



BLIND VIAS IN PCBS

HOW TO USE
BLIND VIAS IN PCBS,
AVOIDING CRITICALITIES

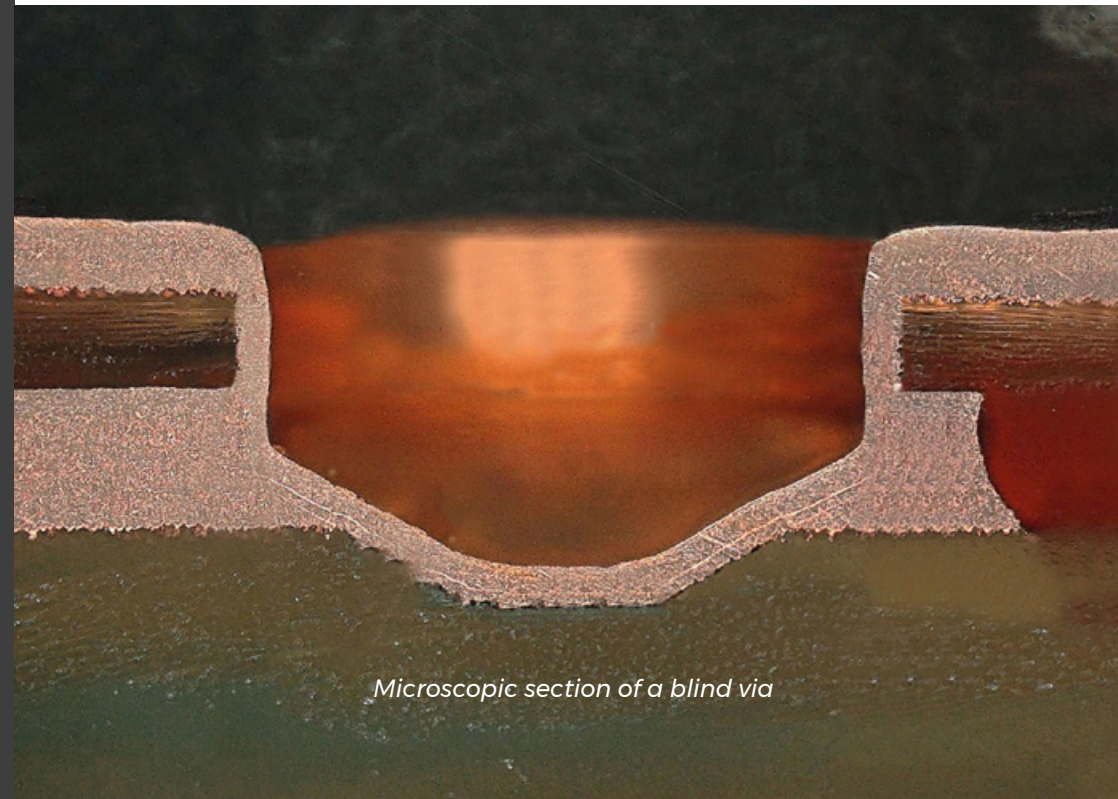
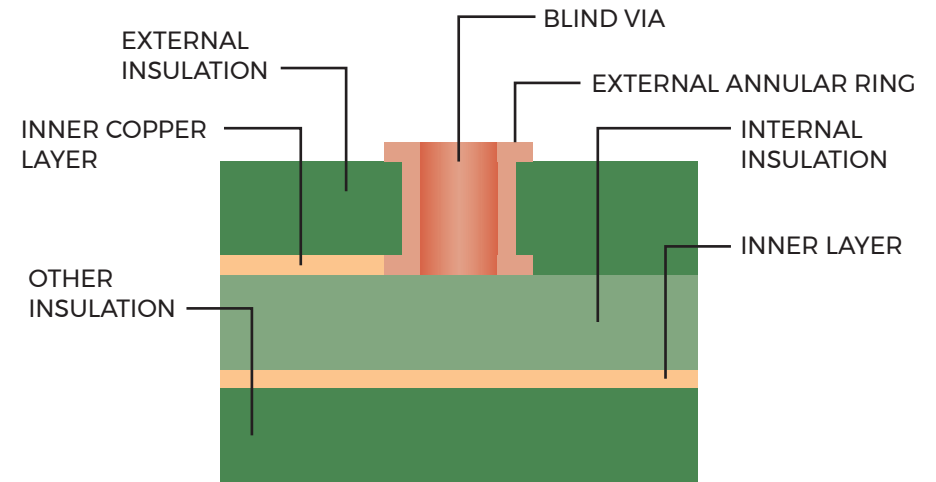


BLIND VIAS AND CONSTRUCTION PARAMETERS

Blind vias are plated-through holes that connect the external layer of a printed circuit board with internal ones without going through the entire PCB.

The hole is drilled on the Z-axis, and its diameter must ensure optimal metal coating on the walls.

SOME PARAMETERS HELP YOU choose a suitable diameter and thus ensure quality and reliability over time.



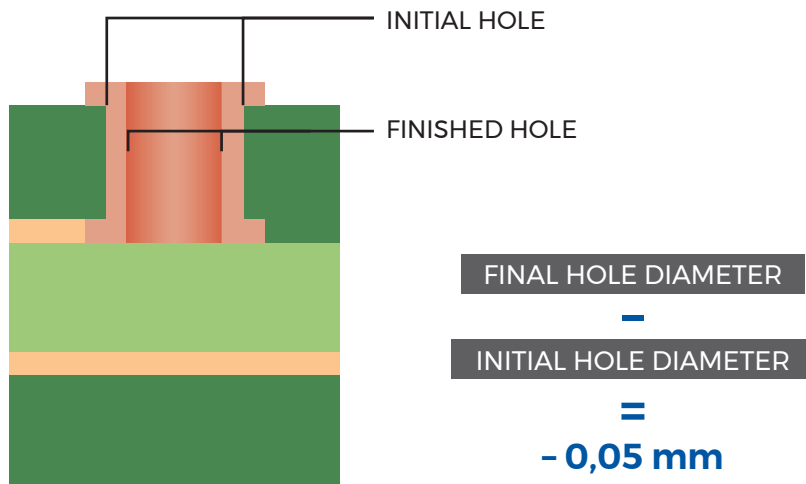
Microscopic section of a blind via

PARAMETERS TO CONSIDER WHEN USING BLIND V

DURING PCB DESIGN TO AVOID CRITICALITIES AND ENSURE RELIABILITY

HOLE DIAMETER

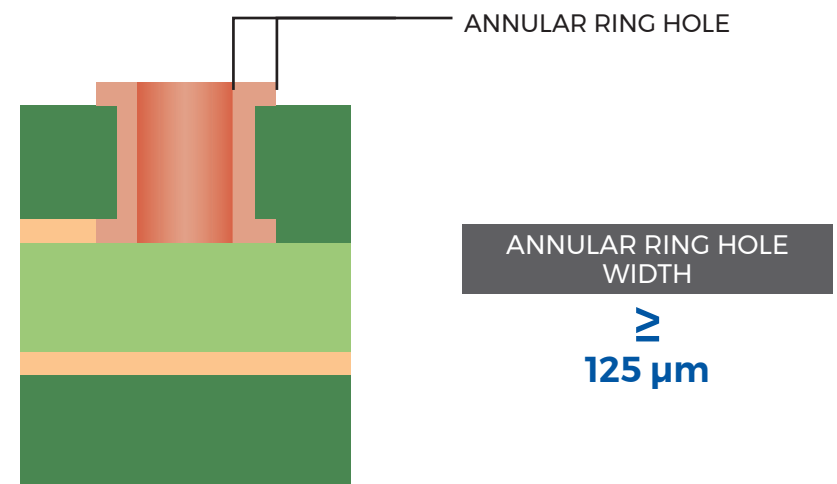
The smallest blind via obtainable is 0.15 mm, for mechanically-drilled holes or 0.1 mm for laser-drilled holes. The final size of the hole after plating process (minimum 12 μm on the hole wall, as per IPC standard) is 0.05 mm smaller than the one made. (Note: the mechanically-drilled hole admits a negative tolerance of 50 μm).



SIZE OF COPPER PAD CONNECTED TO THE BLIND VIA

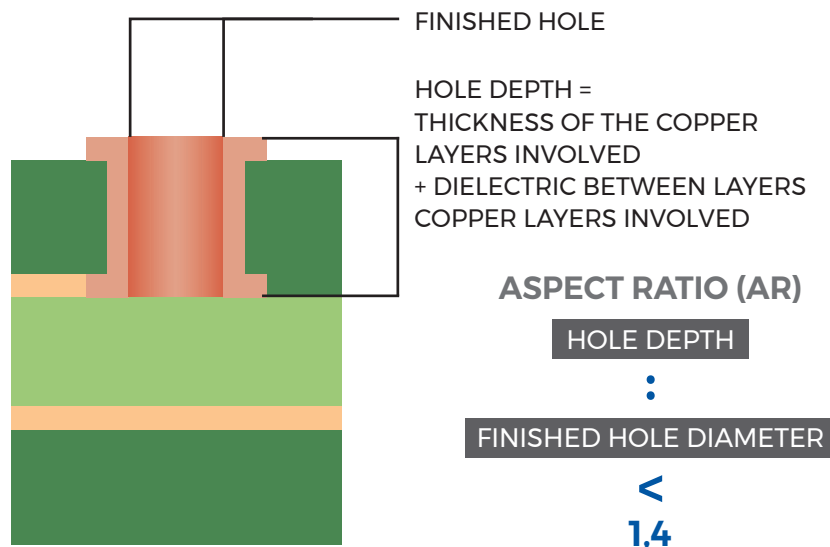
The blind via pad must have a diameter of at least 300 μm (11.8mls) larger than the blind via diameter (after CU plating - considering layers with a copper thickness of 35 μm) to comply with IPC standards and ensure an optimal PCB.

This aspect is essential for centring the hole on blind via pad while maintaining the required hole size. For laser-drilled holes, this value can be reduced to 250 μm (10 mls).



ASPECT RATIO (AR)

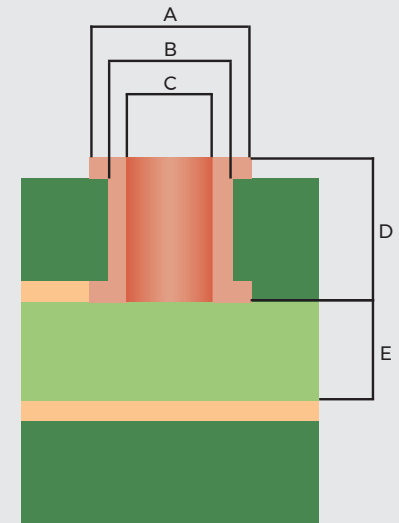
Aspect ratio refers to the ratio between the hole depth (given by the copper layer thickness and the dielectric thickness between the involved layers) and diameter after the metallisation and galvanic coating processes. This ratio must be a maximum of 1.4 to ensure a strong metal coating on the hole wall and a reliable electrical connection.



EXAMPLE

Example of a reliable blind via:

suppose you have a PCB with 35 μm -thick copper layers, where 100 μm of dielectric material insulate the top and the first inner layer, and 150 μm (E) of material insulate the second inner layer. In this case, you can engineer a blind via with a 150 μm diameter (C) (no laser) centred on copper pad having a diameter of 450 μm (A). This hole is made with a 250 μm -diameter tool (B), which crosses a material thickness (FR4 + copper layer) equal to 170 μm (D). This way, AR reaches 1.13:1 (D/C), which is lower than 1.4:1. This ensures excellent performance.



- (A)** 450 μm
- (B)** 250 μm
- (C)** 150 μm
- (D)** 170 μm
- (E)** 150 μm
- (D/C)** AR 1,13:1



PCB COMPETENCE

MANUFACTURING AND SERVICE

ALBA PCB Group

via Leonardo Da Vinci, 3 | Mogliano Veneto (TV) | 31021

Tel: +39 041 5902638 | Fax: +39 041 5904408

info@alba-pcb.com | www.alba-pcb.com