

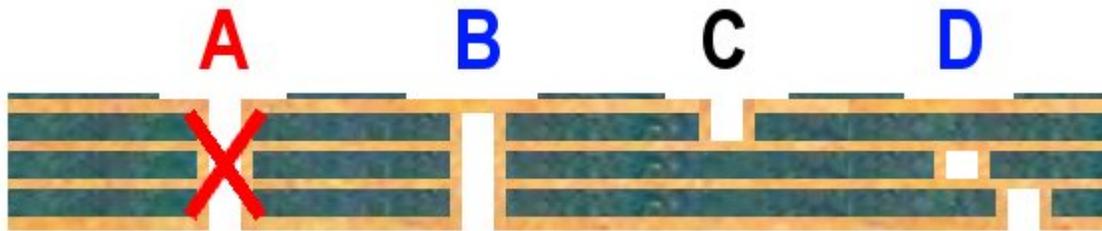
Five Via-In-Pad Myths

By Duane Benson, Screaming Circuits Marketing Manager

Is via-in-pad a viable design technique? Or is it a blight on manufacturing lines the world over? How about both? More likely, it's just another tool that, like all tools, can be either properly used or horribly misused. Let's examine some of the more common myths about the technique.

Myth #1: No. No. No. Not ever.

That's what many manufacturing folks say. They'll tell you not to put vias in pads. They'll say that it can't be reliably built so you should never do it. And, they'll say this despite the fact that so many chip companies virtually require via-in-pad with some of their new small form-factor components. In many cases, especially in the past, those naysayers would be correct, but today, there are enough proper techniques so that you can, and in some cases, *must* put vias in your pads. The key is to do it properly.



(A) shows an open via which we no one wants. If you must do this, give your assembly service provider a call first. There are things you can do to make it better or worse. Sometimes with leadfree solder and very small vias (6mil or narrower) you can get by leaving them open. But again, check with the folks that will be building it first. And, never do this with a BGA pad.

(B) shows one of our favorite options; the copper capped via, also called a blind via by some. You can have the via hole filled with a variety of materials at the board house. If this is done properly, we can't even tell that a via exists. This is the best option for BGAs.

(C) shows a blind micro via. This can work pretty well, but you should still call your assembly house first to talk about it. Sometimes the solder paste or the BGA solder ball can flow down into the microvia leaving insufficient solder to make a good mechanical or electrical connection. At the least, you will likely end up with a void. At the worst, you'll end up with an open. The smaller the BGA pitch and the larger the via diameter, the bigger an issue this is. Like (B), you can have the via filled and plated over for best results.

(D) shows the buried and blind via. In this case, the via is completely isolated from the pad so, as with (B), nobody really cares. I know I don't.

The best option we recommend for vias in BGA pads is to have them filled and plated over. You can have this done at the your board house for full vias or microvias. If done properly, it effectively takes the via out of consideration. It doesn't matter any more.



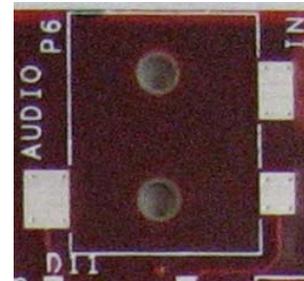
Just watch the results for good planarity. Sometimes the processing isn't nailed down and you can get either a indent or small bump over the filled via. The indent is probably most common when resin is used to fill the via. In either case, the bump or dip can lead to a less reliable assembly, especially with fine pitch BGAs and chip scale packages.

So, if you are going to put vias in your BGA pads, our first recommendation is to have them filled and plated over at the board house. If you do that, make sure you discuss it with them and see what their standards are for creating a planar surface.

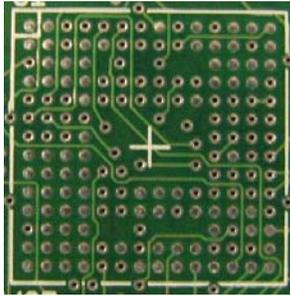
Myth #2: There aren't any good reasons to put vias in pads anyway

With some designs, if done properly, there really can be a number of compelling reasons for putting vias directly in the SMT pads for BGA and QFN packaged components, or even for passives.

- It can make routing easier with big or fine pitch BGAs. Once you get to 0.5mm pitch or smaller, you really can't route traces between the BGA land pads. There may be some rare cases where it works, but in general, it probably won't. You really can't route to vias between the pads either. Properly filled and plated over pads are pretty much your only option with these small geometries.
- It can allow really close placement of bypass capacitors. Right underneath the part is about as close as you can get.
- It can help with thermal management. On a lot of surface mount parts, you can't get away with plopping a heat sink on top of the part. That means the heat must be taken away from a thermal pad on the bottom of the part or from BGA balls under the part to a thermal pad on the other side of the PCB.
- It can help with grounding on high-frequency parts. Straight through the board to a ground plane is going to be your shortest path. That's important in high-frequency designs.
- Strengthening the pad attachment. Some surface mount connectors are subjected to enough stress to tear the pads off of the PCB substrate. In such cases, a couple of well placed vias may add just enough structural hold-down force to create a reliable product. Note the four vias in each pad in the audio connector land in the photo on the right.



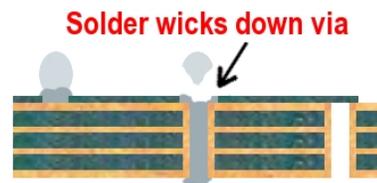
Myth #3: It's okay to put vias in BGA pads because the extra solder (the solder ball) on a BGA will prevent any problems.



Here's another example of a via in pad situation we don't like; A large number of vias in the BGA land pads. They're pretty big via holes too.

The primary reason we don't want to see vias in pads is that when left open, those via holes act like little capillary straws and suck solder off of the pad. A couple of undesirable events can happen depending on the method used during board fab.

If your vias are left open, solder will tend to wick down into the via hole. The larger the diameter, the worse the wicking problem can be. You might end up without enough solder left to secure the component, or even a solder bump on the bottom side of the board which could interfere with other components or lead to shorts.

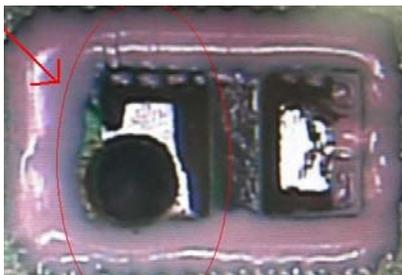


The extra solder in the BGA ball can just make a bigger mess on the back side of the board. And the extra space that BGAs have between the component and the PCB make opens even more likely. BGAs are more susceptible to this problem. Not less.

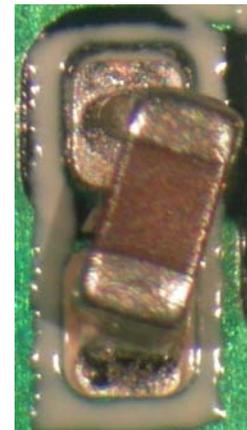
Some people will try to use soldermask to cap the bottom side of the via. However, the solder ball can still be wicked down into the via, leaving an open. Or the soldermask cap may pop off due to thermal expansion or out-gassing, leading to a purely open via. Internal air bubbles can migrate up, leading to voids in your solder joint.

Myth #4: It's not a problem with Passives

This footprint on the left will result in a minimal connection at best on the pad on the left. The via takes up nearly the whole thing so the pad barely has any metal. The open via will suck pretty much all of the solder down in it. There



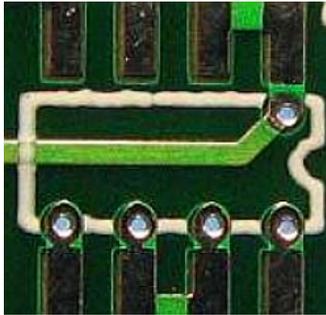
was plenty of spacing below, above and to the left on this board, so the easy solution would simply be to move the via in one of those directions far enough so that it's not in the pad and there is a web of solder mask between it and the pad.



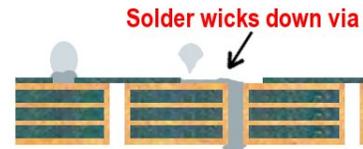
Via in passive pads can cause skewed parts, tombstoning, opens, solder muck on the back side of the PCB – a lot of problems. Maybe the parts aren't as expensive or glamorous as a big processor or fancy FPGA, but without the passives, you still aren't going anywhere.

Myth #5: It's only a problem when the vias are right in the pad. It's always fine if the via is off to the side.

Not so fast. Put the via right next to the pad and leave a clear metal path between the two and you may very well have just the same issue.



Here on the left is a pretty simple via-in-pad example that has both good and bad. Note pin 1. It does have a via connected to the lead land pad, but it also has a little strip of soldermask - a dam between the contact area of the land pad and the via. That little mask dam will stop solder from flowing into the via and everybody will be happy. Well, some people just have a sour disposition so they'll probably never be happy, but at least the assembled PCB will be happy.



Vias between BGA pads

Pins 5 through 8, though, are a different story. They have an unobstructed metal path from the land area to the via. It's quite likely that the capillary action of the via will suck the solder off of the lands leading to an unreliable connection. Since one side is all okay and the other is all not okay, you might see a tug-o-war of sorts with the surface tension of the good side pulling the chip too far that way.

My advice to you: Move the vias a hair further away from the pad and put a little soldermask dam in there, just like was done on pin 1.

On this BGA land pattern photo on the right, it looks like the pads are supposed to have soldermask between the pads and the BGA, but the registration is so bad that quite a few have a direct open path to the via. This is just a mess. Lousy registration. Mask opening too big. Open vias near pads. Yuck.



Myth #6: Via-in-pad causes cooties.

Well, this myth is true, actually.

The Screaming Circuits blog <http://blog.screamingcircuits.com> regularly discusses the issues listed here in greater detail. Feel free to visit and learn more. Screaming Circuits (www.screamingcircuits.com) is an industry pioneer in the online quick-turn prototype pc board assembly market. Located in Canby, OR, the company offers assembly, prototype and short-run production pc boards in as little as 24 hours for short runs with such advanced technology as 0201 passives, QFNs, micro BGAs, Package-on-package and flex assembly. Screaming Circuits is a subsidiary of the Milwaukee Electronics Corporation, which also offers full service contract manufacturing through facilities in Wisconsin, Oregon and Mexico, and design services through its subsidiary, MEC Innovation.