

A simple way of making double-sided printed-circuit boards
Advanced Development Technology Group
Gray Cancer Institute

There are numerous ways of designing and manufacturing printed circuit boards. When only a few boards are required, the expense associated with a professional 'run' is often not warranted, particularly if the design is a prototype (a euphemism for 'full of mistakes'!). Single-sided boards are easy: just place the mask on a photo-resist-coated board, expose, etch and drill. Double-sided boards a little trickier. Here we describe the methods used at GCI which give near-perfect results every time.

The equipment and consumable we regularly use is listed below:

Printed circuit board blanks, single-sided, Eurocard format 100 x 160 mm,	Rapid Electronics,	34-0105
Printed circuit board blanks, double-sided, Eurocard format 100 x 160 mm,	Rapid Electronics,	34-0125
Printed circuit board blanks, single-sided, Eurocard format 100 x 220 mm,	Rapid Electronics,	34-0110
Printed circuit board blanks, double-sided, Eurocard format 100 x 220 mm,	Rapid Electronics,	34-0130
UV transmissive masks, box of 100 A4 sheets,	Rapid Electronics,	39-0776
UV Exposure unit	RS Components,	659-279
Developing solution, 20g Developer,	Rapid Electronics,	34-0395
Etching solution, 2.5Kg Etchant,	Rapid Electronics,	34-0785
Etching tank, Bubble Etch Tank,	Rapid Electronics,	34-0760
Bubble Bar Assembly,	Rapid Electronics,	34-0767

Rapid Electronics
Severalls Lane, Colchester, Essex, CO4 5JS
Tel: 01206-751-166, Fax: 01206-751-188
Web: <http://www.rapidelectronics.co.uk>

The masks can be produced with the aid of any computer program, which can print on A4 paper on a laser printer. We tend to use the 'EasyPC' software, but this is clearly not relevant to the method. Once the masks are printed, making sure the printed side is arranged to be closest to the board emulsion, the trick is to sandwich a scrap piece of printed circuit board between the two masks. One side is glued to the scrap board and the other is carefully aligned to the first mask, then glued to the scrap board.

The production board can then be inserted between the two masks and held carefully in place, making sure that it does not move between exposures (we use a single-sided illumination unit, but double-sided are available; the particular unit we use is in fact no longer readily available). Both sides are then exposed, developed and etched. We find that the following conditions are optimum:

Exposure time:

2'30'' minutes per side

Developer concentration:

For the developer we use the SENO 4007 Photoresist Universal Developer (Rapid 34-0395); this comes in crystal form, in 20 g plastic bags. One plastic bag should not make less than 0.5 litre of developer. As 0.5 litres are almost never enough to develop a board it is advised to make 1 litre of developer using 1¾ of the 20 g universal developer bags. This is a good idea because if the developing solution is less potent you have less a chance of *over-developing* the board, which later may have the effect of producing an *over-etched* board.

Note: The developer strength for the concentration advised above gradually deteriorates. At least fifty 100 x 160 boards can be developed over a period of 50 days. However unlike the Ferric Chloride it does deteriorate and should be disposed of safely after 50 days even if it is not used.

Developing time:

Although the Developer package advises that no more than 30 seconds are needed for the board to develop, if the concentration is as advised above it will usually takes no more than a 1 minute.

Etch concentration:

For the etching solution we use Ferric Chloride Hexahydrate Granules 2.5 kg Etchant (Rapid 34-0785). This includes a bucket and two (2) 1.25 kg plastic bags that contain the etchant. It is advised that for 4 litres of etchant, 1½ of the 1.25 kg bags are used. The amount of etchant made is then enough to fill the etching pool tank.

Etching time/Etching conditions:

The etching time depends on a number of factors. It mainly depends on the size of the board, the state of the etchant (how many boards were previously etched) and how effective the stream of air bubbles is in the etching tank. Assuming that we have a 100 x 160 mm Eurocard board, that the etchant has just been made and that the air coming out of the tank is pumped vigorously, the etching time should be no more than 10 minutes. If we now have a 100 x 220 mm Eurocard board and the above assumptions are again made the etching should take no more than 12-14 minutes.

We have not found a really simple and effective way to test the strength of the etchant. If the etchant is made with the concentrations detailed above, it can be expected to etch at least forty 100 x 160 mm Eurocard boards. The state of the etchant does not deteriorate with time and if 2-3 100 x 160 mm Eurocard boards are made per week (which was the average production of boards per week in 2003-2004 at GCI) the etchant will last for up to 4 months. During this period, the Ferric Chloride is potent enough to etch adequately, it is inevitable that the 'U' like tube at the bottom (the Bubble Bar Assembly) of the etching pool tank that distributes the air, will get blocked from the copper that is etched from the boards. It is advised that the 'U' tube be changed with a **new** one every 20-30 100 x 160 mm Eurocard boards, or every 2 months. However, it is common for the tube to get blocked faster at certain parts than it does at others. Usually the top and bottom parts get blocked first. This has the effect that, depending on where the Eurocard board gets placed in the tank, certain parts will etch faster than others. This might create a higher risk of **over-etched** tracks at those parts. It has also been observed that the right side of the board when placed in the net inside the etching pool tank etches faster. The two above problems can be solved by flipping the board 2-3 times during the etch process and by moving it along the length of the pool tank to distribute and even out the etching process.

Note: The 'U' tubes can be reused if they get thoroughly cleaned, but this is by no means straightforward: traces of copper tend to remain stuck in the tube.

In case all this sounds rather difficult and hit-and-miss, it should be pointed out that this is not so; with a little practice and care, boards with track widths down to 0.3 mm and 0.2 mm separation are readily made. One final item is needed: an old lab-coat, one that you don't mind getting badly stained. If you are seriously fashion-conscious, there is only solution: perform the etching in a part of the laboratory that is reasonably remote, no-one will see you there. Alternatively, act confident and pretend that you own one of the latest 'designer range' of lab-coats, with carefully and artistically placed nasty-looking brown marks all over it.

Disposing the Developer and Etchant:

You are obliged by law to dispose the Developer and Etchant in a proper manner. At GCI the person in charge of disposing chemical waste is Matt Naylor naylor@gci.ac.uk). If you just started prototyping PCBs it is important that you go and talk to him. You are also obligated to read, understand and sign all the relevant COSHH risk assessment forms (Developing Photoresist PCB's, Etching Copper from PCB's and Using Glue) and any relevant documentation that is supplied from companies about safety precautions.

Note of Caution: It is advised that a lab coat and gloves are used during developing or etching as both these chemicals are irritants and can easily stain your 'designer' clothes. An orange colour mark will appear on your clothes if you get stained. To remove these marks, soap will not work. It is advised that the stained area be sprayed with a few drops of lemon, then left in bright sunlight (not through a window as this attenuates the UVA/UVB) for at least one hour and finally washed thoroughly. However, white marks may appear in place of the orange coloured stains so USE THE ABOVE METHODE AT YOUR OWN RISK.

Finally here is what it looks like in practice:



Figure 1: Preparing the artwork for exposing a double-sided board.

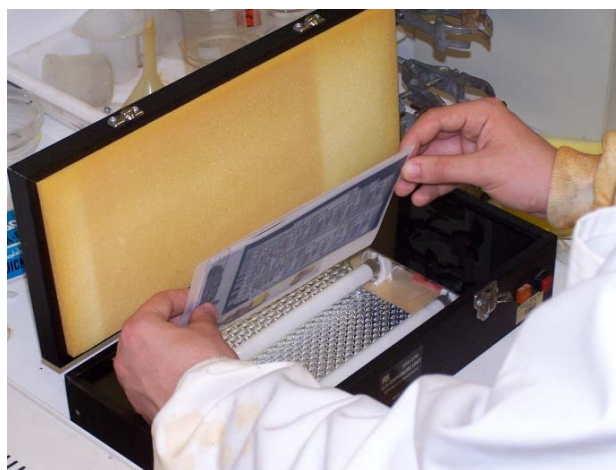


Figure 2: Turning over the board during the exposure of a double-sided board. No sudden movements, jostling or tickling allowed during this process!



Figure 3: Developing the board.



Figure 4: Etching the board.



Figure 5: The least boring final part.....: drilling the holes. It gives you the chance to contemplate about the beauty of life, the Universe and everything, but particularly why on earth you did not simplify the design....



Figure 6: The fun part: putting it all together.

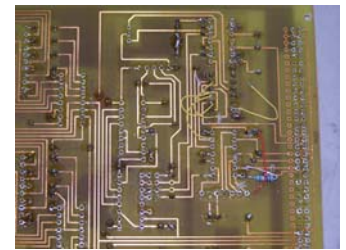


Figure 7: The depressing part when you realise that you need to do it all over again; there's a mistake....but it's only a little mistake. Who designed this anyway?

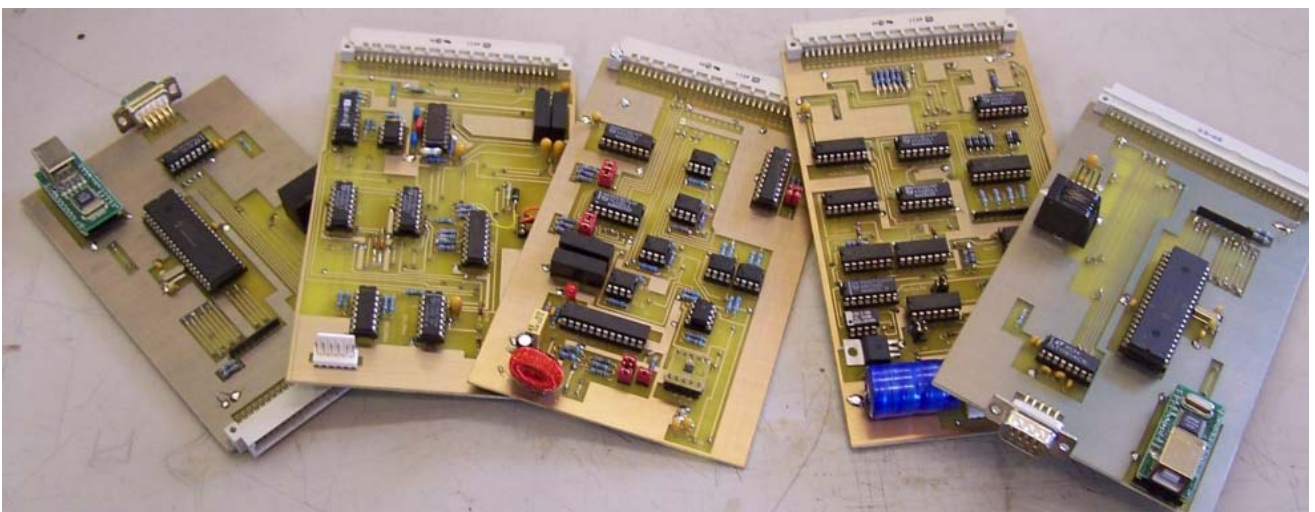


Figure 8: Yes, it has all been worth it.... a collection of completed and tested boards.

Notes on printing UV transmissive masks

The masks (Rapid Electronics, 39-0776) are best printed on laser printer in Electronics Workshop. The settings below are used:

From Easy-PC files, through Ravitz Software to Windows Print files and onto HP LaserJet 4+

Fit then magnify: set to '0.74' (normal magnification 0.5);

Pen width: set to 'dots 4' (normal pen width 'dots 1');

Window margin: set to 'default';

Origin: set to 'top left';

Destination: set to 'Win print manager';

Position: set to 'plot 0, 0 at print origin';

Box: set to 'No box';

Output format: set to 'Laser Jet 4, 600 x 600';

Scaling points: set to 'Fit with 0.25" margin';

Modify output: HV size multiplier 1.07, 1.07.