

Making Printed Circuit Boards

By Justin Giles-Clark VK7TW
May 2007

This demonstration is of a process for one-off high quality fine tracked PCBs (suitable for SMD applications) for either single or double sided components (two layer) using the toner transfer film method. It is suited more to CAD designed PCBs and I will be demonstrating the use of Schematic and PCB CAD software.

Software

- Many types available – DOS, Windows, Linux, Mac: Google for “PCB Design Software” and “Freeware”. See: http://www.pcb-pool.com/html_uk/uk_service_1.htm for a good listing of what is available.
- The one I am demonstrating today is:
 - ExpressPCB - http://www.expresspcb.com/ExpressPCBHtml/Free_pcb_layout_software.htm
 - ExpressSCH - http://www.expresspcb.com/ExpressPCBHtml/Free_schematic_software.htm
- Get used to working in imperial measurements (mils or thou of an inch – 1 mil = 0.001” = milli-inch = 25.4 microns = 1 thousandth of an inch. Standard measurement is 100 thou or 0.1” or 2.54mm) there is a movement toward metric however many PCB organisations still work in imperial.

Design Hints

- Use a standard grid while placing the board. For this kind of design a .025" grid is good for placement (default for ExpressPCB)
- Try to place integrated circuits in an even matrix. This allows space for components around the IC and allows power and ground connections
- Place support parts such as resistors and capacitors based on where they are connected within the circuit (net function is handy in ExpressPCB)
- Arrange the parts so they are evenly spaced, so they do not overlap, so their pads are a sufficient distance apart to allow the interconnecting traces to pass between. Note that pads should have at least a .020" gap between each other for a general purpose board.
- Place parts in only one or two orientations, vertical and horizontal. It is best to place polarized parts such as caps and diodes all in the same orientation (i.e. all diode cathodes to the right or up) to keep assembly and repair errors to a minimum.
- Axial parts such as resistors and diodes should be placed so there is an equal amount of lead wire on either side of the body, and the two pads are at a similar distance as other axial parts.
- Route traces using 45° corners not 90° corners when changing directions.
- Allow sufficient diameter around mounting holes for the hardware to be inserted and tightened without affecting parts placed around it.
- Allow sufficient distance between parts that their leads will not touch when bent over to secure them before soldering.

Equipment:

- Press & Peel PCB transfer film – Jaycar CAT. NO. HG9980 - \$31.50 for 5 sheets of 215 x 280mm size (a little smaller than A4) **BE CAREFUL WHEN PHOTOCOPYING!!!**
- Blank copper PCB single 150x150mm (\$6.20) or double sided 150x150 (\$6.95)
- Ammonium Persulphate Etch - Jaycar - CAT. NO. NC4254 (\$9.95 for 400gms)
- PCB Pen for touchup - DSE Cat No. N5181 (\$7.68)
- Temperature Controlled Iron (non-steam or ability to turn off steam)
- Plastic trays/basins/buckets
- Sharp Scissors
- Packing Tape
- Personal Protective Equipment:
 - Rubber Gloves
 - Safety Glasses
 - Old clothes!!!
- Drilling – High Speed Dremel or equivalent (drill stand but not essential)
- Drills – minimum of 3.5mm (mounting screws), 1.5mm (connectors), 1mm (normal resistors, caps) and 0.8mm (smaller leads & components, etc).
- Total Cost:
 - Without Drill & Drills its about \$80 for the equipment
 - With Drill & Drill about \$160 for all equipment

Cleaning

- Important to get a good bond for the resist (whatever type used)
- Fine steel wool with or without soap
- Fine >300grit Wet & Dry paper
- Scotchbrite pads
- Soap and warm water (removes the Cu oxide)

Photocopying

- Direct print to laser printer or photocopier (toner based) – **DO NOT USE INK JET** it will not work
- If no laser printer available then print using photo quality on your ink-jet and take to a photocopier place and transfer to the film or
- Can use PDF printer (CutePDF, PDF995, Adobe) to create a 1:1 image which can be put on disk or memory stick and taken to printing place.
- When printing from a PDF **MAKE SURE** in the print dialog box that the scaling = **NONE** or the PCB will come out a different size than the design.
- If using a design from a magazine check:
 - Scale = 1:1?
 - Does it need reversing – use transparent (OHP) film to do this?
 - Increase contrast to >80% to remove any newsprint background
- Photocopy or print the design (in reverse!) onto the dull blue side of the transfer film
- Leave about ¼ inch of transfer film around the design
- **NOTE:** the transfer film is the same width as A4 **BUT** it is shorter by 17mm **THEREFORE** beware of your alignment when you are using a photocopier

Transferring

- NOTE: For double sided boards I drill two 0.8mm holes in opposite corners matching two component holes so I can line-up (register) the top and bottom images
- Turn steam off on the iron
- Iron Temperature (IMPORTANT):
 - Press-N-Peel recommend 275-325°F (135-163 °C) Acrylic/Polyester setting
 - Depends on the iron – I have found the wool setting to be best
 - Don't go too high as you will start to see the toner pull away from the copper and transfer film – if you can start to see copper when transferring the toner then IT IS TOO HOT!
- Need to get the whole PCB up to the temperature
- Takes about 1.5-5 minutes and you can see the toner tracks through the transfer film go a darker black when the toner has bonded to the copper (spend more time rather than less!!)
- Concentrate on the edges to ensure all tracks are bonded
- Once all tracks bonded – quench the PCB under cold water – using ¼” of transfer film to pickup
- Peel off the transfer film from one corner – discard the film
- If the PCB has fine tracks and not all the blue transfer film has lifted off – stick packaging tape onto the PCB to lift the non-bonded segments off and carefully peel off.
- If necessary repair any non-bonded tracks with dalo pen or adhesive PCB tape

Etching

- **PUT ON THE GLOVES AND SAFETY GLASSES - DO NOT WEAR CONTACT LENSES WHEN ETCHING**
- Ammonium Persulphate – **VERY STRONG OXIDISING AGENT PRECAUTIONS REQUIRED.** - MSDS Sheet – it is sulphuric acid and ammonium – dissolves the copper as copper sulphate (blue colour)
- Mix as per instructions with hot water 75°C into tray that will fit the PCB
- Put tray into sink and quarter fill sink with hot water (may need to do this several times to keep the solution around 75°C)
- Constantly agitate the solution to keep the dissolved copper away from the board being etched. This also ensures uniform etching across the PCB.
- Ensure all unnecessary copper is removed and remove board from etchant and wash PCB under water to remove any etchant.
- Safely dispose of etchant – **BE CAREFUL IT IS A STRONG ACID.**

Drilling

- **PUT ON THE SAFETY GLASSES!!!!**
- Get used to working with metric drill sizes
- Use the following drills for:
 - 3.5mm for mounting holes and TO220/TO3, etc
 - 1.5mm for connectors/headers and larger components
 - 1.0mm for med to large resistors/capacitors – Jaycar CAT. NO. TD2421 \$14.50 for 10.
 - 0.8mm for most resistors/caps/ICs/etc - Jaycar CAT. NO. TD2420 \$14.50 for 10.
- Use micrometre or vernier callipers to check lead thicknesses and drills
- For the small drills <1.5mm run the drill speed as fast as possible and tend to punch the drills through the board – this will improve drill longevity
- Always drill with the copper side on top

Cleaning/Sealing

- If not assembling the PCB for a while then suggest leaving the resist on the copper to protect it – or clean the resist off with steel wool, scotch-brite pad or wet and dry paper then shellac the PCB.
- The shellac protects the copper from tarnishing (oxidising), can be soldered through and is also a mild flux when you get around to assembling the board (thanks Mike, VK7MJ).

Assembly

- Print out the silk screen layer of the PCB as a full page print to assist the assembly
- Unfortunately I haven't worked out how to get the silkscreen to print out the component values yet, therefore I write the values on the printout and follow that.
- Once assembled - smoke test the board!

Useful References

- Package I use: <http://www.expresspcb.com/>
- Helpful list of utils and software: <http://www.electronicsaustralia.com.au/cgi-bin/downloads.pl?area=7>
- Tracewidth Calculators (needed for higher current carrying):
 - <http://www.geocities.com/CapeCanaveral/Lab/9643/TraceWidth.htm>
(<http://www.geocities.com/suppanz/TraceWidth.xls>)
 - http://www.circuitcalculator.com/4pcb/trace_width_calculator.php
 - http://polysat.calpoly.edu/documents_cp1/systems/trace_capacity.pdf
- Trace Spacing (dependent on voltage):
<http://www.mitsi.com/PCB/Hints/spacing%20guidelines.htm>
- <http://www.rcsradio.com.au/> (Australian PCB Manufacturer - Bob Barnes) – It's for sale!
- Tolerances: http://www.4pcb.com/fast_design_quote_online_pcb_tolerance.htm
- Drill Sizes: <http://www.mitsi.com/PCB/Hints/Drill%20sizes.htm>
- Solid Wire info: http://www.mitsi.com/PCB/Hints/solid_wire_information.htm
- SMT Design Hints: <http://www.pcbdesignandfab.com/pcbdesignarticles.htm>
- General Design Tips:
 - http://www.4pcb.com/production_boards_custom_proto_prototype_design_tips.htm
 - <http://www.smps.us/pcb-design.html>
 - <http://www.epanorama.net/links/basics.html#pcb>
 - <http://mywebpages.comcast.net/pcb.george/>
 - <http://www.pcbstandards.com/forums/archive/index.php/f-16.html>
 - <http://www.mitsi.com/PCB/Hints/tips.htm>
- Glossary of PCB Terms:
http://www.4pcb.com/custom_quickturn_pcb_tooling_fast_glossary_of_terms.htm
- Interesting Forum: <http://www.pcbstandards.com/forums/index.php> (check the archives for detailed threads)
- PCB Design Blog: <http://blog.screamingcircuits.com/>