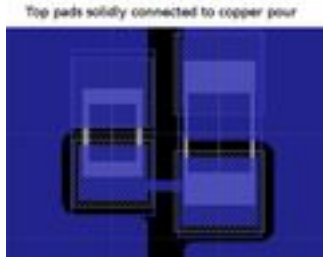


Pads on ground plane

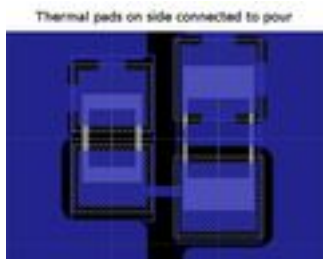
Screaming Circuits



[1] Generally, small pads for passive parts are connected with a single PCB trace of equal size to each pad. That's the right way to do it.

However, sometimes, circumstances dictate a little different approach. The illustration on the upper right here shows something of a worst-case. This is for a snubber (resistor, capacitor pair) between two power planes.

A couple of things will likely happen. The power plane will act as a heat sink, preventing the solder paste on one side from melting, resulting in a poor connection. Or, the unequal melting could lead to surface tension pulling the part up, causing tombstoning.



[2] Most designers are aware of that, but sometimes, thermals will be deliberately turned off to allow for better current capacity to and from the large power Mosfets (not shown). If that's the case, make sure that you can turn the thermals (see image on lower right) on or off by the part, rather than just by the plane.

Duane Benson

The rain falls mostly on the ground plane due to static attraction

Source URL (retrieved on 02/01/2015 - 8:01pm):

<http://www.ecnmag.com/blogs/2013/08/pads-ground-plane>

Links:

[1] <http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef01901eec1b15970b-popup>

[2] <http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef0192acab560297>

Pads on ground plane

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

0d-popup