



ROYAL CIRCUIT SOLUTIONS

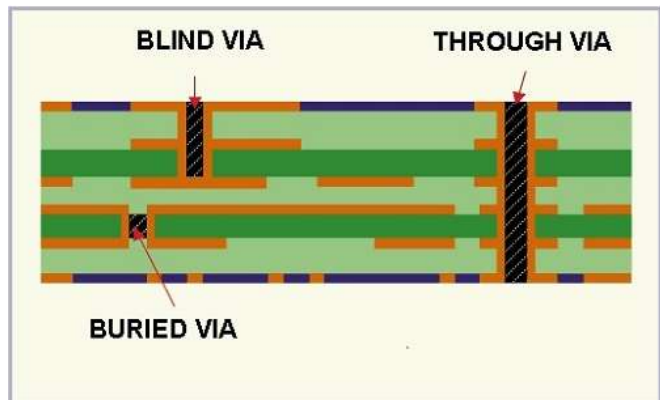
Designing with Vias

High Density Interconnect (HDI) circuit board designs have a higher wiring and pad density than conventional PCBs, along with smaller trace widths and spaces. For this reason, they require advanced PCB technologies such as blind vias, buried vias, and via-in-pads. To manufacture these types of vias in a multilayer PCB requires either a process of sequential lamination of prepared components, or precise drilling of a completed multilayer PCB. Both methods demand a trusted PCB fabricator with experience in advanced manufacturing technologies. Utilizing vias a multilayer PCB can significantly increase the number of connections without increasing the size.

About Vias

Vias make electrical connections between layers on a printed circuit board. They can carry signals or power between layers. The following lists the types of via configurations uses on multi-layer PCB designs:

- **Through Via:** Through vias are the oldest and simplest via configurations originally used in 2-4 layer PCB designs. The signals originate and terminate from the outer layers of the PCB.
- **Blind Via:** A blind via is a copper plated hole, just like a regular via, except that it interconnects only one external layer of the PCB with one or more internal layers, but does not go all the way through the board.
- **Buried Via:** A buried via is a copper plated hole that interconnects one or more internal layers, but does not connect to an external layer, hence the hole is completely internal or buried within the board.
- **Micro Via:** A micro-via is a form of blind via. Because the holes are so small (0.006 inches or less), they are formed using lasers, and cannot penetrate more than one or two layers at a time. They are most commonly used in high-density PCB designs like cell phones, or in FPGA and custom ASIC chip packaging.
- **Via-in-Pad:** Via-in-pads allows for the via to be placed beneath the component contact pad. This allows for more routing in less space and a smaller PCBs





ROYAL CIRCUIT SOLUTIONS

This technology offers a viable design technique to help meet the density constraints of lines and pads on a typical design without increasing the layer count or board size.

Designing with Buried Vias

Buried vias are easy to incorporate into your design, because they do not impact any trace or surface mount component on the top or bottom layers of the board. This is because they are completely buried within the board. As long as the designer maintains the typical clearances from other pads or traces within the inner layers, they are really no different than conventional vias. Since they are completely buried, the designer can now place a trace or an SMD pad on the outer layers directly over the buried via, thereby gaining some added space on the outside layers.

Designing with Blind Vias

The placement of blind vias is not much different. Again, standard clearance rules apply. It is often tempting to place blind vias under SMD components to save space, but this practice is not recommended. Solder flux can become trapped in the blind via hole and cause corrosion problems over time.

When using blind vias, one aspect of the design that must be carefully monitored is drill depth. In a typical 8 layer board, 62 mil thick, using 35 mil vias with 18 mil hole, the maximum depth of the via hole must not exceed 18 mils.

Designing with Via-in-Pad

You can also place your blind vias within pads (via-in-pad). Via-in-pads allows for the via to be placed beneath the component contact pad. This allows for more routing in less space and a smaller PCBs footprint. Via in pad helps to reduce inductance, increase density and employ finer pitch array packages.

When using via-in-pads, consider having the board fabricator plug the via and then plate copper over it. This can give you the benefits of via in pad without causing problems in assembly. Please be aware, however, that designing with via-in-pads can be more expensive due to the need for additional manufacturing processes and material cost for the conductive fill.

Recommended Hole to Pad Ratios

For internal layers, a via pad that is 0.016" over the drill size is a good choice, as it provides a good yield in manufacturing. For external layers, a via pad 0.012" over the drill size is acceptable, although the same via size for both internal and external layers is preferred.



ROYAL CIRCUIT SOLUTIONS

Every design has different requirements and constraints. However, when possible, utilize these tips to select the appropriate via technology and correct ratio will help reduce the overall lifetime costs and increase the reliability of your end-product.