Making Printed Circuit Boards

Bob House, October 2002

Develop your artwork with pc layout software from your advisor or group. Evaluation layout packages are available from Eagle (www.cadsoft.de/), Orcad (formerly MicroSim) (www.orcad.com/), Protel (www.protel.com/etech/trial_home.html), Cadstar Express (www.cadstarworld.com) and Electronics Workbench (www.electronicsworkbench.com). These evaluation packages may be limited in their library size, physical size or ability to output in Gerber format, so make sure you can finish the job before getting deeply involved.

If pc layout software cannot be located, AutoCAD compatible CAD files may be translated into Gerber files. This doesn't always work and it's difficult to do, so use as a last resort.

When the design is complete, output photoplots, Gerber or RS-274 files onto a floppy or email attachment to Bob House bob.house@ece.gatech.edu in Van Leer room W118. You will need one file for each layer (top, bottom, drill, outline, component placement, etc), plus an apertures file and drill rack. The nc (numeric control) drill file provides hole locations, but not drill size. The drill rack tells the drill file what size drill to use. The apertures file provides the gerber files (top, bottom, silk) with dimensions of layout features (ie - round pads = .050" diameter, square pads = .065", etc).

You can verify your gerber and nc drill files with GC-Prevue. Download is available from http://www.graphicode.com.

Preferences on circuit design layout:

Preferred trace width is > .030". Trace widths down to .010" can be made but the traces are fragile below .015" and sometimes break. Designing a simple board with .010" traces when wider traces could be used is not acceptable.

Standard trace separation is .031" where permitted. If more is needed (ie - high frequency isolation), it must be made known. Isolation greater than .100" is of limited value.

Minimum trace separation is .010". Minimum preferred trace separation is .015". Trace separations to .006" - .008" can be achieved with great difficulty.

Pads should be twice as large as drill holes. Don't ask for a .040" hole on a .050" pad because you won't be able to solder it: The holes are not plated through leaving limited copper surface to hold the component to the board.

Drill sizes are .025", .032", .040", .046", .055", .060" and anything greater than .062". I have an .018" drill bit that is very expensive and frequently breaks.

Vias are not plated through, they need to be hand soldered for top / bottom conductivity. Try to combine vias with components to ease the assembly process.

If holes for mounting hardware are desired they must be on the artwork.

Board outlines may be irregular shapes to fit enclosures or work around fixed objects.

Board material is available in double sided and single sided copper laminate. Multi layer boards are not available. The finished board will have a thin (~20um) layer of tin to slow the oxidation process and make soldering easier.

Common board materials are FR4 (.015", .032" and .062"), GIL Technologies MC5 (.020" and .030") and GML1000 (.020" and .030"). Dielectric constant is 4.8 +/- 1.0% for FR4, 3.25 +/- .05 for GIL MC5 and 3.05 +/- .05 for GIL GML1000. More exotic materials may be purchased and milled. Alumina substrate cannot be milled.

Copper thickness is 0.5 oz copper, or .0007" thickness. Higher thicknesses (.0014" and .0028") available by special order.

There is some good reading at http://www.merix.com/main_res.html and www.expresspcb.com/ExpressPCBhtm/Tips.htm

Best,
- bob

revised 7-17-01