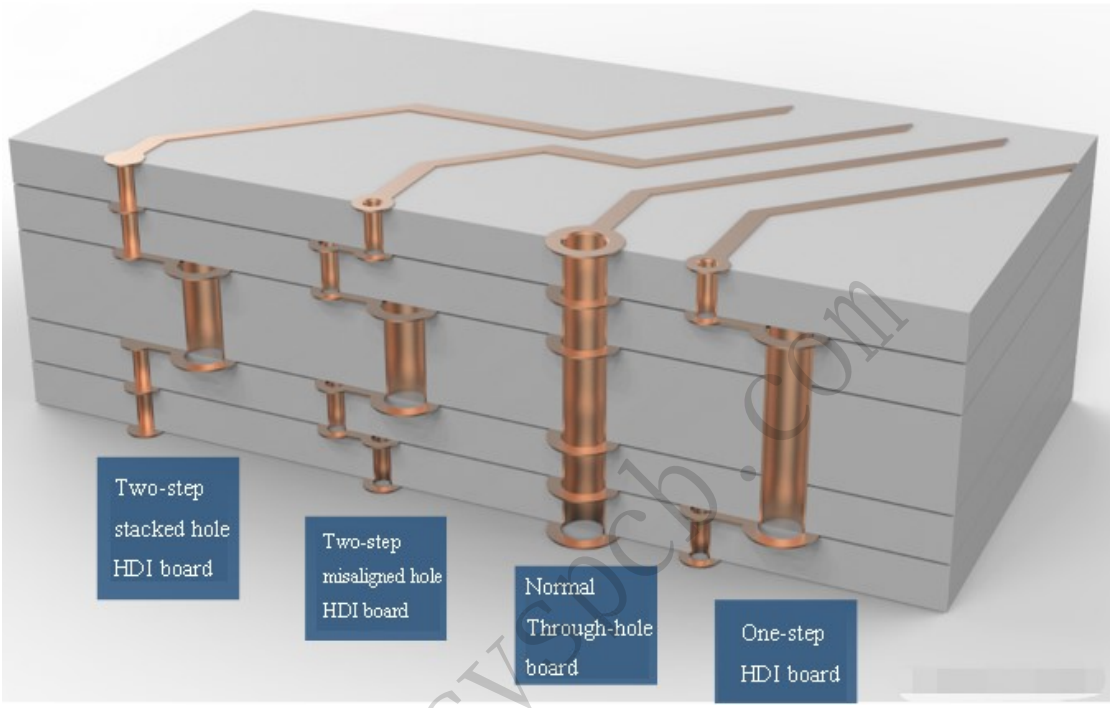
What does the inside of a multilayer PCB look like? The 3D large picture analyzes the design process of high-end PCB boards

When freshman hardware engineers look into multilayer PCBs, it is easy to get dizzy. There are ten and eight layers inside PCB, and the lines are like spider webs.

Today, we drew several internal structure images of multilayer PCB, and use tridimensional graphics to show the internal structure of PCB with various laminated structures.



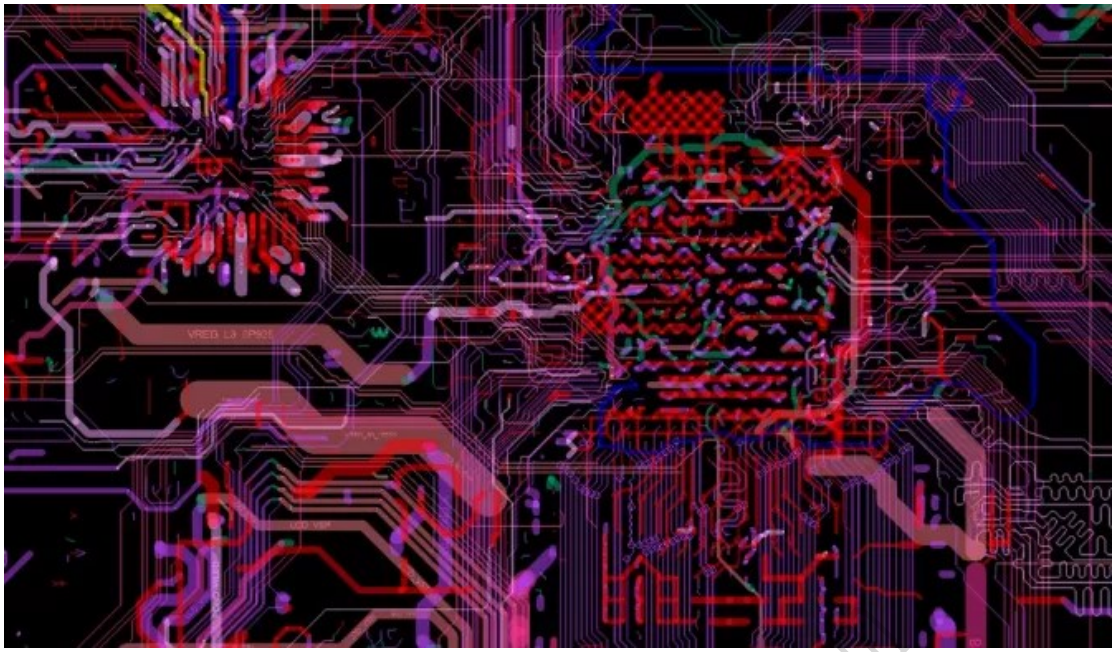
1. The core of the high-density interconnection board (HDI) is in the via

The circuit manufacturing process of multi-layer PCB is no big different from single-layer and double-layer, and the biggest difference lies in the process of via.

The lines are all etched, and vias are all drilled and then plated with copper. Everyone who does hardware development understands these, so we won't give unnecessary details.

Multilayer circuit boards usually include through-hole boards, 1 step HDI boards, 2 step HDI boards, and 2 step stacked-hole boards. Higher-end boards, such as 3 step boards and any-layer interconnect boards, are usually used very little because of expensive price, so I won't discuss them too much first.

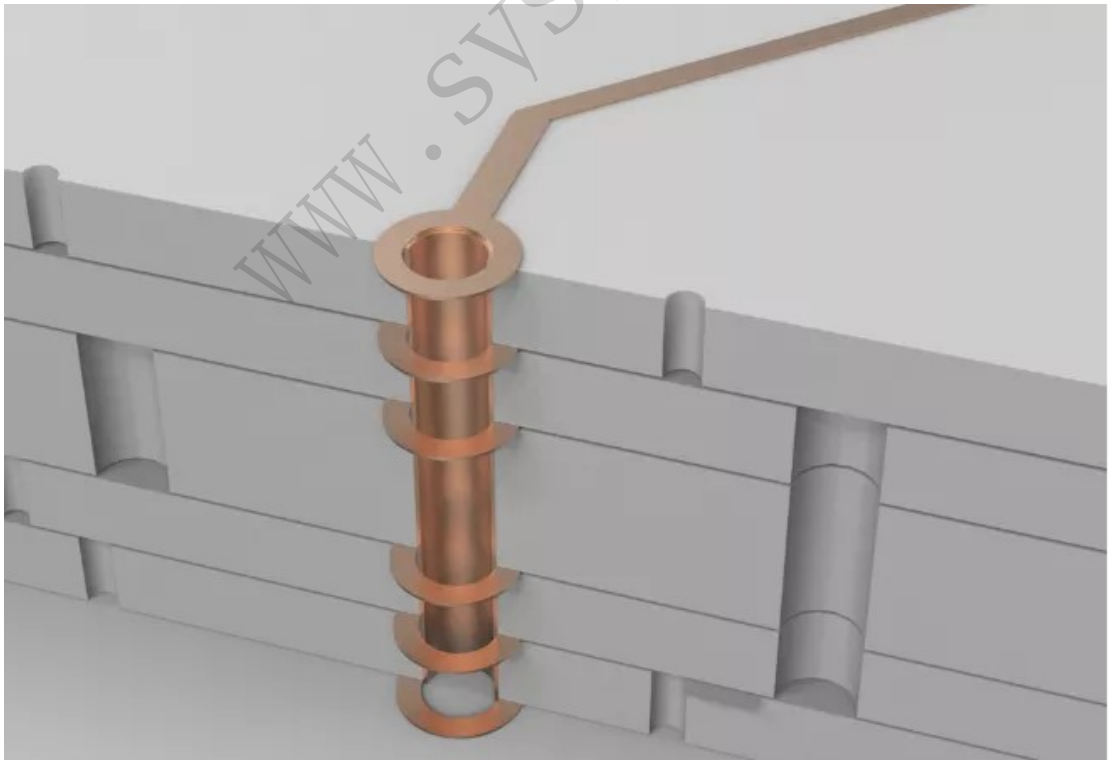
In general, 8-bit MCU products use 2 layer through-hole boards; 32-bit MCU smart hardware uses 4 layer to 6 layer through-hole boards; Linux and Android level intelligent hardware uses 6 layer to 8 layer through hole PCB or 1 step HDI board; smart phones generally use 8 layer 1 step to 10 layer 2 step HDI circuit boards.



8-layer 2-step stacked hole, Qualcomm Snapdragon 624 processor PCB

2. The most common through holes

There is only one type of via, from the first layer to the last layer. Regardless of whether it is an external circuit layer or an internal circuit layer, the holes are drilled through, which is called through hole boards.



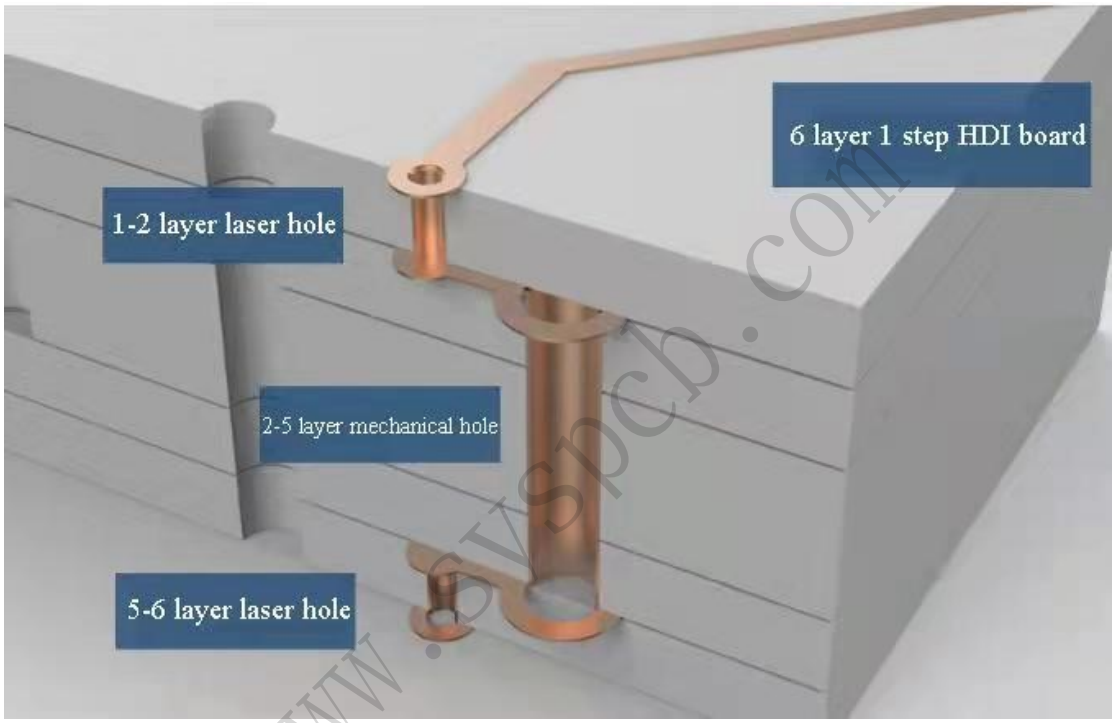
Through hole boards have no relationship with the number of layers, and two-layer boards usually used are all through-hole boards, and many Ethernet switches and military circuit boards are

20-layer through-hole boards.

Use a drill bit to drill through the circuit board, and then plate the hole with copper to form a interconnection.

It should be noted here that the diameter of the through hole is usually 0.2mm, 0.25mm and 0.3mm, but generally 0.2mm is more expensive than 0.3mm. Because the drill bit is too small, easily break, and the drill speed is also slower. The extra spent time and cost of the drill bit are reflected in the increase in the price of the circuit board.

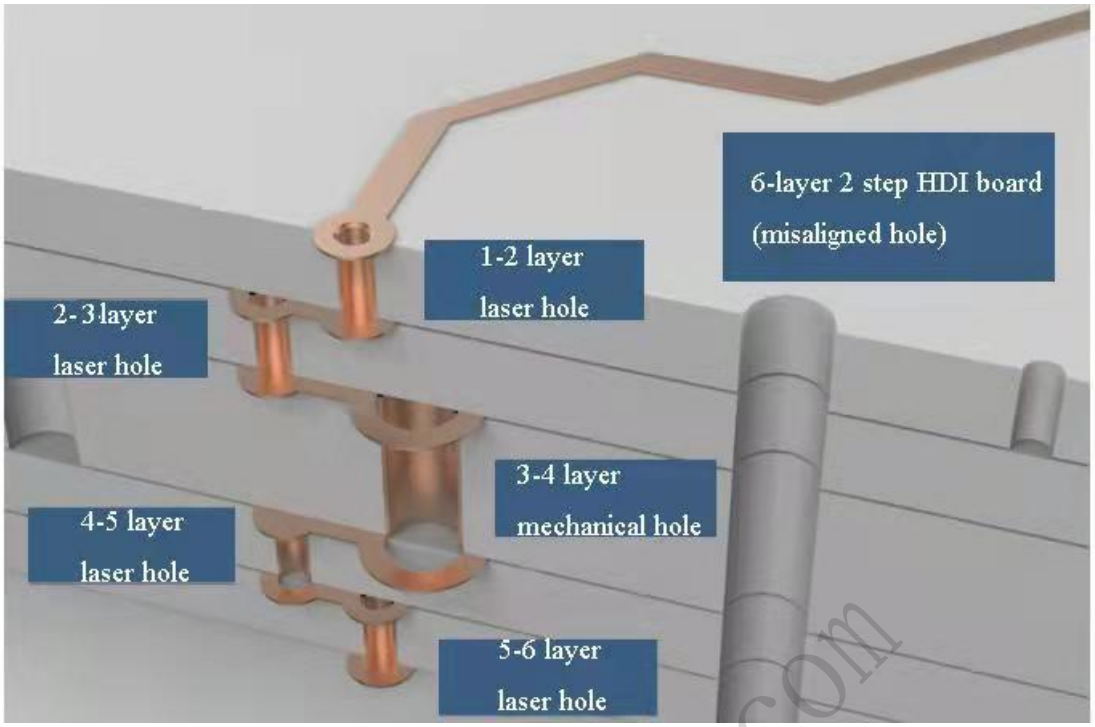
3. Laser hole of high-density board (HDI board)



This picture is a laminated structure picture of a 6-layer 1 step HDI board. Holes on Both surface layers are laser holes with an diameter of 0.1mm. The inner layer is a mechanical hole, which is equivalent to a 4-layer through-hole board, whose outer layer is covered with 2 layers.

The laser drill can only penetrate glass fiber sheet material, can not penetrate metal copper. Therefore, the outer surface drilling will not affect other internal circuits.
After the laser drills the hole, go to copper plating, and the laser via is formed.

4. 2 step HDI board with two layers of laser holes



This picture is a six-layer, 2 step HDI board with misaligned laser holes. Usually, people seldom use six-layer 2 step HDI PCB, and most of them start with eight-layer 2 step HDI PCB.

The so-called 2 step means that there are two layers of laser holes.

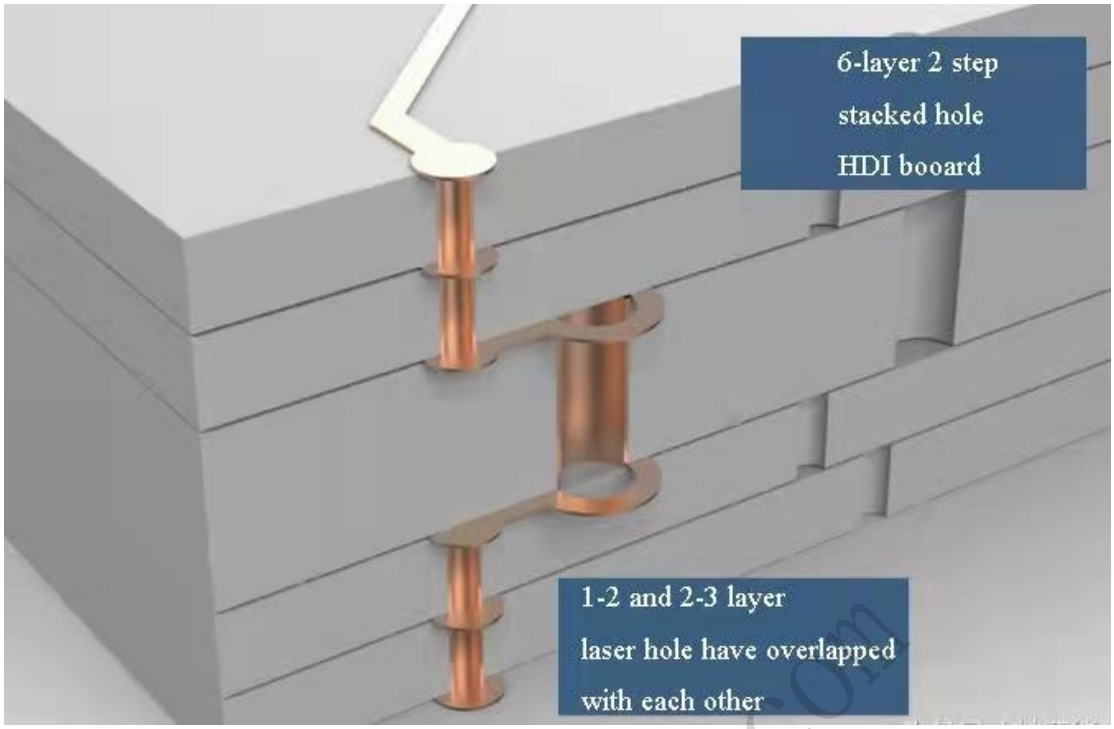
The so-called misaligned hole means that the two layers of laser holes are staggered.

Why should it be staggered? Because the copper plating is not full, the inside of the hole is empty, so you can't drill holes directly on it, and you have to stagger a certain distance, then make a layer of hole.

6 layers of 2 step HDI= 4 layers of 1 step plus 2 outside layers.

8 layers of 2 step HDI = 6 layers of 1 step plus 2 outside layers.

5.Stack up hole board is more complicated and more expensive



The two layers of laser holes of the stack up hole board overlap each other. The conduct line and components will be more compact.

The inner laser hole needs to be electroplated and filled, and then the outer laser hole is made. The price is more expensive than the stagger hole HDI PCB.

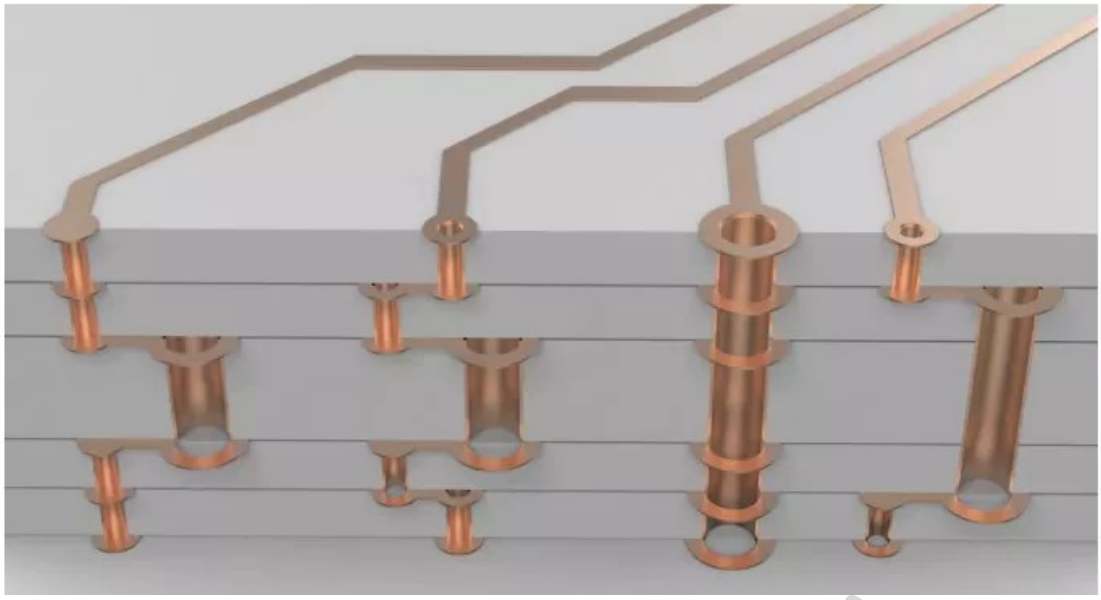
6. Super expensive any-layer interconnect boards with multi-layer laser stack up hole
That is, each layer connected using laser holes, and every layer can be connected. You can route the wire as you want, and add holes as you want.

PCB Layout engineers feel cool to think about it! Don’t be afraid to draw anymore!

The purchasers want to cry when purchasing, because it is more than 10 times expensive than ordinary through-hole boards!

Therefore, only products like the iPhone are willing to use it. For other mobile phone brands, I have never heard of any brands that have used any-layer interconnect boards.

Summarize



Finally, add a picture and compare carefully.

Please pay attention to observing the size of the hole and whether the solder pad of the hole is covered or open.