HOW TO MAKE GOOD HOMEMADE PCBs?

INDRANI BOSE

Do not use sodium hydroxide for developing photoresist laminates. It is completely a dreadful stuff for developing PCBs. Apart from its causticity, it is very sensitive to both temperature and concentration, and made-up solution doesn't last long

Here is a guide to produce high-quality PCBs quickly and efficiently, particularly for professional prototyping of production boards. Unlike most other PCB homebrew guides, emphasis is placed on quality, speed, and repeatability rather than minimum materials cost, although the time saved by getting good PCBs every time usually saves money in the long run.

With the method described here, you can produce repeatedly good single- and double-sided PCBs for through-hole and surface-mount designs with track densities of 40 to 50 tracks per inch and 0.5mm SM pitches. Only photographic methods have been dwelt in depth because other methods such as transfers, plotting on copper, and 'iron-on' toner transfer are not really suited for fast, repeatable use.

The problem with toner transfer systems is that the 'expensive part' is the film, and you can't really feed much less than an A5 sheet through a laser printer, so you waste a lot on small PCBs. With photoresist laminate and cheap transparency media, you only use as much of the expensive part (the board) as you need, and offcuts can usually be used later for small boards.

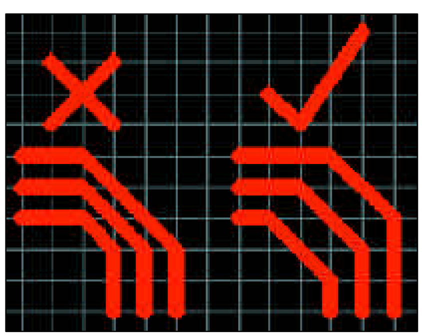
Artwork generation

You need to generate a positive (copper black) UV translucent artwork film. You will never get a good board without good artwork, so it is important to get the best possible quality at this stage. The most important thing is to get a clear sharp image with a very solid opaque black.

Nowadays, artwork is drawn using either a dedicated PCB CAD program or a suitable drawing/graphics package. It is absolutely essential that your PCB software prints holes in the middle of pads, which will act as centre marks when drilling. It is virtually impossible to accurately hand-drill boards without these holes.

If you're looking to buy PCB software at any cost level and want to do hand-prototyping of boards before production, check that this facility is available. If you're using a general-purpose CAD or graphics package, define pads as either a grouped object containing a black-filled circle with a smaller concentric white-filled circle on top of it, or as an unfilled circle with a thick black line (i.e. a black ring).

When defining pad and line shapes, the minimum size recommended for vias (through-linking holes) for reliable results is 50 mil, assuming 0.8mm drill size; 1 mil = (1/1000)th of an inch. You can go smaller with smaller drill sizes, but through-linking will be harder. 65mil round or square pads for normal components and DIL ICs, with 0.8mm hole, will allow a 12.5 mil, down to 10 mil if you really need to. Centre-to-centre spacing of 12.5mil tracks should be 25 mil—slightly less may be possible if your printer can manage it. Take care to preserve the correct diagonal track-track spacing on mitered corners; grid is 25 mil and track width 12.5 mil.



The artwork must be printed such that the printed side is in contact with the PCB surface when exposing, to avoid blurred edges. In practice, this means that if you design the board as seen from the component side, the bottom (solder side) layer should be printed the 'correct' way round, and the top side of a double-sided board must be printed mirrored.

Media

Artwork quality is very dependent on both the output device and the media used. It is not necessary to use a transparent artwork medium—as long as it is reasonably translucent to UV, its fineless translucent materials may need a slightly longer exposure time. Line definition, black opaqueness, and toner/ink retention are much more important.

Tracing paper has good enough UV translucency and is nearly as good as drafting film for toner retention. It stays flatter under laser-printer heat than polyester or acetate film. Get the thickest you can find as thinner stuff can crickle. It should be rated at least 90 gsm; 120

gsm is even better but harder to find. It is cheap and easily available from office or art suppliers.

Output devices

Laser printers offer the best all-round solution. These are affordable, fast, and good-quality. The printer used must have at least 600dpi resolution for all but the simplest PCBs, as you will usually be working in multiples of 0.06cm (40 tracks per inch). 600 dpi divides into 40, so you get consistent spacing and line width.

It is very important that the printer produces a good solid black with no toner pinholes. If you're planning to buy a printer for PCB use, do some test prints on tracing paper to check the quality first. If the printer has a density control, set it to the blackest. Even the best laser printers don't generally cover large areas well, but usually this isn't a problem as long as fine tracks are solid.

When using tracing paper or drafting film, always use manual paper feed and set the straightest possible paper output path to keep the artwork as flat as possible and minimise jamming. For small PCBs, you can usually save paper by cutting the sheet in half. You may need to specify a vertical offset in your PCB software to make it print on the right part of the page. Some laser printers have poor dimensional accuracy, which can cause problems for large PCBs. But as long as any error is linear, it can be compensated by scaling the printout in software.

Print accuracy is likely to be a noticeable problem when it causes misalignment of the sides on double-sided PCBs—this can usually be avoided by careful arrangement of the plots on the page to ensure the error is the same on both layers; for example, choosing whether to mirror horizontally or vertically when reversing the top-side artwork.

Photoresist PCB laminates

Always use good-quality, pre-coated photoresist fibreglass (FR4) board. Check carefully for scratches in the protective covering and on the surface after peeling off the covering. You don't need darkroom or subdued lighting when handling boards, as long as you avoid direct sunlight, minimise unnecessary exposure, and develop immediately after UV exposure.

Instagraphic Microtrak board develops really quickly, gives excellent resolution, and is available in thin (0.8mm) and heavy copper flavours. On using spray-on photoresist, you will always get dust settling on the wet resist. So it is not recommended unless you have access to a very clean area or drying oven, or you only want to make low-resolution PCBs.

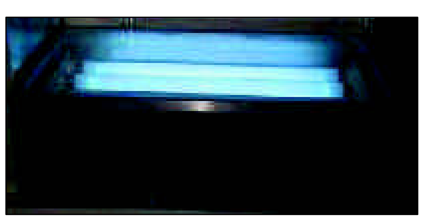
Exposure

The photoresist board needs to be exposed to UV light through the artwork, using a UV exposure box. UV exposure units can easily be made using standard fluorescent lamp ballasts and UV tubes. For small PCBs, two or four 8-watt, 30.5cm tubes will be adequate. For larger (A3) units, four 38cm tubes are ideal. To determine the tube-to-glass spacing, place a sheet of tracing paper on the glass and adjust the distance to get the most even light level over the surface of the paper.

Even illumination is a lot easier to obtain with 4-tube units. The UV tubes you need are sold as replacements for UV exposure units, 'black light' tubes for disco lighting, etc. These look white, occasionally black/blue when off, and light up with a light purple. Do not use shortwave UV lamps like EPROM eraser tubes and germicidal lamps that have clear glass, because these emit shortwave UV which can cause eye and skin damage.

A timer that switches off the UV lamps automatically is essential, and should allow exposure times from 2 to 10 minutes in 15- to 30-second increments. It is useful if the timer has an audible indication when the timing period has completed. A timer from a scrap microwave oven would be ideal.

Use glass sheet rather than plastic for the top of the UV unit, as it will flex less and be less prone to scratches. A combined unit, with switchable UV and white tubes, doubles as an exposure unit and a light-box for lining up double-sided artworks. If you do a lot of double-sided PCBs, it may be worth making a double-sided exposure unit, where the PCB can be sandwitched between two light sources to expose both sides simultaneously.



To find the required exposure time for a particular UV unit and laminate type, expose a test piece in 30-second increments from 2 to 8 minutes, develop, and use the time which gave the best image. Generally speaking, overexposure is better than underexposure.

For a single-sided PCB, place the artwork's toner side up on the UV box glass, peel off the protective film from the laminate, and place its sensitive side down on top of the artwork. The laminate must be pressed firmly down to ensure good contact all over the artwork.

To expose double-sided PCBs, print the solder-side artwork as normal and the component side mirrored. Place the two sheets together with the toner sides facing, and carefully line them up, checking all over the board area for correct alignment, using the holes in the pads as a guide. A light box is very handy here, but exposure can also be done with daylight by holding the sheets on the surface of a window.

If printing errors have caused slight mis-registration, align the sheets to average the errors across the whole PCB, to avoid breaking pad edges or tracks when drilling. When these are correctly aligned, staple the sheets together on two opposite sides, about 10 mm from the edge of the board, forming a sleeve or envelope. The gap between the board edge and staples is important to stop the paper distorting at the edge. Use the smallest stapler you can find, so that the thickness of the staple is not much more than that of the PCB.

Expose each side, covering up the top side with a reasonably light-proof soft cover when exposing the underside. Be very careful when turning the board over, to avoid the laminate slipping inside the artwork and ruining the alignment. After exposure, you can usually see a faint image of the pattern in the photosensitive layer.

Developing

Do not use sodium hydroxide for developing photoresist laminates. It is a completely and utterly dreadful stuff for developing PCBs. Apart from its caustic-

ity, it is very sensitive to both temperature and concentration, and made-up solution doesn't last long. When it's too weak it doesn't develop at all, and when too strong it strips all the resist off. It is almost impossible to get reliable and consistent results, especially when making PCBs in an environment with large temperature variations.

A much better developer is a silicate-based product that comes as a liquid concentrate. You can leave the board in it for several times the normal developing time without noticeable degradation. This also means that it is not temperature critical—no risk of stripping at warmer temperatures. Made-up solution also has a very long shelf-life and lasts until it's used up. You can make the solution up really strong for very fast developing. The recommended mix is 1 part developer to 9 parts water.

You can check for correct development by dipping the board in the ferric chloride very briefly—the exposed copper should turn dull pink almost instantly. If any shiny copper-coloured areas remain, rinse and develop for a few more seconds. If the board is under-exposed, you will get a thin layer of resist which isn't removed by the developer. You can remove this by gently wiping with dry paper towel, without damaging the pattern. You can either use a photographic developing tray or a vertical tank for developing.

Etching

Ferric chloride etchant is a messy stuff, but easily available and cheaper than most alternatives. It attacks any metal including stainless steel. So when setting up a PCB etching area, use a plastic or ceramic sink, with plastic fittings and screws wherever possible, and seal any metal screws with silicone. Copper water pipes may get splashed or dripped-on, so sleeve or cover them in plastic; heat-shrink sleeving is great if you're installing new pipes. Fume extraction is not normally required, although a cover over the tank or tray when not in use is a good idea.

You should always use the hexahydrate type of ferric chloride, which should be dissolved in warm water until saturation. Adding a teaspoon of table salt helps to make the etchant clearer for easier inspection.

Avoid anhydrous ferric chloride. It creates a lot of heat when dissolved. So always add the powder very slowly to water; do not add water to the powder, and use gloves and safety glasses. The solution made from anhydrous ferric chloride doesn't etch at all, so you need to add a small amount of hydrochloric acid and leave it for a day or two.

Always take extreme care to avoid splashing when dissolving either type of ferric chloride, as it tends to clump together and you often get big chunks coming out of the container and splashing into the solution. It can damage eyes and permanently stain clothing.

If you're making PCBs in a professional environment, where time is money, you should get a heated bubble-etch tank. With fresh hot ferric chloride, a PCB will etch in well under five minutes. Fast etching produces better edge-quality and consistent line widths. If you aren't using a bubble tank, you need to agitate frequently to ensure even etching. Warm the etchant by putting the etching tray inside a larger tray filled with boiling water.

Tin plating

Tin-plating a PCB makes it a lot easier to solder, and is pretty much essential for surface mount boards. Unless you have access to a roller tinning machine, chemical tinning is the only option. Unfortunately, tin-plating chemicals are expensive but the results are usually worth it.

If you don't tin-plate the board, either leave the photoresist coating on (most resists are intended to act as soldering fluxes) or spray the board with rework flux to prevent the copper from oxidising.

Room-temperature tin-plating crystals produce a good finish in a few minutes. There are other tinning chemicals available, some of which require mixing with acid or high-temperature use.

Ensure that the temperature of the tinning solution is at least 25°C, but not more than 40°C. If required, either put the bottle in a hot water bath or put the tinning tray in a bigger tray filled with hot water to warm it up. Putting a PCB in cold tinning solution will usually prevent tinning, even if the temperature is subsequently raised.

For a good tinned finish, strip the photoresist thoroughly. Although you can get special stripping solutions and hand applicators, most resists can be dissolved off more easily and cleanly using methanol (methylated spirit). Hold the rinsed and dried PCB horizontal, and dribble few drops of methanol on the surface, tilting the PCB to allow it to run over the whole surface. Wait for about ten seconds and wipe off with a paper towel dipped in methanol.

Rub the copper surface all over with wire wool until it is bright and shiny. Wipe with a paper towel to remove the wire wool fragments and immediately immerse the board in the tinning solution. Don't touch the copper surface after cleaning, as finger marks will impair plating. The copper should turn silver in colour within about 30 seconds. Leave the board for about five minutes, agitating occasionally; do not use bubble agitation. For double-sided PCBs, prop the PCB at an angle to ensure the solution gets to both sides.

Rinse the board thoroughly and rub dry with paper towel to remove any tinning crystal deposits. If the board isn't going to be soldered for a day or two, coat it with either a rework flux spray or a flux pen.

Drilling

If you have fibreglass (FR4) board, you must use tungsten carbide drill bits. Fibreglass eats normal high-speed steel (HSS) bits very rapidly, although HSS drills are alright for odd larger sizes (>2 mm). Carbide drill bits are expensive and the thin ones snap very easily. When using carbide drill bits below 1 mm, you must use a good vertical drill stand—you will break drill very quickly without one.

Carbide drill bits are available as straight-shank or thick (sometimes called 'turbo') shank. In straight shank, the whole bit is the diameter of the hole, and in thick shank, a standard-size (typically about 3.5 mm) shank tapers down to the hole size. The straight-shank drills are usually preferred because they break less easily and are usually cheaper. The longer thin section provides more flexibility.

Small drills for PCB use usually come with either a set of collets of various sizes or a 3-jaw chuck. Sometimes the 3-jaw chuck is an optional extra and is worth getting for the time it saves on changing collets. For accuracy, however, 3-jaw chucks aren't brilliant, and small drill sizes below 1 mm quickly form grooves in the jaws, preventing good grip. Below 1 mm, you should use collets, and buy a few extra of the smallest ones, keeping one collet per drill size, as using a larger drill in a collet will open it out and it no longer grips smaller drills well.

You need a good strong light on the board when drilling, to ensure accuracy. A dichroic halogen lamp, under-run at 9V to reduce brightness, can be mounted on a microphone gooseneck for easy positioning. It can be useful to raise the working surface about 15 cm above the normal desk height for more comfortable viewing. Dust extraction is nice, but not essential—an occasional blow does the trick! A foot-pedal control to switch the drill 'off' and 'on' is very convenient, especially when frequently changing bits. Avoid hole sizes less than 0.8 mm unless you really need them.

When making two identical boards, drill them both together to save time. To do this, carefully drill a 0.8mm hole in the pad near each corner of each of the two boards, getting the centre as accurate as possible. For larger boards, drill a hole near the centre of each side as well. Lay the boards on top of each other and insert a 0.8mm track pin in two opposite corners, using the pins as pegs to line the PCBs up. Squeeze or hammer the pins into the boards, and then into the remaining holes. The two PCBs are now 'nailed' together accurately and can be drilled together.

Cutting

A small guillotine is the easiest way to cut fibreglass laminate. Ordinary saws (bandsaws, jigsaws, and hacksaws) will be blunted quickly unless these are carbide-tipped, and the dust can cause sink irritation. A carbide tile-saw blade in a jigsaw might be worth a try. It's also easy to accidentally scratch through the protective film when sawing, causing photoresist scratches and broken tracks on the finished board. A sheet-metal guillotine is also excellent for cutting boards,

provided the blade is fairly sharp.

To make cut-outs, drill a series of small holes, punch out the blank, and file to size. Alternatively, use a fretsaw or small hacksaw, but be prepared to replace blades often. With practice it's possible to do corner cutouts with a guillotine but you have to be very careful that you don't over-cut!

Through-plating

When laying out double-sided boards, give some thought to how top connections will be made. Some components, such as resistors and unsocketed ICs, are much easier to top-solder than others (radial capacitors). So, try to make the top connection to the easier component. For socketed ICs, use turned-pin sockets, preferably the ones with thick pin section under the socket body. Lift the socket slightly off the board, solder a couple of pins on the solder side to tack it in place, and adjust so that the socket is straight. Solder all the solder-side pins, and then the required top-side pins by

reheating the joint on the solder side.

For vias, holes which link sides without components, use 0.8mm snap-off linking pins. These are much quicker than using pieces of wire. Just insert the bottom of the stick into the hole and bend over to snap off the bottom pin. Repeat the process for other holes and then solder both sides.

If you need 'proper' through-plated holes—for example, to connect to inaccessible top-side pins, or for underneath surface-mount devices—Multicore's Copperset system works well, but the kit is very expensive. It uses bail bars consisting of a rod of solder, with a copper sleeve plated on the outside. The sleeve is scored at 1.6mm intervals, corresponding to the PCB thickness. The bar is inserted into the hole using a special applicator and bent over to snap off the single bail in the hole. It is then punched with a modified automatic centre-punch, which causes the solder to spray over the ends of the plated sleeve and also pushes the sleeve against the side of the hole. The pads are soldered on side to join the sleeve to the pads, and then the solder is removed with braid or a solder sucker to leave a clear plated hole.

Fortunately, it is possible to use this system for plating standard 0.8mm holes without buying the full kit. You can buy the bail bars separately as refills. For the applicator, use a 0.9mm automatic pencil that works much better than the original applicator, as you get one bail for every press of the button. Get a small automatic centre-punch and grind the tip off to make it completely flat—this works fine for punching the bails. For an anvil, use a thick flat piece of metal. Plate all the holes before fitting any components so that the bottom surface is completely flat. Holes must be drilled with a sharp 0.85mm carbide drill to get the hole size right for the plating process.

Note that if your PCB package draws pad holes of the same size as that of drill, the pad hole can come out slightly larger than the drilled hole, causing connection problems with the plating. Ideally, the pad holes should be about 0.5 mm, regardless of hole size, to make an accurate centre mark. ❑