

CUED

Electronics Development Group  
Tel : 32701

### **Advice on producing a PCB layout**

We offer PCB production and also PCB layout, please ask if you need assistance.

#### **Our Facilities**

We have two systems for producing PCBs

**WET** - This uses ferric chloride to eat away the excess copper. This we use for most PCBs, it is cheapest and quickest. We do not drill the holes or cut the board outline (though you can use our guillotine), through-plating is not generally available. We can accept any file format as we only need a printout. Smallest track and space we recommend is 13<sup>th</sup> track and 12<sup>th</sup> space, though we can manage down to 8<sup>th</sup> track and space.

**LPKF** - This is basically a machine that mills the excess copper from the board. This is more expensive and slower. We generally only produce PCB's PROTEUS or EASY PC files (due to the long setup times for each layout package). We usually drill holes and cut out your PCB (to the size you have specified in your layout). We can offer limited through-plating and a variety of PCB materials (thinner for high-frequency, flexible PCBs etc). Smallest track and space we usually do is \*\* though we can manage \*\* with a special cutter

- PCB's left for WET production before 1:00 will usually be produced by 1:00 the following day (Batches of PCBs will be treated as a job and may take longer).
- PCB's left for LPKF production - See nDavor Dukic or Richard Taylor for delivery time
- We supply PCB material and charge your job number.
- If you want more than one board producing it is easier for us if you tile them on a A4 sheet
- The largest WET size we usually work to is an A4 sheet (portrait) with a 1/2" blank margins all the way around.(We can work from A3, but this takes a little longer). For LPKF see Davor Dukic or Richard Taylor
- Label the Top and bottom layers if you are leaving us artwork so we can tell which way to make the PCB.
- Do not leave large empty areas on the board. If the board has large empty areas either make the board smaller or fill the board with an area of copper. (Large spaces waste etching solution because lots of copper has to be etched away).
- Reliability of production and durability when soldering is increased with track width, make your tracks as wide as possible

### Advice for beginners

**The following advice does not represent the limits of our PCB production, but is a suggestion for making a PCB that is easy to solder and work on.**

- Try to get your design to fit into the smallest area possible & keep tracks short.
- When using a PCB layout package the layout is viewed from the component side of the board.
- Produce a board outline so you have an outline when cutting the board to size.
- PCB's are laid out on a grid size of 0.1" (also called 100 thou - 100 thousandths of an inch). A standard DIL package (most IC's) have their pins spaced on a 0.1" grid.

### Tracks and Clearances

- Make track widths as wide as possible. (30 thou standard, 15 thou minimum).
- Make track clearances as wide as possible. (30 thou standard, 10 thou minimum).
- If you are using a two layer board and wish to connect tracks on the top layer to components either:-
  1. Use a small piece of track on the bottom layer that goes to a via or check that it is possible to solder to the top of the board using that component e.g. ( resistors OK, Electrolytic Caps, molex connectors and some IC sockets not OK).  
(A via is a small pin or wire that is pushed into the board and soldered on the top and the bottom to join both sides.  
Or if absolutely necessary we can produce an LPKF PCB with through-plating -  
Speak to Davor Dukic or Richard Taylor about this.
- If using vero-pins to connect to a PCB place the vero pins 0.3" apart to allow space to solder the wires.

### Ground and Power Supplies

- Power supply tracks should be as wide as possible. Lay the power supply tracks first, make them as wide as possible (30-50 thou).
- Ground connections can be made with ground planes or ground tracks.  
A ground plane is a solid plane of copper on the top or bottom of the board, all ground connections are connected to this. A ground plane produces a lower impedance ground path and is recommended for work with low level signals, work with high ground currents or high frequency work including high frequency digital logic.  
Ground tracks are just tracks that link all ground connections. If tracks are used make sure that they are as short and thick as possible.

### Pads

- Make pads as large as possible.(75 thou standard 65 thou min) to allow enough copper to be soldered to after a hole has been drilled through the centre.
- Don't forget to make the pads bigger if you are using components with large diameter pins eg. large capacitors or resistors, power connectors, PCB mounting transformers etc.
- When using packages with 0.1" pins spacing (eg. 14 pin DIL chips) use symbols with oblong rather than round pads as these leave more pad to solder to after drilling.
- If making a WET PCB make all inner pad sizes (the size of the hole in the pad) 15 thou. When drilling a PCB the drill bit tends to wander from side to side before cutting into the board. Having a small hole size reduces this wandering in the same way as using a centre punch on metal.

One other document that may be of interest is one titled 'Electromagnetic Compatibility for PCB design'. This covers enclosures, VHF, microprocessors, high frequency matched lines and switched mode power supplies. It also covers the impedance and inductance of PCB tracks. This document is available from Electronics Development.