

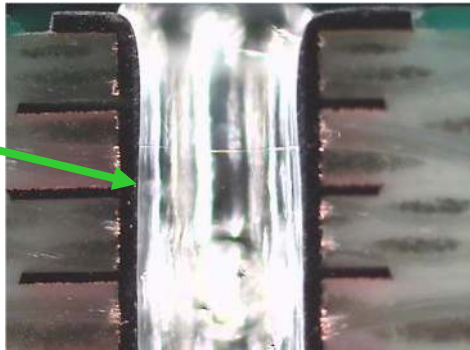
Guidelines/recommendations on “Solder resist design for vias”

(The recommended values are not legally binding and the layout specification is subject to a user-specific evaluation)

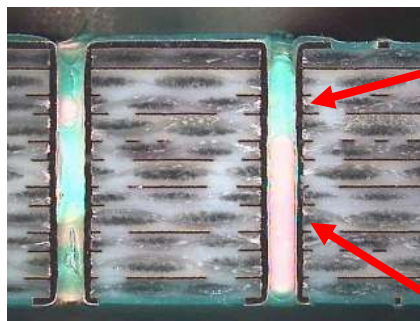
Objective 1:

- No undefined conditions, no exposed copper in vias caused by solder resist residues
- Fully metallised/ plated surface on pads and in via barrels
- Scope: photoimageable solder resist, all solder surfaces

Objective:

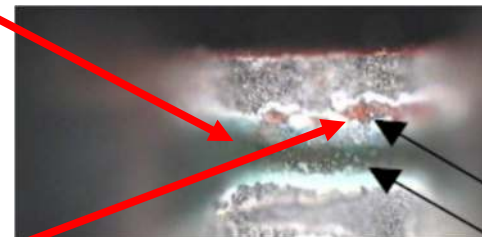


Soldering issues



Solder resist residue in hole

Exposed Cu in barrel



3 Photos courtesy of: ZVEI Quality Working Group

Guidelines/recommendations on “Solder resist design for vias”

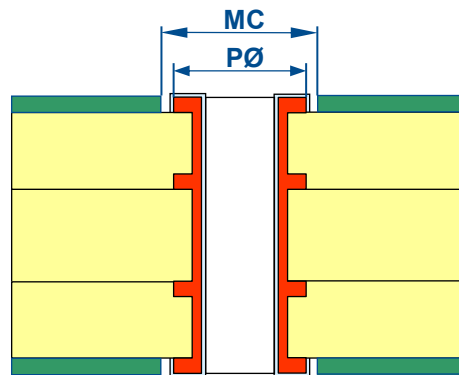
(The recommended values are not legally binding and the layout specification is subject to a user-specific evaluation)

Methods to reach the objective:

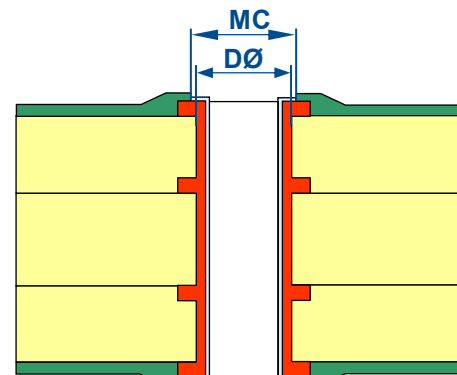
- Creating solder resist clearances around vias
- Reliable development process for solder resist application

Recommendations:

Mask clearance (MC) = Pad diameter (PØ) + 0.1 mm



Mask clearance (MC) = Drill diameter (DØ) + 0.15 mm



Preferred, process-reliable for drill diameter ≥ 0.3 mm, typical aspect ratio 1:5

2 Graphics: courtesy of ZVEI Quality Working Group

Guidelines/recommendations on “Solder resist design for vias”

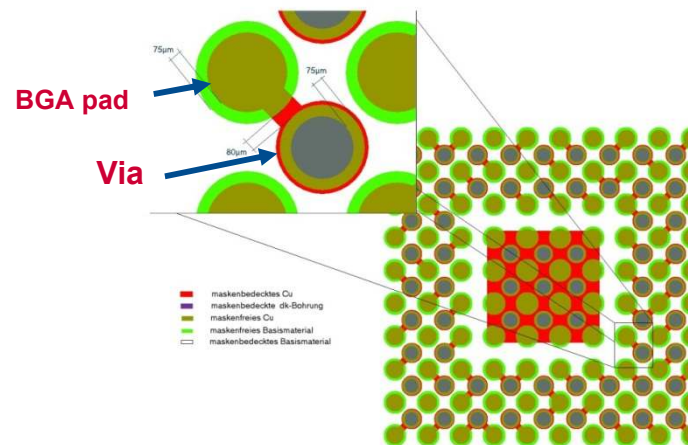
(The recommended values are not legally binding and the layout specification is subject to a user-specific evaluation)

Objective 2:

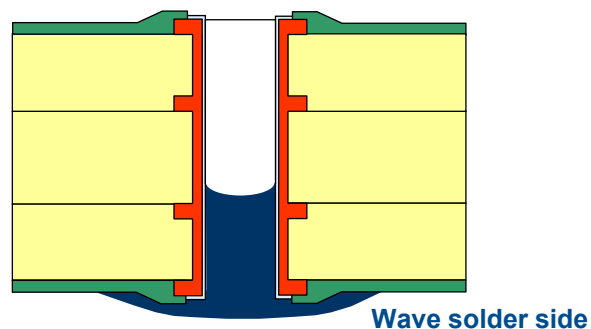
- Preventing solder migration from pad to via
- Preventing solder penetrating through vias to the other side
- Enabling vacuum suction for machining

Recommendations:

- Trimming of solder resist clearance to provide a solder resist bridge between pad and via



- Plugged via according to IPC-4761 type III-a on the PCB wave solder side enables the use of vacuum suction

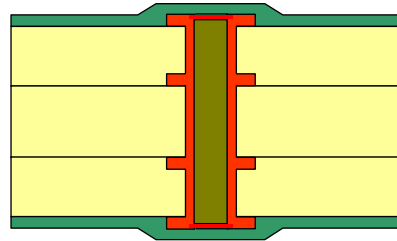


2 Photos courtesy of: ZVEI Quality Working Group

Guidelines/recommendations on “Solder resist design for vias”

(The recommended values are not legally binding and the layout specification is subject to a user-specific evaluation)

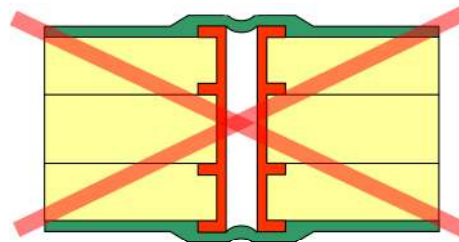
- Filled via according to IPC-4761, hole completely filled with plugging material (image shows filled & capped via according to IPC-4761 type VII-b)



Note: Filled via types according to IPC-4761:

Description	Type according to IPC-4761	Via filling options
Filled	Type V	Via with filling material
Filled & Covered	Type VI-a, VI-b	Via with filling material and pad covering
Filled & Capped	Type VII	Via with filling material and copper cap

Not recommended:
Tented via and/or mask clearance smaller than hole diameter + 0.15 mm



Risk of air or chemical entrapment, exposed copper, “blow-ups”:
⇒ No warranty assumed by PCB manufacturer

2 Photos courtesy of: ZVEI Quality Working Group