

Q.P.I. Design strategy and recommendations with regard to the protection of through via holes in Printed Circuit Boards (PCBs).

With reference to the IPC suite of PCB standards (in particular design guideline IPC-4761).

Q.P.I. findings have led us to conclude that coverage of via holes (and pads) with solder mask and / or plugging from one or both sides can be detrimental to the reliability of the circuits (particularly with higher aspect ratio holes). This is due to the finishing chemistry becoming trapped in the via hole leading to either corrosion of the copper barrel or leakage onto the PCB surface, compromising solderability. Even with the pad and hole covered with solder mask, corrosion can still occur due to lifting of the solder mask (because of its reduced thickness) allowing finishing chemistry to become trapped underneath. Plugging from one side leads to a blind hole scenario, exacerbating the situation and also promoting solder ball formation; plugging from both sides can lead to air pockets which can explode during subsequent operations e.g. assembly. All potential failure mechanisms are described in IPC-4761.

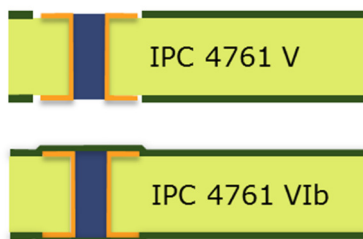
Q.P.I.'s recommendation is to either:

- **Clear the via hole of solder mask completely** with a 50µm+ clearance around the hole. This will ensure no solder mask covers or enters the holes and they acquire a full, reliable finish e.g. ENIG, Immersion Tin. This will, however, mean the pad is not fully covered with solder mask and hence is not fully insulated. It also means that the hole is not plugged / filled and therefore any vacuum operations may be hindered. Movement of chemistry, flux, solder etc. through the hole is also possible with this method.

OR

- **Resin fill the via hole**, copper plate over and fully cover with solder mask. This will also enable a reliable finish but will additionally cover the via hole pad fully with solder mask, offering the usual solder mask insulation. (Solder mask could be cleared from pads to enable electrical connection, if desired.) Holes will be filled, helping to enable any vacuum operations. This is Q.P.I.'s preferred option for ultimate reliability, however, it is also the more expensive option.

Recommended resin fill via hole options



Disclaimer:

All information is provided on an as is basis. Although Q.P.I. has done its utmost to have this information complete and correct, Q.P.I. cannot be held responsible for any incorrect and/or incomplete information. Q.P.I. can never be held responsible for any damages resulting from the use of this information. Q.P.I. has the right to alter this information at any time without notice.

Reference: IPC-4761: The IPC-4761 document describes various methods to protect vias and presents a potential failure mechanism in case of a blind hole scenario. One of the recommendations is that via protection should be part of procurement documentation.

In the case that solder mask is applied without the above mentioned QPI-recommendations, coverage of holes is not guaranteed and trapping of chemistry may occur. For a copy of the IPC-4761 contact: www.ipc.org