical constraints won't allow.

Signal lines can be allowed to wander around a bit and power supply lines, so long as they're properly bypassed, can wander even more but the earth must not. Keep on adding in blocks of your circuit diagram until you've got it all in including the mandatory large electrolytic capacitor across the supply terminals.

Then, when you reckon it's all done have a look and see if you can reroute tracks to tidy things up. Once you've actually got the whole thing down on paper it's usually pretty easy to see changes that will improve it a lot.

If you want a really pukka job you'll almost certainly have to work the layout down a bit. Even the pros who do this sort of thing for a living still have to rework part of their layouts (apart from one bloke I know but he's a genius with a mental flair for visualising these things!).

Finally when you're really happy that every component on the board is connected up correctly and all the components are drawn in so they don't touch each other (or worse cross over each other!) you're ready to start making the actual artwork that will be turned into copper on fibreglass. This is where you really start to make the board and this is where you find the worst errors in your layout.

Making the artwork

The artwork is taped up on plastic draughting film that can be bought from the same place you bought the artwork aids. There are two sorts available; one is just plain translucent film and the other has very faint tenth-inch grid lines printed on it.

If you get the plain film then you'll have to stick it down on some tenth-inch graph paper. Use small pieces of masking tape on all four corners and make sure you can see through the film to the grid. If you use the film with a grid on it already then it can be worked on over a plain white sheet of paper and it isn't necessary to stick it down. Either way you should have a sheet of film with a grid easily visible on it.

Start by sticking down all the pads of your layout. Taping is done using a small penknife or, more usually, an exacto knife. Exacto knives can be bought from the same place as all the other artwork aids and have many other uses as well (such as modifying copper tracks on the final board). I personally use surgical scalpels which have long, easily held handles but anything with a small blade will do.

To lay down pads slip the tip of the blade under the edge of the pad and peel it off the backing sheet. The knife is normally held in three fingers of one hand leaving the forefinger and thumb free. When the tip of the blade is just under the pad the forefinger is placed over the tip of the blade and the pad to hold it firmly while the pad is peeled off.

The pad is then carefully aligned over the grid on the plastic film and the pad *firmly* pressed down on the film so the crossed lines on the grid are still visible in the hole in the pad. If you're putting down IC pads you must line up all 14 or 16 pads correctly. It looks terrible to have an IC drawing to see where they should go. You'll have to count off grid lines to locate the position of some pads but others simply lie side by side.

Keep on making checks to make sure you haven't slipped a grid line or so by holding the artwork being taped and your original layout together up to the light so both are visible over each other. Any errors will be immediately obvious. Also when you're laying down pads you'll probably find the occasional place where the layout shows a track going between two pads and there isn't enough space. Not to worry; this can be fixed later.

Once all the pads are down you're ready to start connecting them up. The golden rule here is to try and make the tracks and the spaces between them of equal width. When you finally use the artwork to make a printed circuit you'll discover that tracks less than about 0.020″ tend to be etched right through and similarly spaces between tracks less than 0.020″ tend to not separate so these are the absolute minimum spacings to be used. To make things easier until you're familiar with the process I strongly recommend that you use minimum dimensions of 0.040″ both for tracks and spaces. These rules (like any) can be bent if you know what you're doing but if not it saves rework to stick to them.

To connect between pads first trim the end of the tape square. Hold the end of the tape over the first pad and press it down firmly. Run the tape carefully in the path you want to follow until it reaches the second pad. Press it down firmly onto the second pad. Cut it off by pressing the blade of your knife onto the tape *not hard enough to cut it* then pull the tape up to cut it off. If you try and cut the tape while it's stuck on the pad you'll cut both tape and pad.

When you're laying down tape you must try not to put it down under tension. If the tape is stretched as it's stuck down it tends to creep back after it's cut off and open up spaces. This particularly applies to taping around corners.

It's inevitable that the layout will include some curved tracks. When you tape them *the tape must have no tension; better to be slightly in compression.*

To do this you must, as you're laying down the tape, continually push it back on itself slightly with the tip of the knife blade. It's something of a knack to do it correctly.

After every piece of tape is placed and cut off press it down *hard*. Pounding on it with a closed fist is quite acceptable — you can't press it down too hard.

During all this you *must* keep all grease or oil away from the artwork. Contact adhesives simply will not stick to oil and all your work will fall off if the film is dirty.

If you find that you must run tape between two pads that are too close and you can't maintain clearance then as a last resort it's permissible to carefully cut away a bit of the pad. There must be at least 0.030″ of pad left around the centre hole or when you drill the hole in the board you'll break through the side of the pad. Then you won't get a good solder joint. This method should always be thought of as an act of desperation — not a standard technique.

When you're taping up the artwork it helps a lot to always refer back to the original diagram to keep track of what you're doing. You'll probably find a mistake or two.

Once the taping is completed and checked, all that remains to be done is to mark the mounting holes with large diameter pads. Mark the corners of the board so you know where to trim after the board's been etched. You can put tape all around the edge of the board if you want but it's more usual only to mark the corners. The tape for the corner marks goes outside the actual edge of the board so when the board's trimmed the corner marks are cut off. When you think it's really finished check it again. This is the last chance you'll get to have an error free layout.

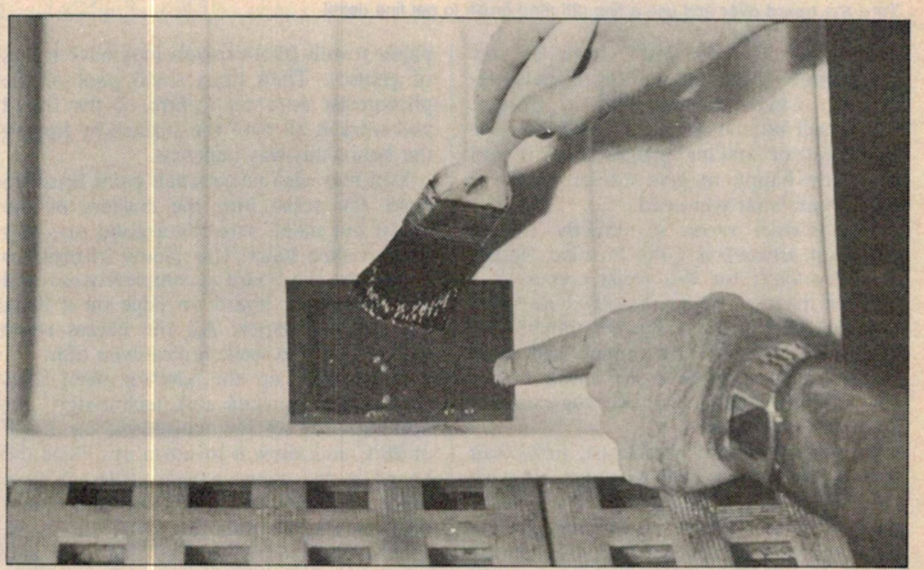
Etching

The next step is to ensure that the artwork is the correct polarity for the photo-etching process you intend to use. If you are using dry resist precoated board material then you must obtain a reversal, or negative, of the taped artwork you've generated. If the original artwork was twice full size this comes naturally, as the photography gives a negative.

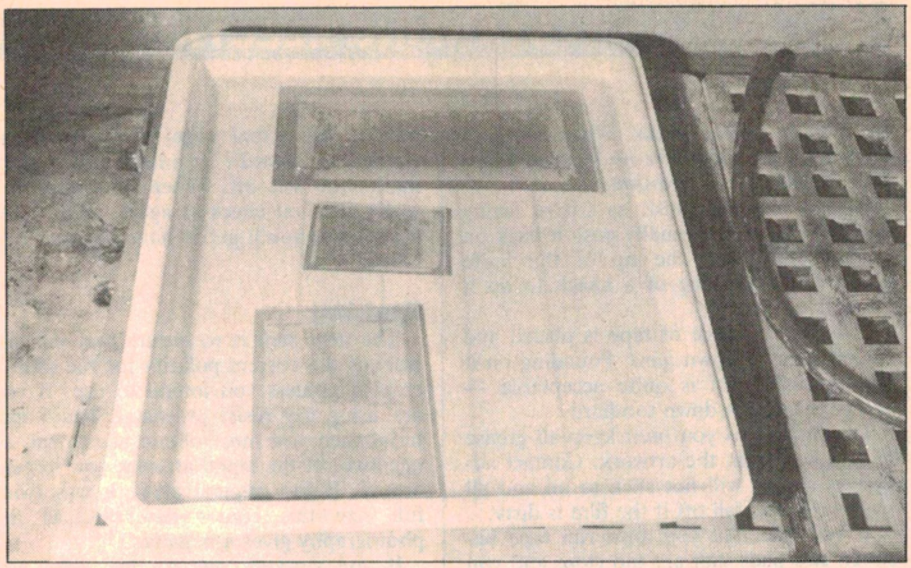
If you worked 'one to one' then you must get some "Scotchcal" exposure film. This stuff is like photographic film except that it's only sensitive to ultraviolet light and you only need one chemical to develop it.

To expose the film place the sensitive surface under the UV source and lay the mask over it. Make sure the mask is the right way up so the pattern that will finally appear is not mirror imaged. Place a sheet of good, heavy glass over the mask. I use a piece 6 mm thick which is enough to press the mask flat against the UV sensitive surface. For the suntan lamp UV source it's necessary to cover the whole assembly with something opaque (say a book) for about sixty seconds until the lamp warms up. After that the mask and sensitive material can be uncovered to commence the actual exposure.

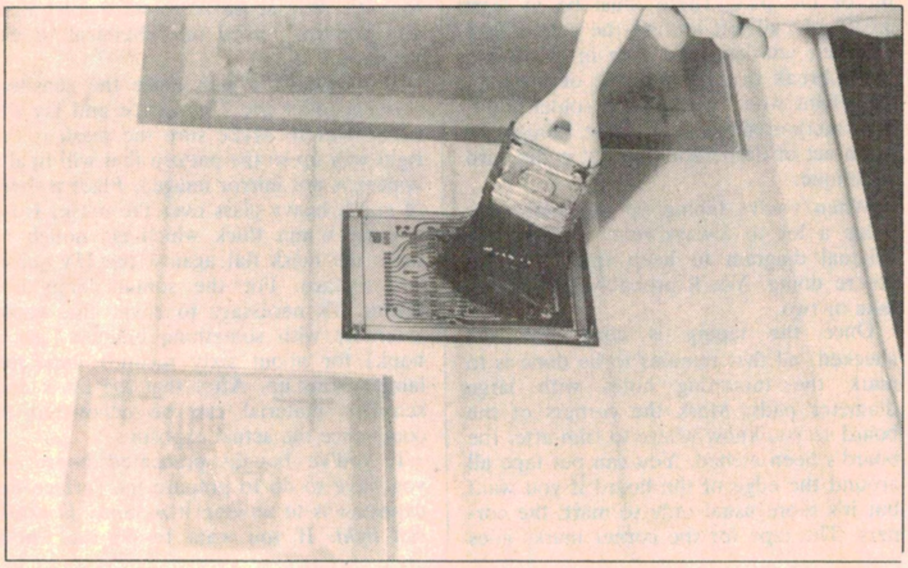
If you've bought precoated board all you have to do to prepare the surface for exposure is to unwrap it *in dimly lit artificial light.* If you want to do the whole ▶



Developing the resist. The developer comes in powder form and has to be dissolved in water to make up a solution. Treat this solution with care as it is poisonous.



The etching process. Place packing under the tray to tilt it at a slight angle and three-quarters cover the bottom with etchant. Drop the board into the high end of the tray, copperside up.



Turn the board over and use a fine (25 mm) brush to get fine detail.

Exposing the film

The photo etching process used with circuit boards works with ultraviolet light. This has the advantage that it does not require a darkroom. All the equipment can be handled in normal ambient light, with the proviso that you need to avoid any strong sources of UV. Since this includes the sun, it's probably a good idea to work at night if at all possible. If not, then make sure you are well away from windows, etc.

For the actual exposure, you can use either a sunlamp or a purpose built UV light. The recommended UV source for both Scotchcal and photoresist is a Philips UV mercury discharge tube. This will fit into a 20 W fluoro batten.

A quite effective exposure box can be made by nailing together a box with one side open. The internal surfaces should be painted white, and the lamp mounted in its batten opposite the opening. The opening should be filled with a sheet of glass. The distance from the top of the glass to the tube should be about 50 mm.

For exposure the box should be positioned with the glass side up, the artwork placed on the glass and held down with another piece of glass.

Note that when UV tubes have been unused for a while, or are being operated in very cold ambient temperatures, they take a while to reach their normal operating output. Leave them burning for at least fifteen minutes before you start work with them.

If you don't want to go to all that trouble, a simple, but less accurate way to do things is to use an ordinary old sunlamp. Position this about 350 mm above the artwork. But be careful. If you expose your hands to the rays for too long you'll get sunburnt.

thing yourself then you'll have to buy some photoresist and it's appropriate developer. Circuit Components in Bexley, NSW, sell all the necessary materials for all stages of making printed boards and are quite happy to give advice if you're not certain what you need.

The correct resist to directly expose taped up artwork is CPR Positive Resist. The developer for this resist comes as a powder that has to be dissolved in water to make up a solution. Have a thought for the neighbourhood kids though and *don't keep the developer in a soft drink bottle.* It's a poison and doesn't do rug-rats any good at all to drink.

To coat the board with resist, first clean the copper thoroughly with an abrasive cleanser such as "Gumption" using clean paper towels. Grease or fingerprints on the copper is a disaster for the coating process so don't touch the surface at all. After scrubbing the copper dry it with paper towels (cloth towels may have traces of grease). Then tip a small pool of the photoresist onto the centre of the board and work it all over the surface by tipping the board this way and that.

You may also use a small paint brush to work the resist into the corners of the board but make sure it's a good one that doesn't shed hairs. Use plenty of resist to make sure the board is completely covered then stand the board on edge on a thick wad of newspaper. All the excess resist will drain off to leave a nice even film.

Next stoke up the kitchen oven (this isn't called "kitchen sink technology" for nothing) and set the thermostat for 175°F or 80°C and allow it to warm up. Place the board in the oven — resist side up on a piece of paper towel or newspaper and cook it for 15 minutes. This serves to drive off all the solvent in the resist and leave a hard dry film.

If you leave it in too long the resist won't develop properly; too short and the developer takes off the lot so time it carefully. This process is called the prebake and is vital to good resist processing.

But note that some resists have special requirements you should be aware of. Circuit-components insist that "gas ovens, fan-forced electric ovens and sealed electric ovens are not suitable. Beware of stove ovens with grease deposits." They also warn against direct exposure to infrared from the elements of an electric oven.

After the 15 minutes are up take the board out. Lay the artwork over the board and cover the lot with a piece of glass as described. If you use the suntan lamp as a UV source the CPR film needs between 3 minutes 30 seconds and 4 minutes for correct exposure with the lamp 350 mm from the resist.

For the 20 W UV tube advice on exposure is given with the resist. After exposure lay the board face up in a glass dish and cover it with developer. *Don't* use aluminium or iron dishes as the developer will attack them as well.

Gently agitate the dish by rocking it. The resist will dissolve away where it's been exposed to light.

For the precoated boards you will have exposed it through a Scotchcal reversing film mask and the resist will dissolve where the mask protected it.

The developing process should only take a few minutes and when it's complete there should be absolutely no trace of resist where it should be dissolved. Even the minutest trace of resist will prevent proper etching and ruin the board so carefully examine the board to make sure the coloured resist has been completely dissolved.

To stabilize the resist film after development turn the oven up to 200°F or 95°C and postbake the board for 20 minutes. This bake isn't quite as critical as the prebake (once I forgot and left it in for 3 hours with no ill effects!). Finally take the board out and let it cool and stabilize for about half an hour and it's ready to etch.

The easiest etchant to use is ferric chloride and you can buy bottles of ready made solution. Be warned though, this stuff is the most foul staining corrosive gunk imaginable and the tiniest drop will leave a totally unremovable yellow mark on clothes or anything else.

Good housekeeping is essential. It will also etch stainless steel just fine (like kitchen sinks!!!) so don't spill it. If a few drops are spilt on a stainless steel sink and it's rinsed down the faint traces left will corrode the sink in a few hours and leave rust marks that are hard to get off. It's best to use the etchant in a tray placed on several thicknesses of newspaper.

As well as the etchant and tray you'll need a very cheap 25 mm paint brush with the bristles trimmed off about 10 mm. Place some packing under one end of the tray so it's tilted at a slight angle and tip enough etchant into the tray to three-quarters cover the bottom. Drop the board into the high end of the tray, copperside up, and brush the etchant over the copper.

You'll see a black powdery deposit form on the bare copper then dissolve and wash away. Keep on brushing, concentrating on the areas that have no tracks and have large bare areas to be etched. For some reason the etching process seems to favour areas that have lots of tracks. In five to ten minutes, depending on solution age and temperature, all the areas not covered by resist will completely dissolve.

It's possible to use ammonium persulphate instead of ferric chloride if you like. This is not as easy to use, but it doesn't stain and it's not as corrosive.

The process is essentially the same. However you need to heat it to 50°-60°C. It won't work at room temperature. Also, it can't be stored. It slowly releases gas and decomposes. If you attempt to seal it in a bottle, the bottle will explode.

As soon as all the copper is etched away remove the board from the tray and rinse it using megalitres of water. Examine the board carefully to make sure that all the copper is really dissolved then put the solution back in its bottle to be used again.

Rinse out the tray and brush once again using vast quantities of water and make sure you haven't spilled a drop of etchant anywhere. If you have you'll find out soon enough from the tirade of abuse from wife and/or mother.

Finally, remove the resist with solvent acetone and a paper towel. It comes off easily unless you forgot and postbaked it for hours. Trim the board to size with a hacksaw and clean up the edges with a file so you just remove the corner marks. Then drill all the holes with a small hand drill. The drill size should be about 0.8 mm for most components but some need a larger size, say 1.2 mm.

At this stage you should have a nice neat printed circuit board of your own design ready to assemble and try. For the first time the process seems long and messy but after a few tries you'll find it easy, quick and most satisfying. ●