



PCBTrain Printed Circuit Board Technical Guidelines

November 2007 issue 4

Purpose of this document

The purpose of this document is to help users of the PCB Train service get the best from our service.

BGA's

Cover all via holes under BGA's with solder resist. This prevents wicking of solder away from BGA pads whilst undergoing reflow. Failure to do this may make successful BGA placement difficult.

Blind & buried vias

These techniques should only be used if all else fails to solve the problem. A pcb with blind or buried vias will cost considerably more than a conventional 4 or 6 layer pcb. For smaller quantities it would normally be more cost effective to have an 8 layer pcb rather than a 4 layer with blind vias. We manufacture blind or buried via pcbs but not on the PCB Train service. Ask us for a specific quote.

Conventions

Place text on each side of a pcb to identify the nature of the layer. "Side 1" means a component side or top side. "Side 2" means a solder side or bottom side. Inner layers should be numbered starting from the inner layer adjacent to the component side. This would be inner layer 1. Never design symmetrical pcbs because of the risk of mirroring during production. Always put some text on each layer (right reading when printed) to ensure correct orientation.

Copper Layers

Copper pattern density should be balanced over each side of a pcb and from side to side. This improves plating uniformity and reduces bow and twist problems.

It is preferable to create all copper planes as solid rather than hatched copper areas. This applies to inners as well as outer layers. The reason is that large amounts of copper fragments are created after etching hatched copper areas. These fragments have the potential to cause short circuits. Hatched copper areas should be avoided.

Data files

Gerber files should be accompanied by an aperture-list, containing D-code-values and the shapes used.

Gerber files should be in Extended-Gerber-format with embedded apertures (RS274X)

Use flashed pads as often as possible.

- Use an aperture-list containing the same number of different apertures as there are used in your design.
- Output "pads" as filled.
- Indicate the used data-format.
- Indicate readability of the layers (preferably from component side, or solder side).
- Add a build-up drawing showing how all the layers "laminates" together.
- Don't mirror any layers. Send all layers as seen from top-side.
- Data files must contain just one type of pcb only.
- The pcb images may be step and repeated. We recommend that you zip your files with PKzip.
- If you upload your files on our website pcbtrain.com, the files must be in *.zip file format.
- All design notes must be placed in a .readme file and zipped up with the appropriate gerber data.

Drill files

The CAD design software default output should be satisfactory for CNC drilling. Ensure that finished hole sizes are included in the data. Do not make via holes unnecessarily small. A reasonable small size is 0.5 mm although smaller ones are possible.

File names

File extensions are used to identify each Gerber file. Protel give this advice with which we broadly agree. When you generate the Gerber output a series of files are created, each one corresponding to one of the layers enabled in the Gerber set up. These files are then loaded into a Gerber photo plotter, which produces the necessary photo tools for PCB manufacture.

Each Gerber file is given the name of the PCB document, with a unique extension that identifies that layer and plot type. For example, the Top layer Gerber file for a PCB called MyDesign will be saved as MyDesign.GTL, to indicate "Gerber Top Layer". Because each design normally generates numerous Gerber files, these extensions help identify each file.

We recommend that you follow this convention which conforms to general industry practice. The following table shows the extensions that are used:

Top Overlay	.GTO
Bottom Overlay	.GBO
Top Layer	.GTL
Bottom Layer	.GBL
Mid Layer 1, etc.	.G1, .G2, etc
Power Plane 1, etc.	.GP1, GP2, etc
Mechanical Layer 1, etc.	.GM1, .GM2, etc
Top Solder Mask	.GTS
Bottom Solder Mask	.GBS
Top Paste Mask	.GTP
Bottom Paste Mask	.GBP
Drill Drawing	.GDD
Drill Drawing; Top to Mid 1, Mid2 to Mid 3, etc.	.GD1, GD2, GD3, etc.
Drill Guide	.GDG
Drill Guide; Top to Mid 1, Mid 2 to Mid 3, etc	.GG1, GG2, GG3, etc.
Pad Master, Top	.GPT
Pad Master, Bottom	.GPB
Keep Out Layer	.GKO
Gerber Panels	.P01, .P02, etc.

' ©Protel International 1999. All rights reserved.

Surface Finishes

The standard and recommended pcb finish is **IS** (Immersion Silver) which is inherently lead free. Lead free **HASL** (Hot Air Solder Leveled) and **Au/Ni** (0.1 micron of immersion gold over 5 microns electroless nickel) are also available on some services.

Gold tags

We manufacture pcbs with gold tags but do not offer this on the PCB train service. Send us your data for a quote.

Inner layers

Identify inner layers with appropriate text descriptions printed in copper on the inner layer. The text should be right reading when viewed as printed on the pcb. So when you pick up a finished inner layer, you can read correctly say "inner layer 1".

Layer counts

Very broadly, costs of manufacture increase in proportion to the layer count.

Legends

These will be shaved back if they do not meet the copper pad to legend clearance requirements. For good quality assembly it is crucial to ensure there is no risk of legend ink being printed on pads.

Rout files

Do not generate a CNC rout file. However do ensure that there is outline information in the gerber data. The profile cutting line will be programmed to the centre line of the gerber line. Cut-outs and slots must be identified specifically with text annotation on your drawings. Note that the minimum radius of an internal corner (1.0mm) will be half the diameter of the standard router used (normally 2.0mm.) The thinnest slot will also be limited by the size of the router used (normally 2.0mm).

Solder resist

Identify solder resist layers with appropriate text descriptions printed in solder resist on the inner layer. The text should be right reading when viewed as printed on the pcb. Clearances will be increased if they do not meet the copper pad to solder resist clearance requirements.

Testing

Test files are created from gerber data. Therefore pads and tracks erroneously left unconnected in the gerber data will test as good pcbs. Visually check that all required circuit connections are present. Ensure pads are created from "flashed" shapes rather than from "painted" lines. It is the "flashed" shape which identifies the net end.

Tracking

Identify tracking layers with appropriate text descriptions printed in copper on the pcb. The text should be right reading when viewed as printed on the pcb.