

Ten Tips for Purchasing PCBs

By Marissa Oskarsen,
PCB Girls & Geeks

Want to get the most bang for your buck on PCB purchases? Four industry veterans with backgrounds in PCB materials, chemistry, fabrication and assembly provided their expert advice, often echoing each other's recommendations. If you want to maximize your PCBs' quality, price, performance, delivery or overall value, try following some of these insiders' tips:

Get a first article inspection. Testing a pre-production sample is essential to determining if the fabricator can meet your quality requirements. Mike Carano, Global Manager Strategic Business Development at OMG Chemicals explains "I see many fabricators fail badly when making a new part number. Not because the company is not of high quality, but because they actually lack experience with the design and material sets in question." He also suggests asking the fabricator to build test coupons to measure registration, via formation and PTH reliability.

Get pricing up front. The bottom line is exactly that - the bottom line. A myriad of factors go into PCB pricing, so it's best to know up front if the fabricator can meet your cost goals before you invest your time and money checking their quality or visiting their facilities. For help on determining fair pricing, download a PCB cost/square inch calculator at http://www.pcgandg.com/Pricing_The_Smoking_Gun.html.

If the pricing seems reasonable, Erik Bergum, industry veteran and former chair of the IPC Base Materials Committee advises giving it a closer look. He suggests "PCB purchasers should the know the set up and lot charges for tooling and test, and understand which charges recur and which ones don't. Before issuing the initial PO, capture the cost and delivery timing for the first set of boards and for the subsequent sets."

Always check the fab drawing. This is the place where materials, finishes, plating and any special directions are communicated. Pay attention to these callouts. Chrys Shea, PCB assembly expert and president of Shea Engineering Services, advises "If you have qualified a particular material for storage, processing or reliability reasons, make sure the words "or equivalent" do not appear in its callout. Those two words invite unwanted substitutions that will likely disappoint."

In some cases, especially when expensive metals are involved, more specificity is recommended. If ordering ENIG final finish, Carano suggests explicitly stating the following requirements: 150-220 microinches of Ni, with 150 as the stated minimum, and 1-2 microinches of Au. He also suggests calling out a minimum plating thickness of electroplated copper in barrels of 0.8mils or 20 microns, explaining that "Anything less should be cause for rejection. The 0.8mil minimum is derived from hundreds of

thousands of hours of reliability and field use data. This is how we build reliability into the hole."

Check UL certifications on line. Go to www.UL.com, and look in the lower right corner of the header for a button labelled "Certifications." Clicking on it will bring you to their on-line certifications library, where you can look up the fabricator's UL certifications. If you can't find your fabricator, or if they are not certified to manufacture the narrowest conductor width on your board, contact them to resolve the discrepancy. If your PCB distributor holds the certification instead of the fabricator, you should understand that, as well. As a customer, you have the right to know who is manufacturing your circuit boards and the source of the UL stamp on them.

Ask for dummy boards. Also known as X-outs, electrical test failures, profile boards or solder samples, bare PCBs with quality problems that cannot be assembled and sold are great for profiling, pick and place tuning, process verification, or other engineering experiments on the assembly floor. How to get them? Just ask. Lee Starr, Senior Applications and Quality Engineer for Tech Circuits, explains "Like all manufacturing processes, PCB fabrication is subject to yields that are typically based on design complexity. When a fabricator begins a lot of PCBs, they start more boards than the order calls for, based on their estimates of expected yield. While the good boards are shipped to the customer, the bad ones are reclaimed with little value to the fabricator. New Product Introduction engineers routinely ask for them, but production shops usually don't."

Shea adds "During the production life of a circuit assembly, it can be built on different assembly lines, in different factories, or even in different parts of the world. Due to considerable variation among reflow oven performance, a circuit assembly should always get reprofiled each time it is run on a new line, but this doesn't always happen - often due to cost constraints. The availability of profile boards at zero cost to the assembler vastly increases the odds of the assembly getting profiled and reflowed properly, minimizing solder defects and improving overall reliability."

Don't overspec it. If a little is good, a lot is better, right? Wrong. Adding an unnecessary "safety margin" to material properties may create unexpected issues. Bergum warns "If you over specify critical properties such as glass transition temperature (T_g), degradation temperature (T_d) or Z-axis expansion, you may find that the specialized materials often have sensitivities to processing, handling or storage that may negatively impact yield or cost. Your best bet is to specify materials that meet your performance criteria, and address any questions or uncertainties directly with the fabricator or laminate supplier."

Don't overdesign the board. While designers may be very good at electrical design and layout, they are not necessarily good at understanding the interactions of materials and machines, and certainly not plating limitations. Carano advises a DFM review by the fabricator to identify opportunities to reduce costs or improve yields. Starr seconds that opinion, and takes it one step further, offering "if you are using a new technology

such as 0.3mm BGAs, get advice ahead of time on layer interconnect strategies, target and capture pad sizes, fine lines and spacing. This will insure producibility at multiple suppliers and help keep the design cost competitive.”

Don't “design for panelization.” Designers often make assumptions about vendor panel size, and in an effort to manage unit cost, set the dimensions of individual PCBs to maximize the number of pieces that can be fit onto each panel. While their intentions are good, they may be disappointed to find their efforts are in vain.

Starr explains: “If multiple lamination cycles are required, the usable space on the panel becomes limited. The stretch and shrink of each heat/pressure cycle creates dimensional instability that can cause misregistration near the edges of the panel. As the number of laminations increases, the size of the ‘sweet spot’ in the middle of the panel decreases. The fabricators’ CAD department accounts for this when they perform the panel layout, using as much of the sweet spot as they can for each design, but avoiding areas around the periphery that might cause quality problems. Also, panel sizes vary in offshore fabricators so the shop's location becomes a factor. Designers get frustrated when they find out they compromised their desired PCB size or certain design characteristics to accommodate a panelization layout and their noble efforts were all for naught. Their best bet is to design the single PCB they want, and refer panelization and cost reduction opportunities to the fabricator’s CAD team for expert guidance.”

Don't expect high quality just because the shop has ISO 9001 certification. ISO certification means all the business’ processes are documented. It doesn’t guarantee high quality output from the operation. All it guarantees is that almost everything done in the operation is written down somewhere, and an auditor spot checked multiple areas to see how well the records reasonably matched the actual processes.

Certification indicates that a shop has a quality system in place, and everyone agrees that that’s a good thing, but they also agree that quality output depends on a great deal more than just the presence of a system. It should be considered a minimum requirement of any new supplier, but certainly not the only one.

Don't try to cut costs by cutting the PCB broker out of the loop. Broker/distributors combine the purchasing power of multiple clients, so they often have more leverage with fabricators than singular buyers, especially with off-shore fab shops. And while it's true that the brokers turn a profit as they turn your PCBs, odds are you will still pay a lower unit price than if you go it alone. Plus, brokers' leverage extends beyond pricing: you can often receive better quality, delivery, and response to last minute changes by sourcing through their established supply network.

Whether you are an OEM or CEM, designing PCBs or assembling them, in low volumes or high, these tips are fairly universal and apply “across the board.” And although each one is targeted to a specific goal of improving cost, quality or reliability, in the end they all help us to meet the overriding goal of any business - profitability.

Many thanks to Mike, Lee, Erik and Chrys for sharing their expertise and insights.



Marissa Oskarsen, aka The Printed Circuit Girl, owns The E-TEC Sales Network, a successful national rep firm that focuses on providing PCB solutions for almost any requirement. Email Marissa@pcgandg.com