

Basics For Beginners

Just For Starters

What is a PCB?

Circuit boards are familiar items to electronics enthusiasts and lay people alike. Clear cellphone bezels reveal small circuit boards jammed with tiny components. When you open a PC, the motherboard is visible — populated with a variety of integrated circuits and other components.

With the exception of certain prototype circuitry, the great majority of electronic circuits are constructed using printed circuit boards, or PCBs. If you plan on working with a PCB, it helps to understand some basic terminology, as well as what a PCB actually is. PCB design and construction is a vast area of concentration that fills volumes of text. This article provides a brief overview of PCB basics.

Fiberglass and Copper

A PCB is constructed from multi-

ple layers of thin fiberglass. Some layers are coated with copper foil and others are bare fiberglass. The thickness of each layer ranges from several mils (a mil is 1/1000 of an inch) to tens of mils. Layer thickness is determined by the desired electrical properties and dimensions of the finished PCB. The PCB is built up by stacking layers of fiberglass and copper until the specifications are met. Figure 1 shows a cross-section of a multi-layer PCB. Note the thin copper foil layers sandwiched between thicker fiberglass layers. Electrical connections are formed by etching patterns into the copper layers. This is where the term "printed" comes from. A photographic mask is created and each layer of copper foil is treated with a photo-resistive chemical. Much like a normal photograph, wiring patterns are "printed" onto the copper by exposing it to light through the mask. The exposed copper is chemically etched and individual wires remain behind.

Copper foil is manufactured in three common thicknesses, or weights per square foot. A PCB fabricator, or "fab-house," will specify the copper weights that it stocks. The most com-

mon weight is half ounce, but one and two ounce copper is common, as well. Half-ounce copper is approximately 0.006" thick. Thicker copper can carry more current, but poses etching problems as the patterns get smaller.

Vias

Wires from multiple layers are connected after the stacking process. A small hole — called a via — is precisely drilled through the fiberglass stack and the surface of the hole is metal plated. The plating makes electrical connections to copper that is penetrated in each layer. The vertical metal surfaces in Figure 1 illustrate this concept. Each via appears as a small hole from the surfaces of a PCB. Figure 2 shows a finished PCB with vias clearly visible. Some vias have etched copper connections visible on the top layer.

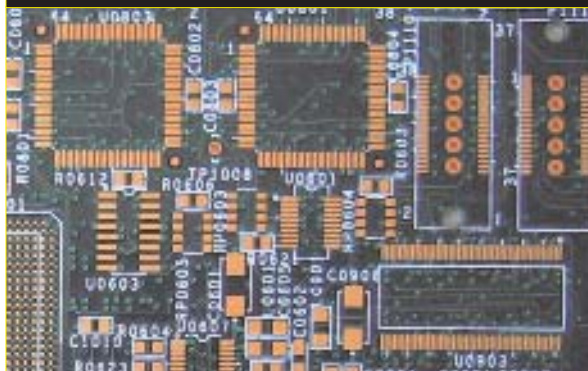
Printed wires on each copper layer are very narrow. Mature construction technologies allow copper "traces" as thin as 8 mils. More advanced manufacturing techniques routinely produce traces down to 3 mils. Below 3 mils is possible, but expensive.

Each fab-house has its own manufacturing capability. Some offer finer pitch PCBs than others. A fab-house should tell you how small they can reliably manufacture traces and the spaces between them. For example, a quote of "8-mil trace and 7-mil space" means that the fab-house can etch copper traces

Figure 1. Cross-section of a multi-layer PCB with a plated hole.



Figure 2. Surface view of vias with wires on the top layer.



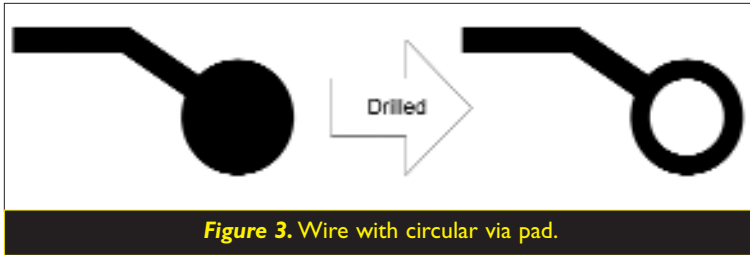


Figure 3. Wire with circular via pad.

down to 8 mils wide with 7 mils between each trace. With such small copper features, properly aligning each layer and drilling vias is not trivial. Vias are not simply drilled to hit wires because a wire's cross-section does not contain enough copper to make a reliable electrical connection with the via plating. Instead, Figure 3 illustrates how via connections are designed with circular pads at the end of each wire. When drilled, the pad forms a ring that surrounds the via. The via's plating has much more area to form a connection.

Blind and Buried Vias

Blind and buried vias are special types of vias used in leading-edge systems. Conventional vias are drilled through the entire PCB after all layers have been stacked. Blind and buried vias are drilled only through a few layers during the stacking process. As

seen in Figure 4, the result is that vias do not take up the entire layer stack and two or more such vias can be located in the same place on different layers. High density PCBs may require blind and buried vias. This technology is more expensive than conventional vias because additional manufacturing steps are required and layer stacking becomes a more difficult process.

Planes

Not all PCB layers are used for conventional wiring. Most PCBs reserve some layers for use as power planes. As their name implies, planes are nearly continuous sheets of copper. A plane is used to carry power to many or all components on the PCB with low inductance and resistance.

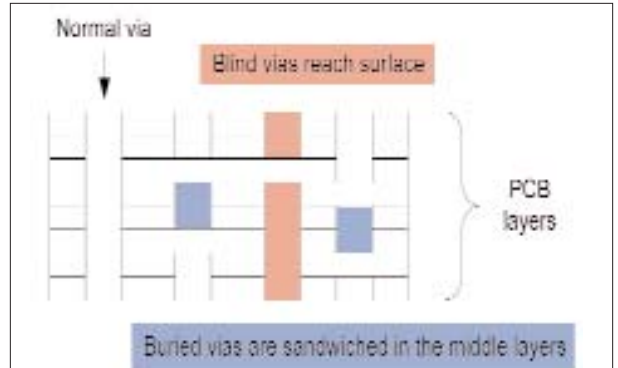


Figure 4. Blind and buried vias connect only a few layers of the PCB.

These electrical properties become critical in fast and sensitive circuitry.

Planes are continuous except for via clearance holes. Clearance holes are necessary to prevent vias shorting their signals to the power planes. This would normally happen because vias are drilled through all PCB layers (or some layers with blind and buried vias). The solution is to etch away a circle of copper in each plane where each via hole is drilled. There can be no electrical connection between the via and the plane if there is no copper between the via hole and plane. Most PCBs have several planes and assign each plane to carry a specific power rail. For example, a microprocessor board may have 5-volt and 3.3-volt power supplies plus ground return. The PCB may have four planes: 5, 3.3, and two grounds. In some cases, a product cannot accept the added cost of one plane for each power rail. It may be possible to split a plane into multiple areas for multiple power rails. Whether or not this split is possible depends upon the circuit's electrical constraints. A split plane is constructed by etching a copper void into an otherwise continuous plane.

Moats

Sometimes a plane is partially split for electrical reasons. Noisy circuits may cause problems in a sensitive system. Some engineers attempt to minimize noise transmission by etching a void into part of a plane, but leaving a small electrical connection for power and signal routing purposes. This type

of void is called a moat. Figure 5 shows several moats on a single PCB as the board is held up to a light. The fiberglass is translucent, while the copper planes are opaque. Moats can cause as much trouble as they solve, so it is best to fully understand their dynamics before proceeding.

Land Patterns

We've discussed wiring up till now, but PCBs exist to connect components. Most components are soldered to PCBs. In order for soldering to be effective, the leads or pins on a component must be capable of reliable mechanical and electrical connections to the etched copper wiring.

Each component is provided with a matching land pattern — a set of etched copper features that directly correspond to the leads or pins of the

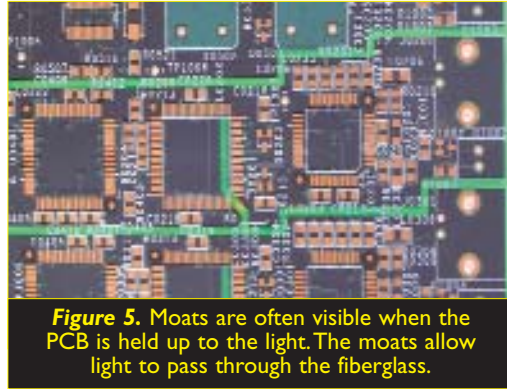


Figure 5. Moats are often visible when the PCB is held up to the light. The moats allow light to pass through the fiberglass.

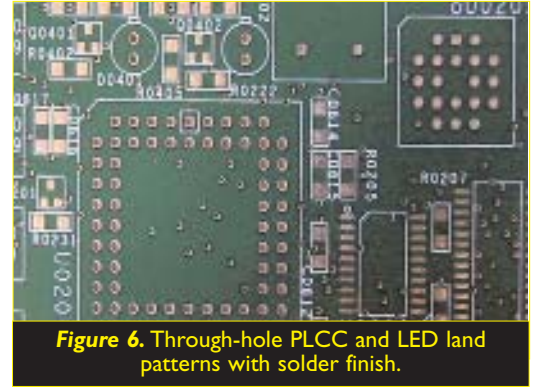


Figure 6. Through-hole PLCC and LED land patterns with solder finish.

component. The features are usually made larger than the leads or pins so that a sufficient quantity of solder can adhere. Land patterns may be combinations of plated holes and surface pads, depending on the type of component being mounted. Figures 2, 5, 6, and 7 show a variety of land patterns. There are patterns for through-hole connectors, quad-flat packs (QFP), small-outline integrated circuits (SOIC), discrete surface-mount resis-

tors, and more. Land patterns sometimes include special visual markers called fiducials. The figures also show several examples of fiducials. Fiducials are typically round patterns located in the corner of surface-mount land patterns that robotic assembly machines key off of to accurately place components onto the PCB. These machines are often called "pick-and-place" machines. The fab-house finishes exposed copper land patterns different-

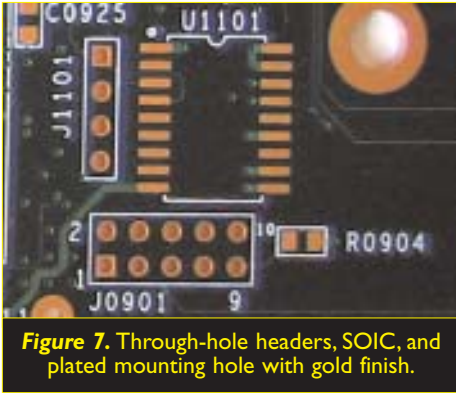


Figure 7. Through-hole headers, SOIC, and plated mounting hole with gold finish.

ly according to your assembly requirements. The basic finish is a solder wash (shown in Figure 6) that takes on a silvery solder appearance. Another finish is gold, which is shown in the other figures. There are other finishes, as well.

Solder-mask and Silkscreen

Did you ever wonder why PCBs tend to be green? Green is simply the

default color for solder-mask that most fab-houses use. Solder-mask is applied over the entire surface of a PCB, except for land patterns and other copper areas that must make electrical connections. The solder-mask eases assembly by preventing solder from adhering. Solder-mask is available in a rainbow of colors including blue, red, and clear. Silkscreen patterns and lettering are applied to most PCBs to ease human inspection and debugging. Assembly machines usually don't care about the silkscreen. However, if you need to find resistor 125 out of 600, then silkscreen is your friend! Silkscreen seems to most often be white, but it, too, is available in a variety of colors.

There's a lot more to say about PCBs. Understanding their electrical characteristics is very important. The dimensional properties already mentioned have significant electrical consequences. This is where transmis-

About the Author

Mark Balch is the author of *Complete Digital Design* (see www.completedigitaldesign.com) He is an electrical engineer in Silicon Valley, CA, who designs high-performance computer-networking hardware. His responsibilities have included PCB, FPGA, and ASIC design. Prior to working in telecommunications, Mark designed products in the fields of HDTV, consumer electronics, and industrial computers. In addition to his work in product design, Mark has actively participated in industry standards committees and has presented work at technical conferences. Mark holds a bachelor's degree in electrical engineering from The Cooper Union in New York City. He can be reached via email at mark_balch@hotmail.com.

sion line theory takes over. There are many books, magazines, and websites out there that cover these topics, so happy reading! **NV**