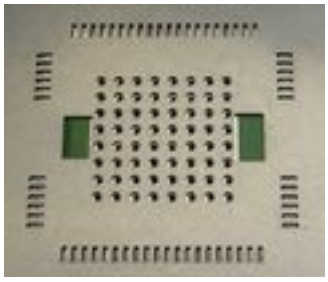
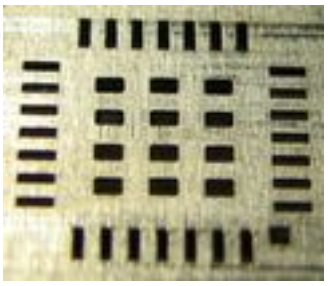
Via-In-Paste

Screaming Circuits

[1] 

[2] I've heard of vias in pads, but vias in paste aren't so common. Here's a QFP footprint with a lot of thermal vias in the center pad area. I'm guessing all of those vias are necessary for heat removal, but they sure aren't friendly to solder paste as created.

Further, the solder paste stencil had good intentions too, but ended up inverted in practice. It's good to keep paste off the vias. Even inverted though, I'm not sure how the stencil would have worked. You can't have dots of stainless steel just floating out there. The vias should be capped, or better yet, filled and plated over. The proper way to create the stencil layer would be with a cross hatch as shown in the third image. 

[3]

Capping (also called tenting) vias with solder mask used to be a pretty common practice. You'd put solder mask over the via out to about 100 microns bigger than the via on the component side. Some people would put the cap on the back side, but that still leaves a lot of volume for solder to wick down into.

The capping with mask practice is kind of falling out of favor these days. For one, if the mask is too thick, it can end up lifting the part up a bit. More likely though, the

caps can break, leaving an open hole for solder mask to wick down.

The optimal method is to have the vias filled and plated over at the board house. There are some thermally conductive materials that do just about as good at heat transfer as would the open via.

Duane Benson
Solder paste is people! It's people!

SOURCE [4]

Source URL (retrieved on *06/19/2013 - 11:53am*):
http://www.ecnmag.com/news/2010/08/paste

Links:
[1] http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef0133f3716376970b-popup
[2] http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef0134869565e5970c-popup
[3] http://screamingcircuits.typepad.com/.a/6a00d8341c008a53ef0133f37165dc970b-popup
[4] http://blog.screamingcircuits.com/2010/08/via-in-paste.html