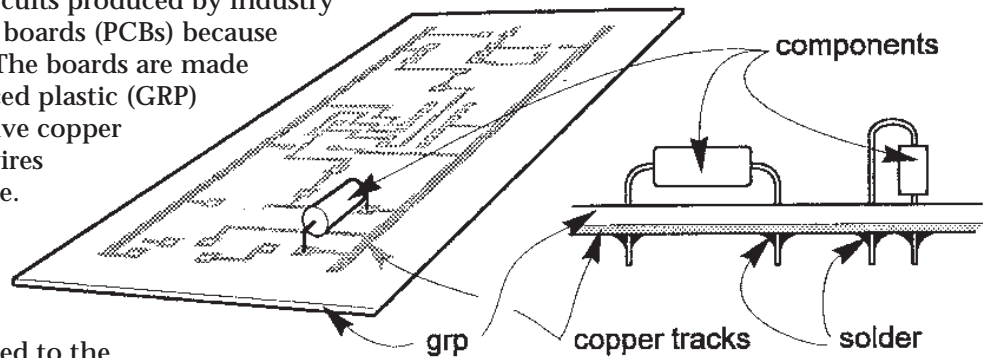


MAKING A PRINTED CIRCUIT BOARD

INTRODUCTION

Most electronic circuits produced by industry use printed circuit boards (PCBs) because they are reliable. The boards are made from glass reinforced plastic (GRP) and have conductive copper tracks instead of wires printed on one side. Holes are drilled through the board to take components which are connected to the copper tracks by soldering.



To start to make your PCB, you'll need:

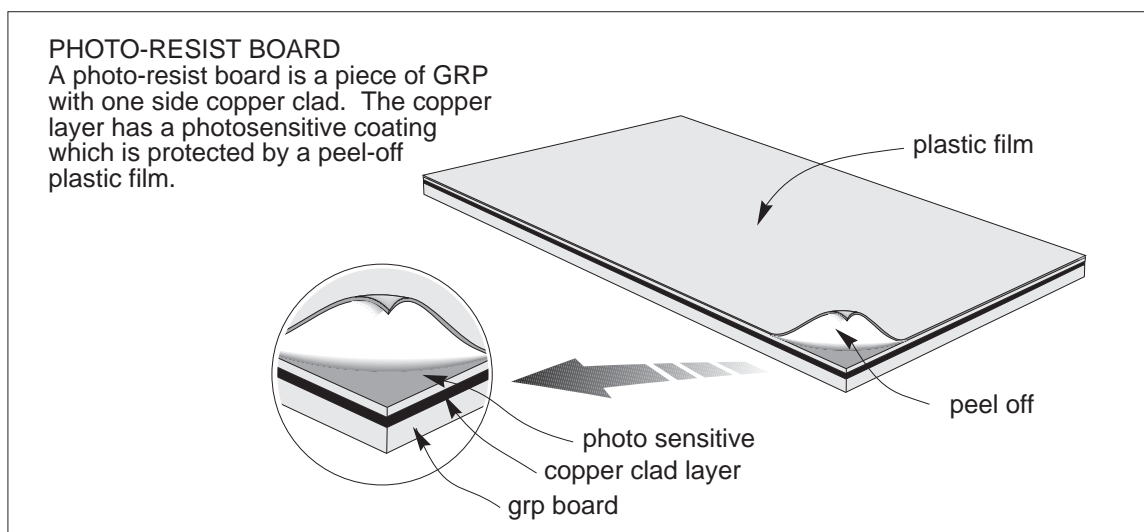
- A PCB mask of your circuit.
- A piece of photo-resist board.

CUTTING THE BOARD

When you cut the photo-resist board to the right size determined by your mask, it is advisable to leave at least 4 mm around the edge of the mask for final trimming. Always leave the black film on the board and cut it with a hacksaw with the black film side uppermost.

It is also best to cut from both ends of the board to prevent it from cracking. It is not essential but you can smooth the edges with a file.

When you are cutting the board, tape the mask on the board and mark the corners of the PCB on the mask with a marker pen. Try hard not to touch the sensitised surface of the film and make sure you do not get dust between the PCB and the mask.



PRODUCING THE PCB LAYOUT DIAGRAM

A PCB layout diagram may be obtained in a number of different ways.

1. Copying the layout from a diagram provided in a magazine or text book.
2. Hand drawing a PCB layout on acetate using rub down transfers to position pads, track and text.
3. Using CAD (computer aided design) software. PCB design software is available with varying degrees of sophistication.
 - i) At the simplest level a software package will allow you to position pads and track using the computer screen and obtain a printout of the PCB layout. The main advantage this method has over using rub down transfer is that errors can easily be corrected and the full range of different types of pads and track are always available.
 - ii) More sophisticated PCB design software will allow you to draw a schematic circuit diagram on the screen and the software will design a PCB layout from the circuit diagram.

The CAD drawings provided at the end of each of the units in this book have been produced on such a package.

In each case four drawings have been produced.

- A schematic circuit diagram
- A PCB layout diagram
- A component layout diagram
- A wire link diagram (this is only necessary in quite complicated circuits where there is no direct route available between two points on the PCB layout).

The name of the PCB design software package used is '**Quickroute 3.0 Curriculum**' and is available from Economics.

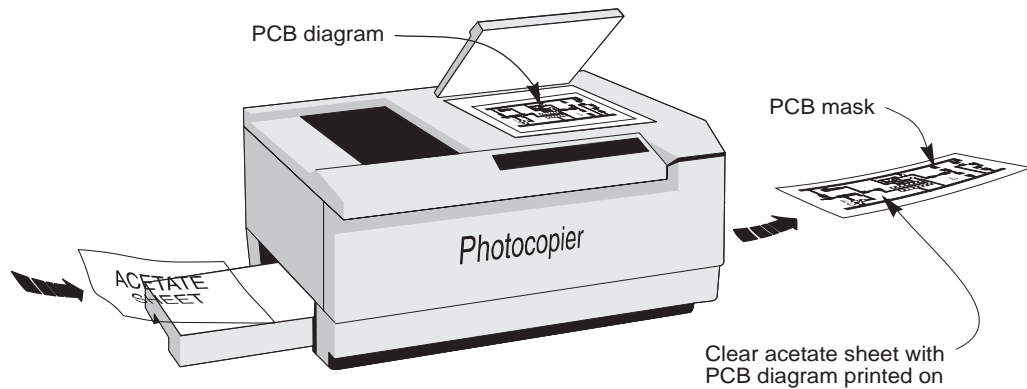
The minimum requirements to run the software are:

A 386 based PC
4 Mega bytes of RAM
3 Mega bytes of free hard disc space
A colour monitor with VGA

A floppy disc is supplied which contains a demonstration program as well as the Quickroute files used in the book. If you have access to Quickroute 3.0, you can load these files into your software program and modify them as required.

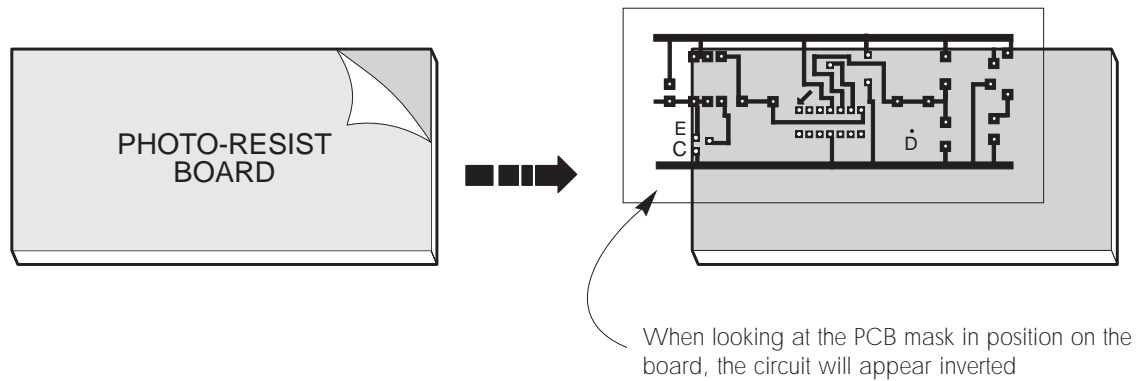
MAKING THE PCB MASK

The PCB mask can be produced by photocopying the PCB diagram onto clear acetate sheet. The PCB diagram may be hand drawn or printed out from a computer.



DEVELOPING THE CIRCUIT

- Cut the PCB mask from the acetate sheet to fit the photo-resist board.
- Peel off the protective plastic film and place the PCB mask on the exposed surface of the board. A diagram of a typical mask being placed on photo resist board is shown below.



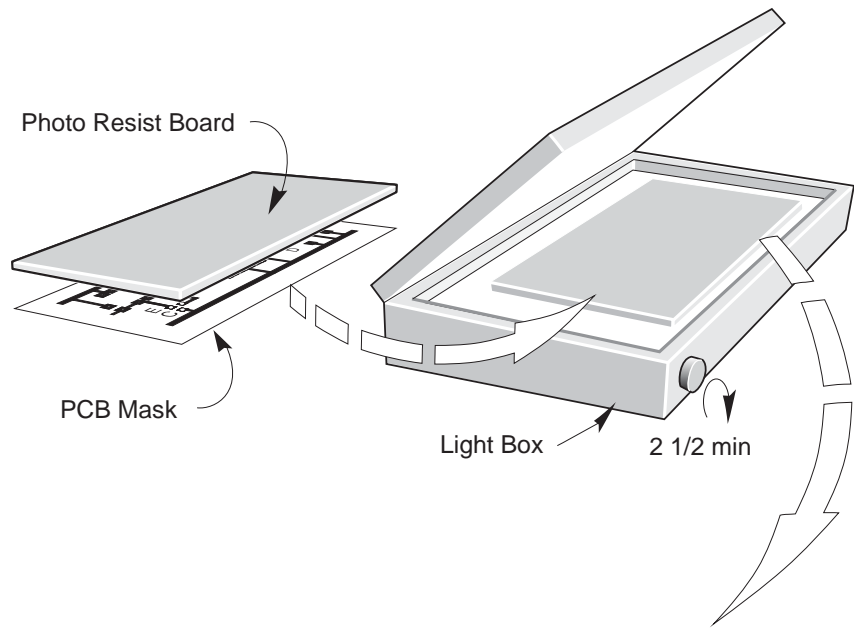
CAUTION:

The next stages in PCB manufacture must be organised safely.

- 1 Ultra-Violet (UV) light can damage eyes. Wear dark goggles.
- 2 Developer and ferric chloride are corrosive. Wear goggles, rubber gloves and apron.

NOTE: A non-corrosive Universal Developer is recommended and available from RS (stock number 690-849). Safer alternatives to ferric chloride are available.

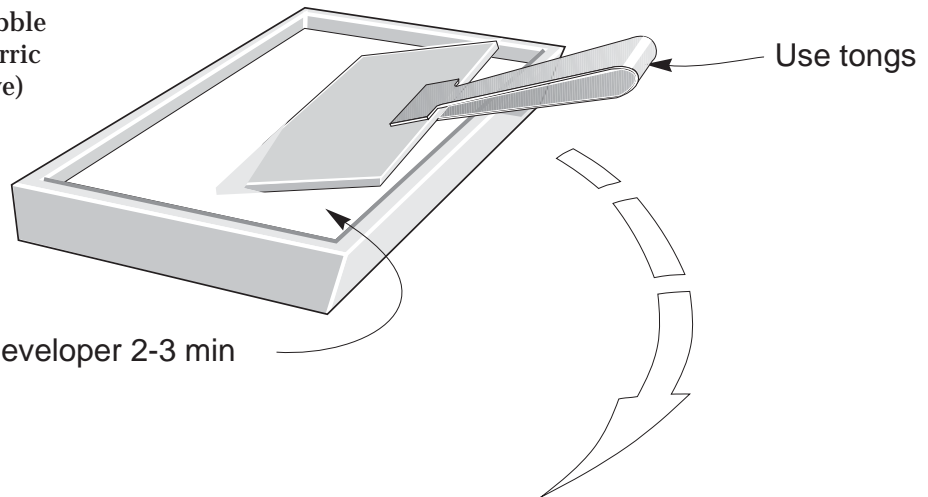
- Place the board, with the PCB mask in contact, face down on a UV light box. Close the light box and switch on for two and a half minutes.



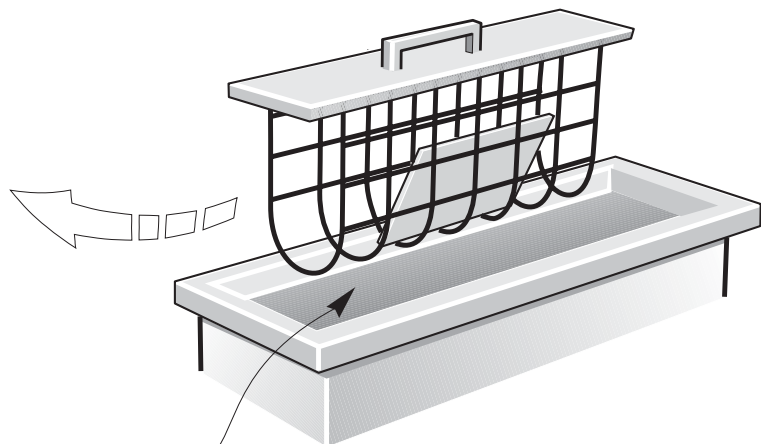
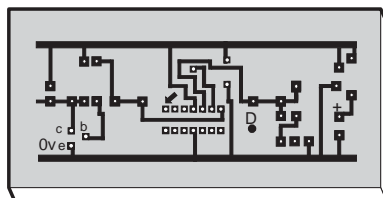
- Remove the board and film from the light box and immerse in developer to remove photo-sensitive layer. Agitate the board for 2-3 minutes until the circuit is clearly visible. The edges of the tracks must be clean.

- Remove the board from the developer and wash it under a tap.

- Place the board in a bubble etch tank containing ferric chloride to etch (remove) the unwanted copper. Inspect it at intervals and remove it from the tank when the etching is complete. This can take from 5 to 45 minutes depending on the strength and temperature of the solution.



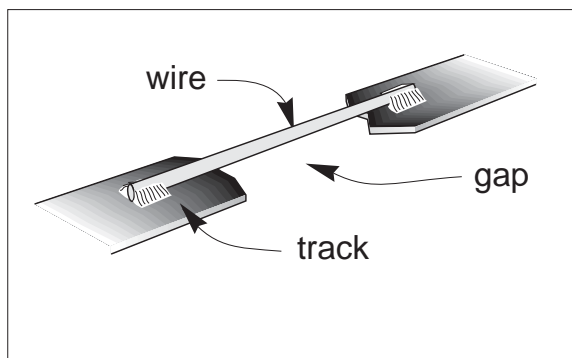
- Wash the board and then use solvent or a PCB eraser cleaner (RS stock number 555-308) to remove film from the copper tracks.



Bubble etch 5-45 min

QUALITY CHECK

- After your PCB has been etched, use a magnifier to check for bridges between the tracks.
- Remove the bridge carefully using a craft knife.
- Follow the same procedure for broken tracks and repair these by soldering a piece of wire across the gap.



SOLDERING

To solder a component onto the printed circuit board, follow these step-by-step instructions.

- Insert the leads of the component (sometimes called legs) through the correct holes in the PCB. The leads will stick out on the copper track side of the PCB.
- Some components can be held in place by gently bending their legs. Hold the component's legs with small pliers to prevent heat from the soldering iron damaging the component.
- The tip (bit) of the soldering iron should be clean and hot. Hold the tip against the copper track and the component lead to heat both through.
- After a few seconds, hold the solder wire against the joint and allow a small amount of solder to melt onto the track.
- Molten solder should flow around the lead at the point where it goes through the track forming a neat shiny joint.
- Electronics solder contains thin cores of **flux**. This melts on the joint and prevents oxidation. Without flux, the solder will not flow.

