

Holes on PCB

Duo Tao

California Institute of Technology

February 17, 2019

Background

This document clarifies the meanings of some terminologies used about holes on PCB.

Vias and pads

We place components on pads, and not on vias. Vias are just for passing signals or powers across layers. However, physically they are the same thing.

Plated through holes

These holes have copper on the walls of the hole, so that you can just solder on the back side. Most holes nowadays are plated [1]. However even though there is copper on the walls, it does not mean it is automatically connected to every internal layers. If it is not connected, area around the hole on that layer is cleared and is thus not connected.

By the way, shift-s on Altium is a very useful shortcut you can use to see only one layer.

Mounting hole

Actually this is what I am trying to do today. Placing a mounting hole on Altium, you just need to place a via there with the right hole size, and perhaps ground it.

References



PCB Unlimited. *Plated through holes*. Feb. 16, 2019. URL:
<http://www.pcbunlimited.com/engineering/plated-non-plated-through-holes.php>.