<u>What's Your PCB IQ?</u>

Marissa Oskarsen & Chrys Shea, Printed Circuit Girls and Geeks

<u>Quiz #7</u>
DFM

Are your designs process-friendly, process-unfriendly, or downright process-hostile? Take the PCB Girls and Geeks 10-Question Pop Quiz to find out!

QUESTIONS

1. Why apply manufacturing principles at the product design stage?
a) to reduce costs
b) to shorten product launch times
c) to simplify sourcing
d) to improve quality and reliability
e) because your boss told you to
f) all of the above

2. What factors influence PCB fabrication cost?
a) PCB area (size)
b) layer count
c) trace width
d) hole diameter
e) final finish
f) all of the above

3. What's the finest line width that is considered standard and does not add extra cost to the PCB?
a) 5 mil
b) 4 mil
c) 3 mil
d) 2 mil
e) none of the above

4. What is the least expensive final finish for high volume production?
a) Organic SolderabilityPreservative (OSP)
b) Hot Air Solder Level (HASL) - tin-lead or lead-free
c) Electroless Nickel Immersion Gold (ENIG)
d) Immersion Silver (ImAg)
e) Immersion Tin (ImSn)

5. What's the minimum solder mask registration that is considered standard and does not add extra cost?

a) <1mil
b) 1mil
c) 2mil
d) 3mil
e) none of the above

6. What's the smallest pitch on high I/O BGAs that can be escape routed with standard via and interconnect technology?
a) 1.0mm
b) 0.8mm
c) 0.65mm
d) 0.5mm
e) none of the above

7. The diameter of a BGA pad on a PCB should be:
a) smaller then the pad on the device
b) the same size as the pad on the device
c) larger than the pad on the device
d) it doesn't matter

8. Keep out zones around BGAs are typically:
a) 50mil
b) 100mil
c) 150mil
d) 200mil
e) none of the above

9. When laying out designs with BGAs on both sides of the PCB, they should be located:
a) Back to back on opposite sides
b) Overlapping on opposite sides
c) No overlap on opposite sides
d) It doesn’t really matter

10. T or F: You can reduce cost by adding layers to your PCB?

ANSWERS

1. f. All of the above. *Including* because your boss said so. Even if you are so much smarter than your boss that you don't normally listen to him or her, then certainly you are smart enough to know that incorporating manufacturing considerations into your design is a great idea.

2. f. Wow, another no-brainer. If you got that wrong, quit now and jump right to the lowest scoring category.

3. b. 4 mil. Anything wider than 4mil won't add cost, but anything smaller than 4mil certainly will. Sometimes you can reduce cost by actually adding layers to increase line width. Hmm…are you suddenly reconsidering your answer to question number 10? Too late!

4. a. OSP. In high volumes, this is your lowest cost final finish. If you're running low volumes, however, the money you save on a low cost finish can get quickly eaten up in poor assembly yields. If the OSP-treated PCBs aren't stored properly, of if they sit on the shelf too long, or if there are more than a few days between the first and final soldering cycles, you can expect solderability problems.

For low volumes, HASL or ENIG are your best bet. HASL is not much more expensive than OSP, but it can cause assembly yield problems if your PCB design has fine pitch (20mil/0.5mm or less) devices. ENIG has great shelf life and is super "assembly-friendly." In small volumes, the cost impact is not significant when compared to the problems it avoids.

What about immersion silver and tin? Like OSP they're good for high volumes, but shelf life issues make them a poor choice for low volumes.

5. c. 2mil. Most fabricators are pretty comfortable holding 2mil tolerances on solder mask alignment these days. That means you can design with 2mil relief windows around your Non-Solder Mask Defined (NSMD) pads, and, *theoretically*, route traces within 4mils of your pads.

We recommend a slightly more conservative approach, however. When you play on the edge of a process window, you're just looking for problems. We suggest you step back from that edge just a bit - maybe 5 or 6mils clearance if possible? If you're at the bare minimum and the mask alignment drifts slightly past the 2mil mark, the edge of the trace will be exposed and probably short to a nearby solder joint. That's a defect that's hard to find and fix - the stuff that bone piles are made from. Nobody likes a bone pile, but everybody's got one. Don't let your designs end up there!

6. b or c. While it does depend on the number and location of the I/Os that need to be routed, and the line width you are using on your PCB, generally speaking, 0.8mm is the finest pitch you can route with traditional vias in an 8 layer PCB. Going to 0.65mm historically meant taking the leap to HDI – a costly jump with higher prices and lower yields.

Recently, Texas Instruments introduced 0.65mm BGA packages that can be routed without HDI layers. They're called Via Channel Arrays, and their design strategically depopulates specific balls from the BGA package to make room for standard vias on the PCB. They say you can escape route this 0.65mm, 423 I/O package in 2 layers, using 20mil mechanically drilled vias. Wow!

7. b. It should be the same size or it could cause premature joint failure.
When that chip gets cookin' and heats up the PCB and package, they expand at different rates, which puts a lot of shear forces on the solder joints. Those forces are applied to the ball-pad interfaces. Since stress = Force / Area, the smaller pad area experiences the higher stress. The higher stress area is where the solder joint will begin to crack and fail. Keeping the pads the same sizes equilibrates the stresses between the package and PCB interfaces.

We understand that this is more of a reliability question than a manufacturing question, but we decided to ask it anyway because it's really important, and your fabricator or assembler might not consider it their DFM reviews.

8. c. 150 mil. Why so much, you ask? To allow room for the hot gas nozzle on the rework machine. We all hate rework, but it's a fact of life in circuit assembly. No process is perfect, and while we don't like to talk about rework in polite company, we do have to plan for that contingency with every device we place.

9. c. No overlap, please! While its not impossible for your assembler, it's a real pain in their neck. Again it comes down to rework. If the automatic x-ray inspection flags a potential BGA defect, the defect has to be verified with a manual x-ray analysis and then reworked. If there's another BGA mounted to the back side of the board, it interferes with both the verification and repair operations. While it is possible to mount BGAs back-to-back or overlapping, it is a huge pain in the neck, and pains in the neck cost money. So keep your BGAs on opposite sides of the PCB separated whenever possible and avoid this potentially expensive pitfall.

10. True! Adding layers may de-densify your circuitry, allowing you to use wider traces and spaces, which give higher fabrication yields and can actually save money on your PCB. This might not always be the case, but the only way to find out is to consult with your PCB fabricator or broker.

SCORING

What's your DFM IQ? Give yourself one point for every correct answer, and deduct one point for every incorrect answer.

If you scored 8-10, your DFM is Design for *Money*! You understand that your product's commercial viability rests largely on its cost to manufacture. So you wisely invest the time up front to consult with your PCB fabricators and assemblers on the lowest cost design options, incorporate their suggestions wherever you can, and enjoy smooth product launches. Your fabricators love you, your assemblers love you, your supply chain team loves you, but most of all, your customers love you, because ***you deliver great value!***

If you scored 0-6, your DFM is Design for *Mediocrity*. Mediocre is the nicest thing anyone can say about your designs. Sure, you go through the DFM motions because your boss tells you to. But unfortunately you only incorporate the simplest of suggestions and summarily dismiss the rest as "unfeasible." It's a shame, really; you've got talent. You need to change that approach, or else your combination of arrogance and ignorance will keep your designs – and you – forever mired in mediocrity.

If you scored less than zero, your DFM is Design for *Misery*. Keep ignoring manufacturing process capabilities and soon you will impart misery on your board fabricator, your assembler, your customer, and eventually your own boss. You know what happens then - it will all roll downhill on you. You will wish you had been smart enough to apply at least one or two good manufacturing principles when you're explaining all those production delays and cost overruns. You'd better start updating your resume and LinkedIn page now, just in case.