More Reliable Assemblies Through Via Post-Processing

By Jim Julian

August 13th, 2014

Vias have traditionally been viewed by the electronics industry as a 'necessary evil'. Starting from the mid-20th century when double-sided boards came into use, mechanical eyelets first allowed signals to cross over each other without the use of jumpers. But they were costly to install and prone to failure.

When through-hole plating technology evolved in the 1960's, it became possible for vias to be fabricated as part of the bare PCB. This allowed for more complex routing and multilayer designs, but vias were still a common point of failure. In addition, fab shops charged for every drill hit. Thus, even in the CAD era of the late 80's and early 90's, designers would still spend hours poring over their routing, trying to eliminate a few vias from a layout.

As fabrication processes became more refined, the cost of using vias (in terms of both dollars and reliability) went down markedly. Today a designer need not worry about using vias as necessary (except in high speed signal applications, but that is a topic for another day). Indeed, vias have become an important power distribution and thermal tool, to be used extensively where there are few or no through-hole components on the pcb.

Where to put them?

For the purpose of this paper, we're discussing through-hole vias only. Partial and micro vias are a whole different animal and will be discussed in a future paper.

Traditionally vias needed to be spaced well away from through-hole pads and surface mount lands. As an open conductive surface, they posed a threat of bridging and shorts. This started to become a problem as board real estate began to shrink. Designers started moving them closer to other features to save space. Sometimes they would put them directly inside surface mount lands, and is commonly referred to as via-in-pad.

Problems during assembly

A number of problems crop up due to the proximity of vias to other circuit features.

1. Via-in-pad immediately became a problem at assembly shops. Solder paste deposited on the surface mount land melts and bleeds down into the via during the reflow process, resulting in a solder-starved joint.

2. A similar problem occurs when a via is so close to a surface mount land (of the same net) that there is no room for a soldermask bridge in-between. This via will also rob solder paste from the land and degrade the joint.
3. Vias still pose bridging and shorting problems when they fall in close proximity to an open conductive feature of a different net. In most cases this is a nuisance requiring rework. But it becomes a real showstopper when it occurs underneath a bga or other part, as the joints cannot be fixed after soldering.

Via post-processing to the rescue!

The above problems are not going away. Board real estate continues to shrink. Via-to-pad-same-net design rules do not exist, or are turned off by default in several ECAD packages. Some designers simply don't think about via proximity to lands. And EE's are always pushing for vias to be closer to lands for electrical reasons.

Fortunately, many fab shops started to come up with creative ways to remedy the problems cause by via proximity. Some of these methods have evolved into industry standards. It is important to stay informed about these methods, as they can turn an unreliable assembly into a reliable one.

One of the earliest methods to deal with via proximity problems has been dubbed *Tenting*. Often referred to by the misnomer 'tinting', via tenting involves applying soldermask over the entire via surface, covering the hole like a tent, and removing solder robbing or shorting potential when vias are very close to lands.

Tenting worked acceptably when dry-film mask was in popular use, because this material has some mechanical strength. With the widespread adoption of LPI mask however, tenting was quickly recognized as causing more problems than it solved. When the mask is applied to both sides of the PCB, an air pocket forms in the via hole, and this air pocket expands during reflow heating, often exploding the mask outward. Even when the tenting was done only on one side of the board (for example, under bga parts), the hole cover often flecked off, re-depositing itself elsewhere and possibly contaminating other parts of the assembly. In addition, the one-sided tented hole becomes a bucket that can fill up with impurities, contributing to the degradation of the hole wall plating. No, tenting is *never* a good idea anymore, especially with several better options available.

A more civilized alternative to via tenting is to cover the via annular ring with mask. It still reduces solder robbing and shorts. Because the hole is left open, none of the dangers of traditional tenting apply.

This method works well when there are many vias near lands, and there is not enough room for a mask dam in-between.

Another option is to plug the via with non-conductive material. The plugging material is similar to soldermask and bonds with it well. Note that the plugging process usually does not completely fill the via hole. But once the material is cured, it is very stable.

In this case, mask can be run over the top of the via, completely isolating it from the outside world. This solves the same problems mentioned above, and is excellent around BGA's or where the vias are placed extremely close to lands of the same net.

However, once the vias become tangent to lands, or actually begin to encroach into them, a newer and even more elaborate method is called for.

Via filling / capping makes the via completely invisible to the assembly, as if it were not even there. In this method, via holes are completely filled with the same resin used for plugging, mentioned above. Then a copper cap is applied over top of the via hole. Finally, the entire via is plated over with the final surface finish, usually ENIG. It becomes coplanar with the rest of the land surfaces, so even vias completely inside lands will not affect reflow soldering.

This newer method is not accessible by all board shops, and does include added processing and expense. There is also the chance of the cap losing its bond and coming off. However, in our experience, this method has proved an excellent solution where vias encroach on, or are completely inside lands.

Applicad has the experience to use the right method.

There are many subtleties, exceptions and caveats to the above. When we at Applicad receive artwork from a customer, we analyze it to determine the best method of via post-processing. Then we accurately convey our intent to the fabrication shop using their own lingo. This results in quality boards that solder well and are thus more reliable. It is just one of the many value-added services done behind the scenes, allowing us to build challenging designs well, where some others have quality problems.

Feel free to ask us about via post-processing in more detail, and how to make your designs the best they can be.