

A Bit More On the LGA

Screaming Circuits

After my last post about [LGA land patterns](#) [1], I received a couple of questions asking for more detail in a few areas.

"The LinearTech LGA apnote (LTM46xx series) shows planes on the mounting layer interconnecting pads that are solder mask defined. This is supposed to be for heat dissipation. Will smaller copper defined pads and vias to full internal copper ground and power planes provide adequate cooling?

What about using LGAs on the same layer as BGAs? BGAs have copper defined pads? We've been sending 1:1 soldermask gerbers to the fab house so they can adjust per their process. Can this be done selectively so the SMD LGA pads don't grow bigger? What kind of Fab Note should be in the "Readme" file?

Also, please warn LGA users to be careful using wizards (eg Pads Layout) to generate the pad numbering. Linear Tech's LGA does NOT follow the standard BGA alpha numeric numbering. I don't know about other LGA mfgs numbering systems but ... Double check the pad numbering and avoid this nasty snake bite!"

First, as far as cooling goes, the answer, unfortunately is "it depends on how closely to the limits you are driving to part." You will get best results with more surface copper. That being said, you can use vias to internal and back-side planes to increase heat dissipation. Ideally, you would have Lot's of surface copper and vias to the internal and back side planes, but that's not always possible. The vias that are not under the LGA pads can be left open. Any vias in an area to be soldered must not be left open. Ideally, you would have them filled with a thermally conductive material and plated over. You do have some flexibility to reduce the surface copper and replace it with vias to other planes, but ultimately, the final answer will only come from your design testing.

You can have NSMD and SMD pads on the same PCB. How to do it is the big question here. Many fab shops will make their own decision on what is "best" for your PCB in this regard. I would speak with the board house and get their recommendations on how best to specify what you need in terms of NSMD and SMD mixed. You'll probably have to follow a slightly different procedure for each different fab shop.

I would double echo the comment about using caution when using wizards to create a land pattern. Not all manufacturers follow the same numbering scheme. You could get bitten badly with this one.

Duane Benson

Who was that soldermask defined man?

A Bit More On the LGA

Published on Electronic Component News (<http://www.ecnmag.com>)

[SOURCE](#) [2]

Source URL (retrieved on 07/30/2014 - 3:50am):

<http://www.ecnmag.com/news/2011/01/bit-more-lga>

Links:

[1] <http://blog.screamingcircuits.com/2011/01/what-about-the-lga.html>

[2] <http://blog.screamingcircuits.com/2011/01/a-bit-more-on-the-lga.html>