

“Tech Talk” for Techies Via Interconnect Holes

“Tenting, Capping, Plugging, or Filling - Which is required by My Design?”

There once was a time when the majority of plated through holes in a printed circuit board had component legs that were soldered within them, but with the advent of surface mounted component technology today's majority of plated through holes are simply for interconnection only--they will be devoid of any solderable pins and the holes may be left open, covered, plugged or filled.

This issue of “Tech Talk” for Techies is intended to provide some basic guidelines in making the best choice.

The main purpose of a via interconnect hole is to provide electrical continuity between the circuit traces that it connects, by simple conductivity through the plated barrel of the hole. As there are no components inserted and soldered within, the actual diameter of the barrel is basically insignificant, and in most cases is designed to be as small as possible while meeting the annular ring requirements of its associated pads.

In the days when wave solder machines ruled the land, they would typically fill with solder, but in the process of doing so sometimes filled irregularly and would provide a bumpy raised surface that resembled Braille on the finished circuit card. This typically was not a problem, but in the modern surface mount component world, absolute surface planarity is more important as adjacent low-profile components may be affected by a surplus of solder. So a certain amount of control is required when manufacturing a flat-surfaced PCB.

Here are the options offered by Advanced Circuits:

1. **VIA TENTING/CAPPING:** One easy way to prevent excess solder in via holes is to cover them over with soldermask, otherwise known as **tenting**. For via holes associated with a ball grid array (BGA) device, the plated holes are typically designed on a "dog bone" pad structure which allows the via hole to be in close proximity to the soldered ball pad, where absolute planarity is required.

By covering them on the device side with solder mask material (achieved by omitting a clearance around the associated via pad) the hole can effectively be "tented" over by the mask thus preventing solder from filling the hole, or being drawn to it by the soldered ball pad nearby.

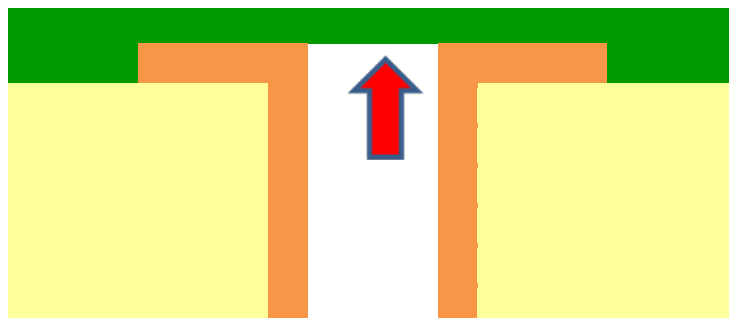




Figure 1: Soldermask Tent Across Pad

However, there is one manufacturing issue associated with tenting a via hole with soldermask. If the hole is covered on one side of the printed circuit board (PCB) then wet processing after the soldermask may allow small amounts of fluids (such as the surface finish process or PCB cleaning solutions) to remain in the barrel of the hole.

As some of these fluid residues may be corrosive to the copper plating over time, it is best to provide a small "pinhole" or "tent-breaker" in the center of the soldermask tent to allow solutions to flow through and escape the hole barrel. The holes are typically small enough to allow fluid movement but will impede solder, so the tent remains functional. Overall, this method is the most cost-effective way to deal with via hole covering, as the tent/breaker feature is a part of the soldermask operation and does not require additional processing steps.

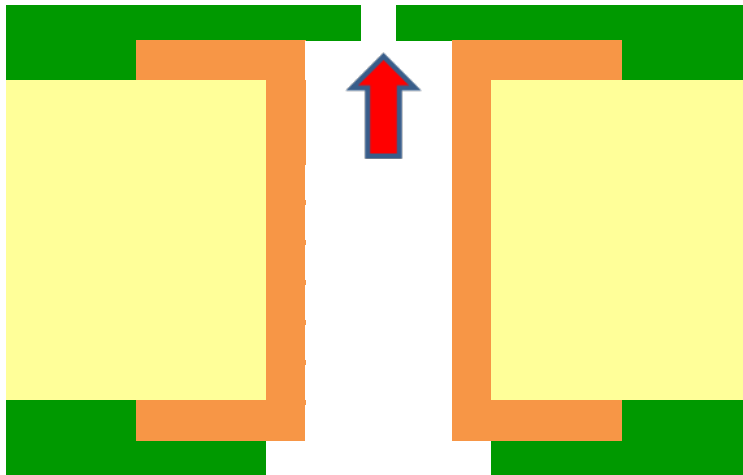
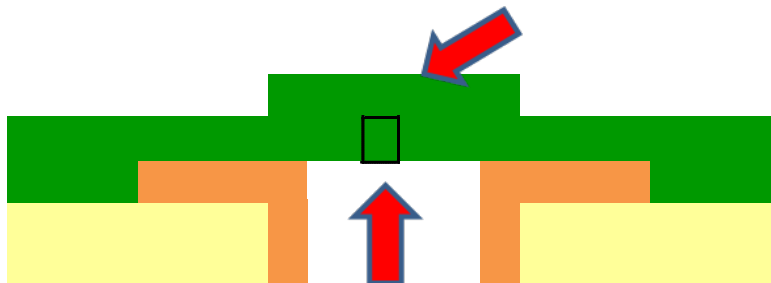


Figure 2: Soldermask "Pinhole" in Tent

Note: If the pinhole is not desirable in the finished PCB, or if it is preferable to completely cover the hole on the one side, the pinhole may be covered in a secondary operation known as **VIA CAPPING**. A small "cap" of soldermask can be silkscreened on top of the pinhole to completely seal and cover it. This does of course add a secondary operation similar to via plugging.



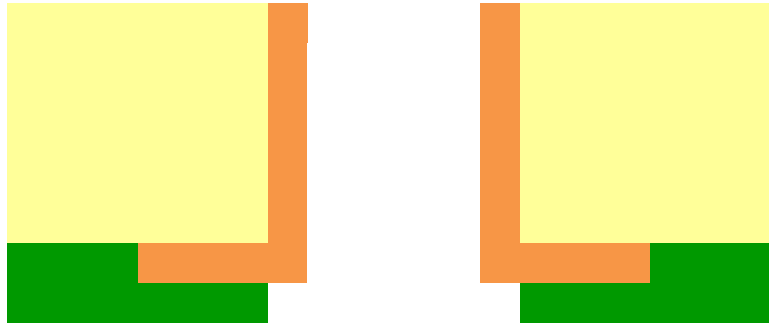


Figure 3: Soldermask "Cap" Over Pinhole

2. **VIA PLUGGING:** The via hole cap over the pinhole and via plugging operations provide essentially the same result, as they both involve a secondary process where the cleared hole is screened with a selective "dot print" of mask material which effectively caps or plugs the hole after the surface finish operation (such as electroless nickel immersion gold or hot air solder leveling). This plugs the hole to avoid solder draw during assembly but allows surface finish process solutions to flow through and out of the hole barrel prior to the screening process. It also provides value during subsequent in-circuit testing if a vacuum is required as the mask material effectively blocks the hole barrel (additionally preventing a majority of light to shine through as well). Note that the via plug is used with a soldermask layer that initially surrounds but does not encroach on the associated pad (typically having a 0.002" or more clearance). This provides a completely clear hole to screen the soldermask material in to, unlike the pinhole which is small and the soldermask sits on top in a "cap" configuration.

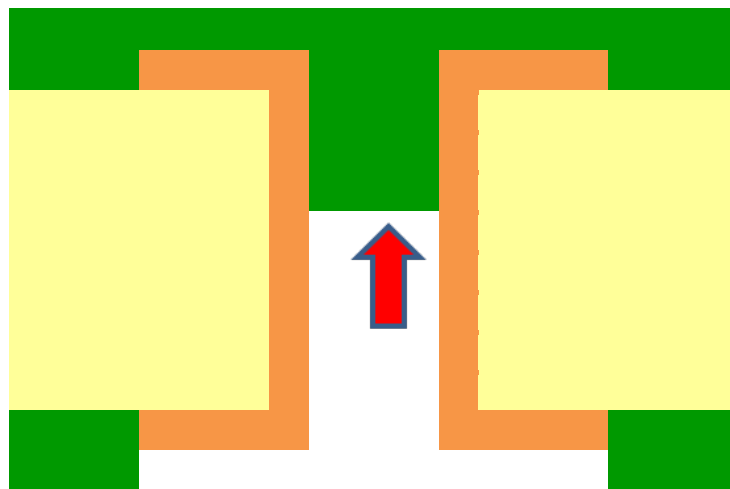


Figure 4: Soldermask Plugged Via

Of course all of this depends upon the actual amount of mask material that is screened into the hole--typically it will not exceed one-third of the hole depth. As there is one additional process this operation is slightly more expensive than simply tenting the hole during the soldermask process.

3. **VIA FILLING:** Finally there is the method of via filling (this process is most commonly used for Via in Pad, or VIP), which involves many more processes and is therefore naturally much more costly. However, it provides many benefits for modern designs as we will see.

The most useful benefit of hole filling is that the vias may be located in a solderable pad, as the filling is planarized flat and then plated over, thus providing a flat pad for device attachment and effectively hiding the via hole from view and exposure to the surface of the board. Another reason to fill the vias is when the via holes are in a blind via subassembly. This avoids the requirement of back-filling with prepreg resin and helps maintain the resin between the layers during the lamination process. And of course the vias may be located within solderable surface pads as well, serving two different purposes! They may also be located in thermal vias placed under a device which act as a heat pipe to move heat away from the bottom side of the device and spread it out to the backside of the board or to inner thermal planes, etc.

So for all of these filled via applications there are various fill materials, but they typically fall into two categories - **conductive and non-conductive epoxy** materials. The epoxy material is intended to provide a similar coefficient of thermal expansion (CTE) in respect to the surrounding board laminate material, but it may also be filled with a silver/copper particulate material to make it slightly more electrically and thermally conductive.

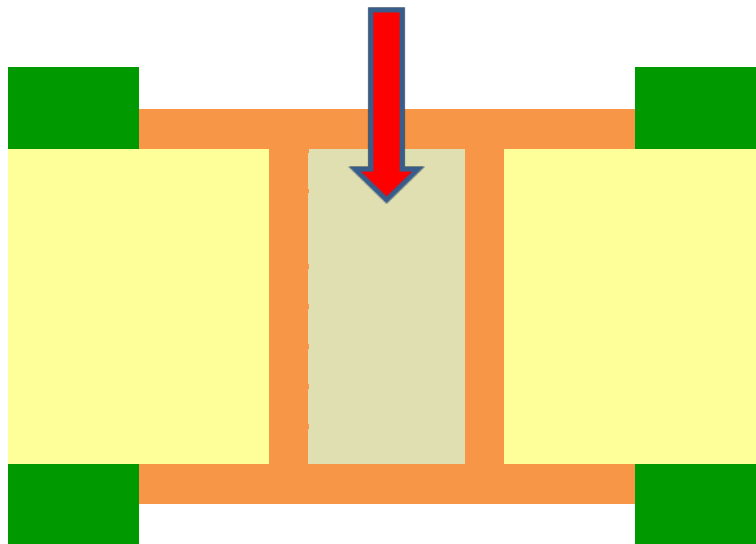


Figure 5: Conductive or Non-conductive Via in Pad

There is a lot of research data on whether or not the conductive material is beneficial or not in either application, but that will be explored in a future tech update. For this subject suffice it to say that either choice can completely fill a via hole and provide a flat, planar surface for device assembly.

We hope this explains some of the mystery and confusion with the choice of via treatment options, as there are distinct advantages, disadvantages and cost differences between all of the various processes. Please feel free to contact our engineering department for more details and additional helpful information!