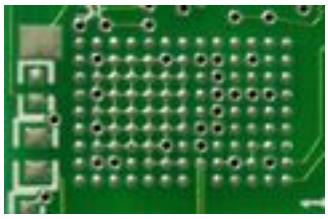
Random Via-in-Pad Myth #3

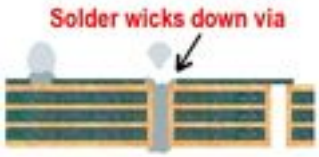
Screaming Circuits

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



[1] Here's an 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



[2] 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 map 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.

Duane Benson
Burma Shave

SOURCE [3]

Source URL (retrieved on *09/03/2014 - 5:02am*):
http://www.ecnmag.com/news/2010/09/random-pad-myth-3?qt-recent_content=0&qt-video_of_the_day=0

Links:
[1] http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef0133f4bf3588970b-popup
[2] http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef0133f4bf32f4970b-popup
[3] http://blog.screamingcircuits.com/2010/09/random-via-in-pad-myth-3.html