SCREAMING CIRCUITS ®

1140 NW 3rd Avenue
Canby, Oregon 97013
866-784-5887
www.screamingcircuits.com

Via In Pad guidelines

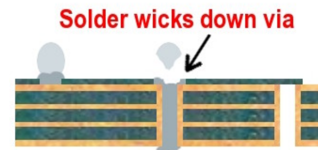
Via in pad seems to be one of the hot topics these days. It's a bit of a tough one too. The smt guys pretty much always say: "don't ever do it." However, with certain parts, the component manufacturer strongly recommends it. You gotta love those mutually exclusive requirements. Rather than just telling you "no, never", we're going to do our best to give a few guidelines on how to do it properly.

We don't like it, but with some designs, if done properly, there can be a number of compelling reasons for putting vias directly in the SMT pads for BGA and QFN packaged components.

- It can make routing easier with big or fine pitch BGAs
- It can allow really close placement of bypass capacitors
- It can help with thermal management
- It can help with grounding on high-frequency parts.

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.



- If your vias are capped or partially filled, the caps might pop off due to thermal expansion or out-gassing. Internal air bubbles can migrate up, leading to voids in your solder joint.

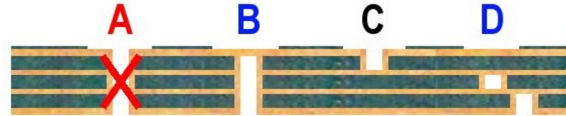
Open vias in pads are not an industry-accepted practice. In a perfect world, we'd like to never see one. However, the real-world is saying otherwise. Manufacturers of QFN parts are starting to recommend vias in the heat-slug pad for improved thermal conductivity. High frequency designs benefit from the shortest possible routing, which may indicate via in pad. Super fine pitch BGAs may not leave any other options.

Our first recommendation would be to re-design the boards so the vias are in between the pads, plug or cap the vias at the board house or to use microvias that don't go all the way through the board. If you can't do any of that, we would suggest that you use as small a diameter as the design will allow and follow the component manufacturer's guidelines for placement and via capping or filling.

Vias in BGA pads.

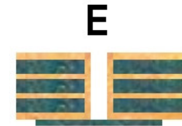
The BGA pad via is typically a more difficult situation than the QFN because there is less room. In general, our advice regarding via in BGA pad is: "don't do it. Don't do it. Don't do it." Oh, and did I say: "don't do it"? However, there are some applications that simply require it so just saying no may not be all that helpful. If you don't absolutely need to put the vias in your BGA pads, don't, but if you don't have a choice, here are some guidelines:

- B) or D) Best: Have your board fab house plug the via and then plate copper over it. They can plug with metal or a thermally and electrically conductive epoxy before the final plating steps. This will give you all of the benefits of via in pad without causing problems in assembly. It works for prototypes and for production. This is industry best-practice.



- C) Okay, but still a bit of a challenge: Use a micro-via that only goes through one layer of the board. This can still cause some problems. The solder can wick down into the via chamber. The stencil aperture may need to be enlarged slightly to ensure enough solder stays on the pad.

- E) Not too horrible, but not good either: Cap the underside of the board with solder mask. This will usually stop the solder from completely wicking out, but sometimes the cap can pop open and sometimes the void is big enough to still suck too much of the solder off of the pad. It can even suck down the solder ball off of the BGA leaving an open. Call us before you send in a board like this. We need to look at it to tell you if we can reliably assemble it.

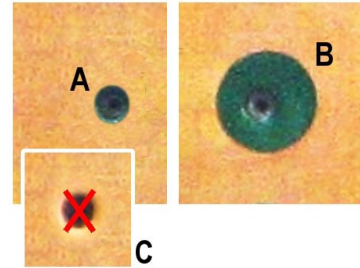


- A) You should never do this: Open vias on the pad. This goes against industry standard practice. We have built some boards like this but there are no guarantees and, if we can do it, it will probably cost extra. The problem is that the via, through capillary action, sucks the solder off of the pad and may even suck the ball off of the BGA. Sometimes, no matter how much solder paste is applied, it still all gets sucked down to the other side of the board. Using really tiny vias or lead-free paste may help but our advice is to see above. If you don't have a choice, go ahead and give us a call, but we may have to say no or we may agree to give it a try without guarantees.

We want to be able to help in any situation that you need and sometimes the best and least expensive option may involve a re-layout with a more expensive board. Think of the cost of those expensive BGAs and the potential cost of having to re-build it.

QFN component-side solder mask Via Caps

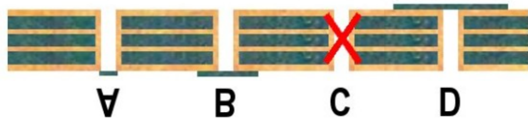
Component-side solder mask via caps aren’t the best way to deal with a via in pad situation, but it is the easiest and can work reasonably well for QFN parts. Don’t do this with BGAs though. You’ll get voids under the solder balls and the connections will not be reliable. Almost any board house can pull this one off with a reasonable chance of success. In the example below, what we're looking at may be the center pad for a QFN, the metal tab on a D2Pak or a similar smt component with a wide copper area that requires vias for thermal relief or grounding.



In example A, the via is capped on top with solder mask just barely larger than the via itself. Most manufacturers recommend that the solder mask cap diameter be 100 - 125 um wider than the via to minimize voiding and thermal insulation. Interference with heat transfer is minimal in this example.

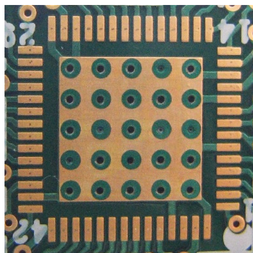
In example B, the via is capped with a wide solder mask circle. This is a decent method for non-critical applications. It is easier to do at the board house and should be adequate in most cases other than extremes of signal or thermal sensitivity. It will insulate a bit so you won't get the maximum heat sinking. With these techniques, the solder paste stencil must be segmented so that solder paste will not be deposited on the solder masked areas.

Example C shows what you should not do - leave the via completely open. Capillary action will likely cause most of your solder to end up on the bottom of your board instead of securing your chip. With very small diameter vias, lead-free paste and careful stencil design you may be able to successfully use this design. If you absolutely need to do this, call and talk to our engineers first.



Example D is the least preferable (after C) method. Capping the bottom side will usually keep the solder from dripping out onto the bottom of the PCB, but several other problems can occur.

Outgassing from the solder paste can cause the cap to pop off, leaving you with an open via. Solder, especially leaded, can still drain down into the via. You can end up with too much voiding under a QFN. This can work, but, as with C, call us first and talk to an engineer.



Here is a real-world example of technique B, from above. This is for a 9mm X 9mm QFN on a board using an ENIG (Electroless Nickel Immersion Gold) finish. The larger size of this part gives a little more flexibility than some of the real small parts. In this case, while the vias are left open, the area around the vias is masked to keep solder away.

This method can work well on larger QFN and QFP parts, but it is absolutely critical that your solder paste stencil be segmented and not put any solder over the masked parts. If the stencil is left fully open or deposits solder paste over the vias, that solder may go down and mess up the bottom side of the board. With a well designed stencil, this method may be an easy and reliable method for dealing with your thermal vias.

One more thing on the plugged or capped via in pad -

If you do this, ask your board fab house about the flatness of the resulting pad. Some manufacturers will plug and plate over or cap a via in a pad but not deliver a flat surface.

The pads might have an indent or a bulge. This can be a problem, especially with the smaller BGA with .5mm or smaller pitch. The flatness of the surface is very critical with these parts. An indent can lead to insufficient solder under that ball and a bump can tilt the part, leading to poor connections under other balls. Talk with your board fab house and make sure the surface is flat and flush.