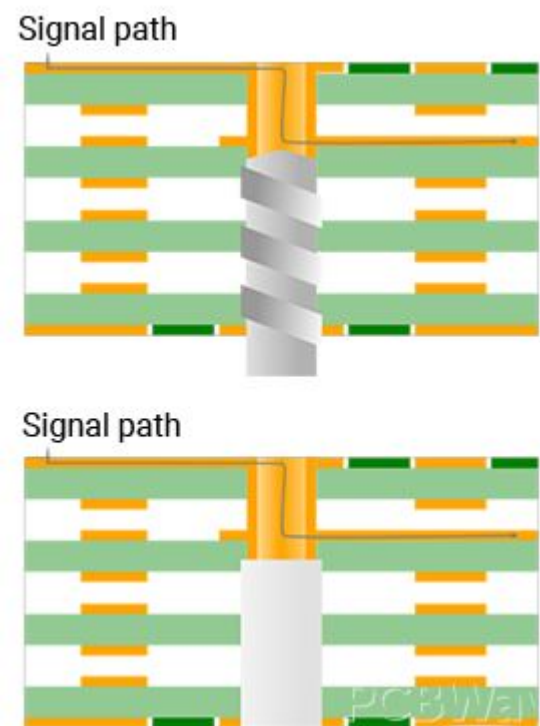


Backdrill - PCB Prototype the Easy Way

Backdrill refers to the removal of stubs present in copper-plated vias. A stub is the unused portion of a via that does not connect to any layer or serve any function. To better understand back drilling, let's consider a simple example: suppose we have a ten-layer board and a high-speed signal only needs to be transmitted from the first layer to the seventh layer. However, we typically place a via that passes through all the layers of the board, including the eighth to the tenth layer. The eighth and tenth layers are not connected by any trace and form a pillar, which can cause signal integrity issues and needs to be drilled out from the backside (secondary drilling), hence the term back drilling.



Unused vias or stubs in multilayer PCBs can cause impedance mismatches in the signal path.

For high-frequency PCBs (such as those requiring impedance control), techniques such as back drilling, blind vias, and buried vias can be helpful so that they can form part of the solution to address such issues.

Back drilling is a technique used to remove via stubs from the inner layers of a multilayer PCB. If you are unfamiliar with back drilling, this video may be of assistance. In this video, we introduce “What is PCB Back Drilling?”, “How Back Drilling Works?” and “Factors that Influence the Effectiveness of Back Drilling”. Watch till the end of the video to find the ordering entrance, if you want to select this technique for your design from PCBWay.

General Characteristics

- Typically used for eight or more layer boards
- Board thickness is greater than 2.5mm
- Mostly are rigid boards on the back
- The minimum hole size is 0.3mm
- The back drilling diameter is 0.2mm larger than plated through-hole diameter

- The tolerance of back drilling depth is +/-0.05mm.

The Advantages of Backdrill

- Improved signal integrity
- Better impedance control
- Reduced signal crosstalk between vias
- Reduction in local board thickness
- Reduced reliance on blind and buried vias, which can simplify PCB fabrication.

The Manufacturing Process of Back Drilling

1. Use the positioning holes on the PCB for positioning and perform the first drilling.
2. Seal the positioning holes with dry film, then electroplate the PCB after the first drilling.
3. Create the outer layer pattern on the PCB after electroplating.
4. Seal the positioning holes with dry film, and then perform pattern plating on the PCB with the formed outer layer pattern.
5. Use the positioning hole used for the first drilling to locate the back drilling, and use a drill to perform back drilling on the plated holes that need to be back drilled.
6. After back drilling, wash the back drilled holes to remove any remaining debris.

The Function of Back Drilling

The purpose of back drilling is to drill off the unused portions of plated through holes that do not serve any connection or transmission functions, in order to prevent issues such as reflection, scattering, and delay in high-speed signal transmission.

The Application Fields of Back-drilled Boards

The application fields of back-drilled boards mainly include communication equipment, large-scale servers, medical electronics, military, aerospace, and other related industries.

PCBWay provides services including PCB Prototype & Batch Production, PCB Assembly (SMT), Electronics Modules Selling, CNC Machining, etc. We are committed to meeting the needs of global makers from different industries in terms of quality, delivery, cost-effectiveness and any other demanding requests in electronics.

More information please check here:

[PCB Instant Quote](#)