

Q&A: Ask an expert

Edmund Geldenhuys has a BSc (Hons) in mathematics and is the senior CAM engineer at TraX Interconnect. e-mail edmund@trax.co.za



Edmund Geldenhuys

Question

What is the first thing to consider when designing a PCB?

Answer

Get to know your manufacturer; find out the specifications that they can work to (track spacing, track width, minimum hole size etc.).

Question

What production information must be sent to the manufacturer?

Answer

Provide complete and accurate production information (track layers, mask layers, legends, profiles, drill and paste files) – TraX prefer files in extended Gerber format or ODB++; provide a mechanical drawing with profile dimensions and hole-sizes; be clear about tolerances and special requirements; spell out all special requirements such as countersinking, castellated holes, edge plating, gold tabs, press-fit holes, special stack-ups, impedance control, conductive ink; make sure that the mechanical drawing shows: type of material, board thickness, copper thickness, mask colour, legend colour and final finish. This will provide a fast and accurate quotation. If there is any doubt about how to indicate any of these, your manufacturer will be happy to help. Never assume that the manufacturer will know what you require; remember that the manufacturer will probably not even know the board's end use. Make sure to pack (zip, etc.) each board's information separately with unique part number and revision number. This will make re-ordering of boards easier and quicker.

Question

What is the best final finish to have on a PCB?

Answer

The final finish chosen will depend on design technology, final use, cost constraints and assembly house preferences. General military use will require hot air solder levelling; BGAs need a flat surface finish so ENIG or immersion-tin are preferable. There are several other finishes in the industry so talk to your manufacturer.

Question

What information is needed for panelising boards?

Answer

Provide clear step and repeat information. Consult your assembly house before you panelise. Each one has their own preferred requirements and panel sizes. It's in your best interest to have efficient panels because panel size always affects the price, even more so on special materials. If the panel is to be v-scored, allow enough scoring clearance. On a 1,55 mm thick panel, TraX requires 0,5 mm from edge of board to closest copper. If the panel is to be routed, find out your manufacture's preferred router sizes. The thicker the board, the larger the router required. This is particularly important when dealing with metal core laminates. TraX prefers a 0,2 mm clearance from the edge of the board to closest copper. Indicate the type of tab for holding the panel together. There are many different types. Tab placement is important if you have components close to the board edge. Make this part of your design. Provide paste layers as your assembler will need a stencil to be made. TraX will send the paste file to you after completion of the CAM process.

Question

What are good practices to follow?

Answer

Always send production files which will load automatically into the CAM software, as mistakes can be made with manual entry, for example inputting apertures for standard Gerber files. Do not use the minimum manufacturer's specifications, unless absolutely necessary and then only where needed. Unnecessary use of tight specifications will increase the price and production time. When your board has a ground plane on the one side of the board and only a few tracks on the other side, copper balancing becomes critical. Filling empty areas with copper assists the plating and etching process and reduces "bow and twist". Avoid having a lone track at one end of the board and solid copper at the other end. That lone track is at risk of being over-plated. Discuss multilayer boards with special impedance requirements with your manufacturer beforehand. They might need special stack-ups. Also ask what standard materials the manufacturer uses. If you require a UL mark on your board check with your manufacturer what their UL specifications and requirements are. If at all possible, visit your manufacturer, meet their engineers and see their plant. The TraX door is always open for visits from customers, assembly houses and students.

Question

What is the procedure for correcting a customer's files?

Answer

It is always best for customers to correct their own files so that their boards can be made by any manufacturer.

We invite our readers to submit comments and/or questions, whether on this topic or any other area covered by EngineerIT, i.e. measurement & instrumentation, electronics, ICT, automation & control. Please send to annette.thompson@ee.co.za and we will do our best to have these answered by an industry expert.