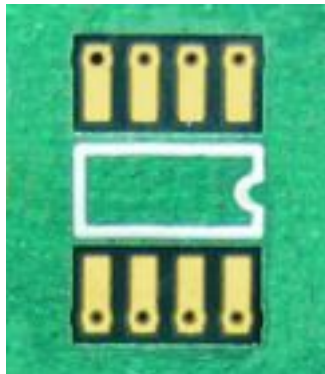


Via in pad x 8

Screaming Circuits



[1]Here's an interesting via in pad case. On the one hand, the footprint is very symmetrical and clean looking. On the other hand, it has open vias in the pads.

At first glance, I thought this was a DIP footprint with extra long pads, but it's not. It's for an SMT part. Personally, I would have put mask between the pads. Looking at the rest of the board (not shown), the spacing between pads and mask is pretty wide, so there may be a good reason. I'm not sure though.

Definitely, though, I would not put the vias in the pads like that. Those open vias will cause solder to flow down to the other side of the board, make a mess there and leave the chips without sufficient solder.

Duane Benson

Sucking solder through a straw - or via

Source URL (retrieved on 01/27/2015 - 9:17pm):

<http://www.ecnmag.com/blogs/2012/09/pad-x-8>

Links:

[1] <http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef0177448e6816970d-popup>