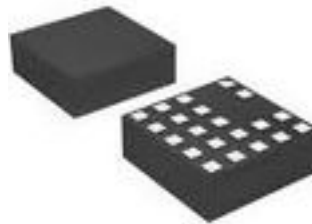


1.0, 0.8, 0.5, 0.4, 0.3...

Screaming Circuits

As pitch gets smaller, things get more difficult. Did that blinding flash of the obvious hurt your eyes? Mine too. It's not always just more difficult though. Sometimes it's different as well.

Like BGAs, for example. With 0.5mm and larger pitch, we and pretty much everyone else recommend NSMD (non solder mask define) pads. [However](#) [1], once the pitch drops down to 0.4mm, some manufacturers are



recommending solder mask define pads to prevent bridging between solder balls. Make sure your fab house can nail the mask registration.

With [LGAs](#) [2] and QFNs, IPC recommends NSMD pads for 0.5mm pitch and larger. Once the LGA or QFN (DFNs too) pitch drops below 0.5, does everyone suggest solder mask defined pads like with BGAs? No. Actually, IPC tells you to go ahead and remove the solder mask web between your land pads with 0.4mm pitch parts. With spacing that close, the solder mask can't reliably stick to the PCB. That would be messy. As to why a solder mask defined pad would not be a good thing, I can only speculate at this point. My assumption is that without the solder ball like the BGA has, the whole land area is needed to ensure a good solder joint.

Duane Benson
Fight Uni!

[SOURCE](#) [3]

Source URL (retrieved on 02/01/2015 - 10:16am):

<http://www.ecnmag.com/news/2010/05/10-08-05-04-03>

Links:

[1] <http://blog.screamingcircuits.com/2009/04/04mm-pitch-bga-pads.html>

[2] http://blog.screamingcircuits.com/2007/01/soldering_lga_p.html

[3] <http://blog.screamingcircuits.com/2010/05/10-08-05-04-03.html>