I·CONNECT007
CONNECTING THE GLOBAL ELECTRONICS SUPPLY CHAIN

PCB Specification: What an Engineer Really Needs to Cover

Tuesday, July 19, 2011 | Pete Starkey, I-Connect007

"It really is inadequate to simply ask your supplier to take your design Gerber files and drilling drawing and to provide a green-coloured FR4 PCB." This statement was the issue debated during SMART Group's Webinar which focused on how to define a meaningful PCB specification and communicate it to the fabricator.

Introduced and moderated by SMART Group Technical Director Bob Willis, the Webinar included presentations by Technical Committee Chair Sue Knight and Vice-Chair Nigel Burtt, engineers with many years of practical experience in high-reliability electronic assembly and the measures necessary to control and assess quality, purchasing, design and production issues associated with PCBs.

Is it what I ordered? Nigel Burtt asked as he summarised the considerations that determine whether a PCB is fit-for-purpose in respect of materials, dimensions and tolerances, compatibility with assembly technology and end-product working environment, compliance with international standards, design and manufacturing constraints, cost constraints and cosmetic considerations.

Burtt was an enthusiastic advocate of designers and assembly engineers establishing good working relationships with PCB fabricators and taking the opportunity to visit PCB shops to appreciate the complexity of the manufacturing process and its capabilities and limitations, as well as understanding what information the fabricator needs to manufacture a design to properly fulfill the requirement, and how to communicate it effectively.

Beginning with the data package, Burtt explained how plot files in Gerber RS-274X format, supported by read-me files to explain details, were generally used to convey image information for copper layers, solder resist, legend, drilling and profiling, and how reference net-list information could be provided in IPC-365A format. But the Gerber files alone did not tell the full story, and needed to be supported by a fabrication drawing and associated notes along with a generic or specific purchase specification.

The fabrication drawing was necessary to provide the PCB manufacturer with a clear description of the mechanical requirements the design--dimensions, tolerances, profile, etc. and fabrication notes detailing the requirements and the limitations of the design. It would normally include a table of finished hole sizes and tolerances and a cross-section drawing showing the positions of all the layers, copper weights and dielectric spacing, overall thickness and tolerance.

The fabrication drawing would also define material specifications: Laminate classification and finished copper weight, solder resist type and colour, legend type and colour and solderable finish. Additionally, a typical fabrication drawing would indicate the preferred location for the manufacturer's identification, UL marking and date code and information concerning revision and internal change tracking, with particular reference to whether the drawing or the Gerber files constituted master data.

Burtt then discussed standards and specifications, referring to IPC-6012 Section 1.3.2 for general manufacturing requirements and IPC-4101 and 4103 for base laminate requirements. Regarding laminates for RoHS-compliant products, he suggested specifying CAF-resistant phenolic-cured material with Tg above 175ºC, Td above 340ºC and T-288 greater than 10 seconds, and nominating particular suppliers and grades where appropriate. Industry-standard specifications were also applicable to solderable finishes, for example IPC-4552 for electroless nickel immersion gold, IPC-4553 for immersion silver and IPC-4554 for immersion tin, and to solder resist: IPC-SM-840E, and he recommended that these be referenced on the fabrication drawing.

The customer's own generic purchase specification normally covered such topics as procedures and consents for deviations and changes, bare-board testing, packaging and shipping.

Having delivered this comprehensive overview of general considerations in specifying a PCB, Burtt handed the podium over to Sue Knight for a more in-depth view, with particular reference to developing supplier

relationships with regard to high-reliability designs, selecting materials for special purpose applications and planning for long-term supply chain security.

Knight considered early involvement with the supplier to be an essential consideration in the successful manufacture of a high-reliability and special-purpose designs. It was not unusual to single-source a design whose requirements were especially sensitive or critical, or where close cooperation was necessary in materials selection and the development of layer stack specifications, particularly in mixed builds where mechanical and compatibility factors had to be taken into account over-and-above nominal signal carrying characteristics.

She stressed the importance of qualifying alternative materials early in the design cycle to avoid expensive re-qualification or, worst-case, re-design, if individual materials became unavailable later in the product life cycle. Shelf-life considerations might not favour the holding of substantial stocks of obscure material for the long term.

A close working relationship with the PCB fabricator favoured rapid feedback of design-for-manufacture information and clear understanding and agreement of allowable modifications. A good knowledge of PCB fabrication processes enabled meaningful specification of critical constructional features, with the emphasis on knowing which features were actually critical and avoiding over-specifying the job to the point at which it became unmanufacturable.

Cost-effective material utilisation was a significant consideration with expensive special-purpose laminates, and panelisation should represent a carefully-considered balance between the physical requirements of the assembly process and the economics of the bare-board fabrication process taking available laminate sheet sizes into account.

The real-time Webinar format gave the opportunity for an interactive question-and-answer session, with Burtt, Knight and Willis responding to queries on dimensional tolerances and temperature ratings for laminates, current ratings for via holes, colour and thickness specifications for solder mask.

Once more, SMART Group has made a valuable contribution to the education of the electronics manufacturing industry, and those engineers, designers and purchasing officers who took about 90 minutes out of their busy schedules to attend this Webinar benefited greatly from the experience.

For more information, visit **www.smartgroup.org**.

Related Articles:

- EuroLED 2011: Solid-State Lighting Gains Momentum
- An Inside Look: ICT's 37th Annual Symposium
- PCB Imaging & Registration Focus of EIPC Seminar