PCB design basics

Mahmoud Wahby - November 13, 2013

This is the first of several articles in the PCB Design Best Practices series, which discusses the different steps of PCB development from the basics of creating a design schematic with specific requirements, to finalizing a board and preparing it for fabrication. The articles will use examples from National Instruments circuit design tools NI Multisim and NI Ultiboard.

This article discusses the major steps in the PCB design flow, from basic terminology to the primary steps required to move an example design through the schematic, layout, and manufacturing stages.

Understanding the Terminology

Schematic capture and simulation tools – A schematic capture program allows the user to draw a document representing the electrical component symbols and the interconnections between them in a graphical way. Before generating a PCB, the symbols are mapped to component footprints and the symbol interconnections are converted to a netlist that specifies the connections between the component footprints in the layout process. A schematic tool that allows the user to do interactive circuit simulation with the same schematic circuit representation used for layout is advantageous. Circuit simulation can be useful for both initial design analysis and testing the design (i.e. verification testing and troubleshooting) once complete.

PCB layout tools – A PCB layout program generates the mechanical and wiring connection structure of the PCB from the netlist. The layout program allows the wiring connection structure to be placed on multiple layers and, once complete, allows the user to generate the computer aided design (CAD) files needed to manufacture a PCB.

Gerber files – The CAD files that need to be sent to a PCB manufacturer so it can build the PCB layer structure are called Gerber files. The RS-274X is the most commonly supported Gerber file format.

NC drill files – The numerically controlled (NC) drill files indicate the size and position of holes used for unplated holes, plated through-holes, or holes for vias. Some quick-turn PCB manufacturers have only select hole sizes available.

Printed circuit board (PCB) – A wafer board defining the mechanical and copper wire structure of the circuit. (It is sometimes called a PWB for printed wiring board).

PCB structure and details

A PCB can be considered a layered structure, usually with multiple copper and insulating layers. The main portion is a non-conductive (insulative) material (substrate) usually made from fiber glass, and epoxy. The substrate material used to separate layers comes in different thicknesses, from 0.005” to 0.038”. Conducting layers consist of copper (Cu) foils that are etched away in specific areas where the user does NOT want connections to occur. A single-layer PCB has the substrate with one layer of copper foil on the top (see Figure 1).

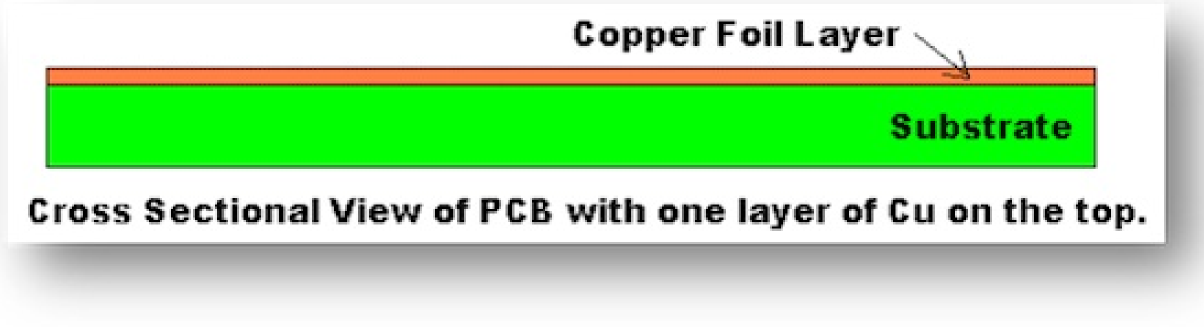


Figure 1: **Single layer PCB**

A double layer PCB (see Figure 2) has two layers of copper foil (one on the top and one on the bottom).

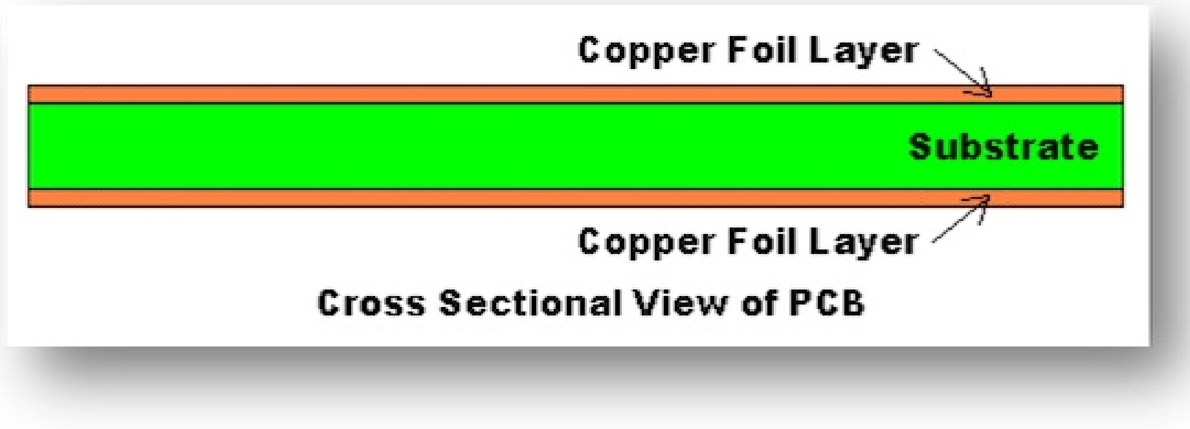


Figure 2: A double-layer PCB

If more than two layers are required due to increased complexity of the PCB, other layers of copper can be built-up or added to the ones shown above (usually in pairs). For example, a 4-layer PCB can be made up of two double-layered PCBs laminated (sandwiched) together with a core material in between. Made out of epoxy/fiber, this core layer is called a prepreg (pre-impregnated), and it insulates and supports the other layer structures. It is common for modestly complex boards to have 6, 8, or 10 layers (with increased manufacturing cost). Some highly complex PCBs have up to 32 layers or more of traces and copper planes (see Figure 3).

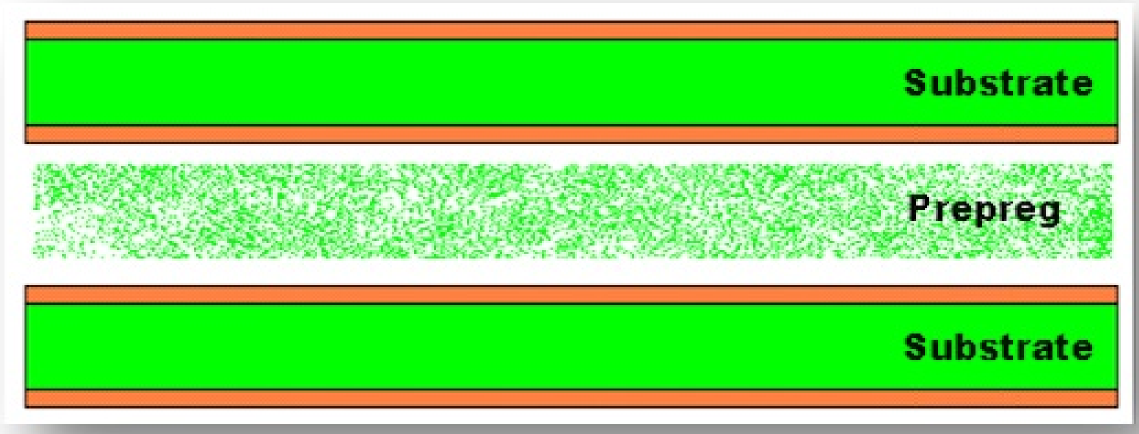


Figure 3: Multilayer PCB

The height of the substrate is usually the thickness of one or multiple sheets of laminate material and is usually much smaller than the height of the core prepreg material layer.

Multilayer PCB – A PCB with more than one copper foil layer. The layers are preferably renamed in the design tool to unique names (such as power or ground) as desired by the user.

Layer Stack Up – The copper organization of multiple layer PCBs with the intent of having specific signal and ground planes on certain layers for routing convenience and electromagnetic shielding purpose. A four-layer board will typically have the following layer structure, where the top and bottom layers are reserved for signal routing and the inner layers are reserved for ground and power planes:

- Copper Top
- Inner 1
- Inner 2
- Copper Bottom

Finished PCB Height - Standard finished PCB thicknesses are commonly found as shown – this thickness includes all copper, substrate and prepeg layers:

- .031" (also .039" is common)
- .062" (most commonly used size)
- .093"
- .125"

Shown in Figure 4 is a more realistic layer stackup of a four-layer PCB showing the various thicknesses of the layer structures from a typical PCB manufacturer yielding the common 0.062" finished PCB height.

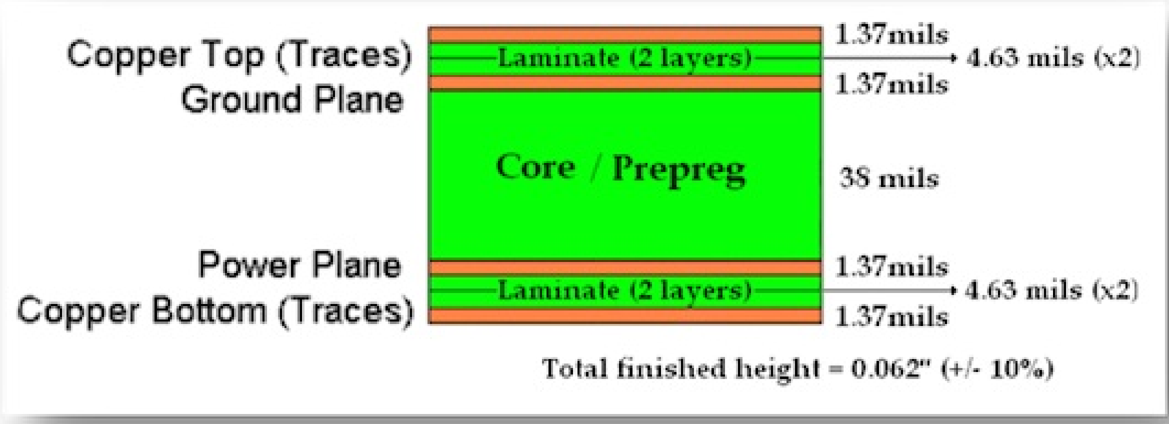


Figure 4: Four-layer PCB

PCB Terminology continued

Bare PCB – A finished PCB without electrical parts and other components (unpopulated or unassembled).

Assembly Drawing – A mechanical and schematic drawing with engineering notations specifying how (and in what order) the PCB needs to be assembled and packaged. For automated assembly, the assembly drawing also contains information about solder paste and other information for component population and other board manufacturing purposes. Some basic mechanical assembly information can be completed within the PCB layout tool; however, complex mechanical assembly information typically needs to be completed in an external CAD package.

Trace – A portion of the copper foil that is remaining on the PCB after the etching process for a signal connection (net) from point A to point B. Traces for nets can have various widths which need to be sized by maximum current expected to run through the trace.

Plane – A large portion of the copper foil that is remaining on the PCB for a signal connection that attaches to many components. This is a layer where little copper is etched away. Power and ground signals are typically connected to a plane, since the power and ground needs to route to many components on the PCB. Since a plane has a lot of copper, and the plane covers a large area, traces can be routed to the plane with an effective lower resistance and inductance from a connection point to the plane. This will create smaller voltage drops (versus placing discrete traces) when the components conduct power and ground current and thus will create a design with lower power dissipation. Also, circuits with power and ground planes will be less likely to radiate electromagnetic (EM) energy, as well as be less susceptible to EM energy (such as 50/60 Hz AC line noise). Figure 5 is an example of a four-layer PCB with power and ground planes:

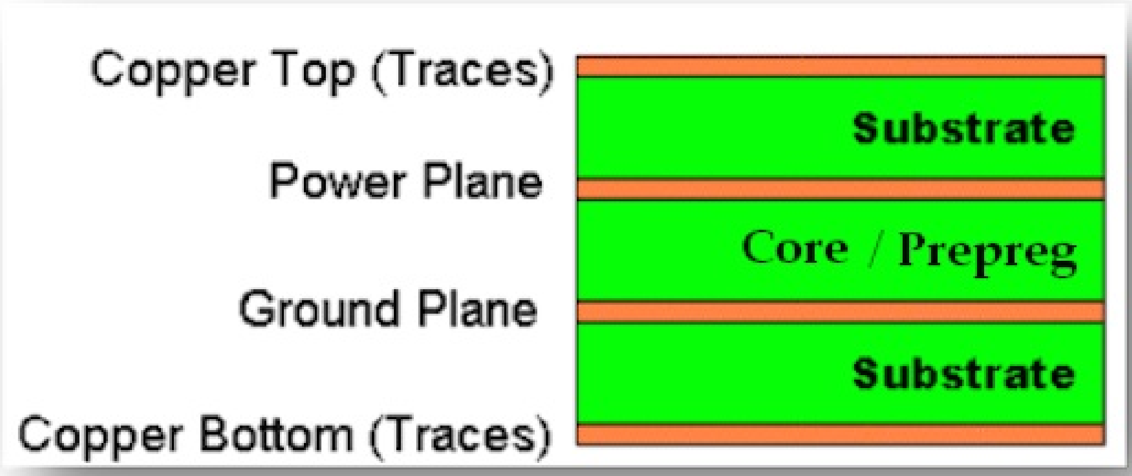


Figure 5: Four-layer PCB with power and ground planes

Substrate – The non-conductive material (substrate) is usually a fiberglass epoxy. A flame-retardant material called FR-4 is the most common PCB material used in North America. For specialized applications, other materials can be used (such as G10) but these other materials will typically exhibit lower flammability resistance. Other substrate materials (with favorable dielectric properties) are available and sometimes materials vary by geographic region.

Copper thickness (weight in ounces) – The copper foil thickness is measured not in linear dimensions, but with the weight of the copper if poured onto a 1 square foot sheet. For instance, a common copper thickness (weight) is 1 oz. This means 1 oz. of copper per square foot. A 2 oz. weight would indicate 2 oz. per square foot. Some common copper weights are as follows:

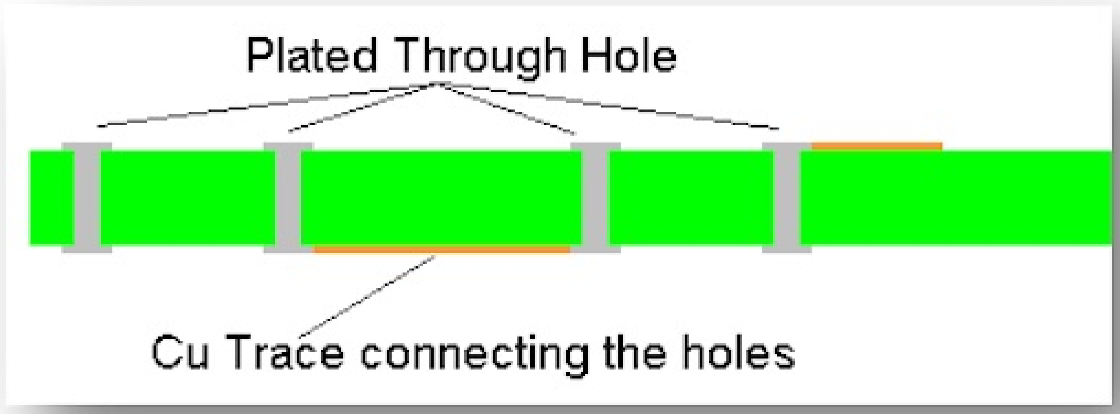
- ¼ oz (not common since it is so thin it can vaporize with a soldering iron)
- ½ oz (common), Copper will be 0.7 mils thick.
- 1 oz (most common), Copper will be 1.35 to 1.37 mils thick.
- 2 oz (very common for higher current boards)
- 3 oz (not common except for high current applications as it acts as a heat sink, also difficult to solder to since it drains the heat from the soldering iron)

The copper weight and width needs to be considered when designing a PCB. Consider the current in the traces versus the added cost for higher copper weight when deciding on the copper weight to use. For application specific information, the user can look up quantitative design criteria by referring to the tables in IPC-2221, Generic Standard on Printed Board Design http://www.ipc.org/toc/ipc-2221a.pdf.

PCB Styles – Most modern PCBs typically have a combination of surface mount and through-hole parts, however PCBs can generally be organized into two categories, plated through hole and surface mount technology.

Plated through hole (PTH or also indicated as TH) – In this type of PCB, the majority of components will have wire leads that extend from the part that are inserted into the holes of the PCB which are copper plated for soldering.

Figure 6 is a cross section of a two-layer board with four PTHs. Also shown are two copper traces on the board. Figure 6 (bottom) is connecting two PTHs together.



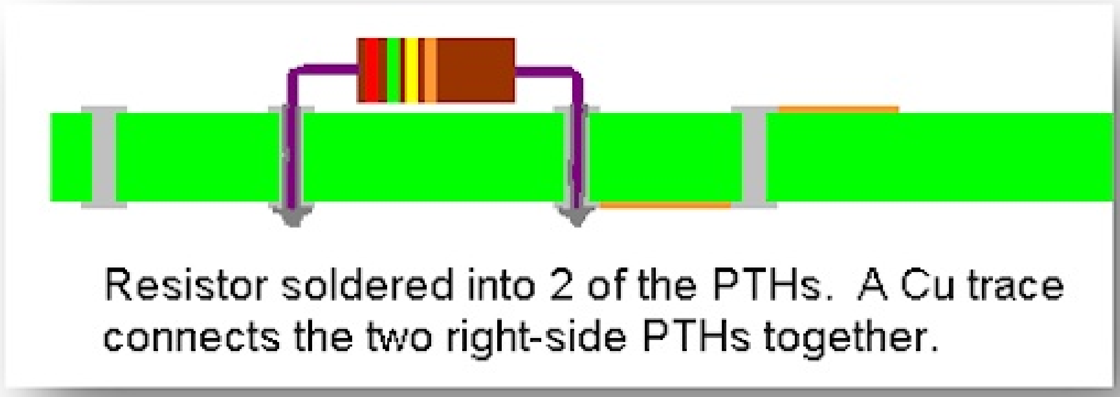


Figure 6 (top and bottom): Cross section of a two-layer board.

• *Surface mount technology (SMT)* – In this type of PCB, the majority of parts have no leads that go through the PCB. These parts will typically have pads or leads that get soldered to the surface of the board. These components are referred to as surface mount devices (SMDs).

Below is a SMT PCB with four SMT pads (see Figure 7). A resistor is shown soldered to the board. With SMT, the designer has the option to place parts on the bottom of the board (since there are no PTHs located near the SMDs to obstruct the placement of the SMD). SMDs can typically be manufactured smaller than their through-hole equivalents, thus the use of SMT can usually lead to significant increases in the parts density within a PCB.

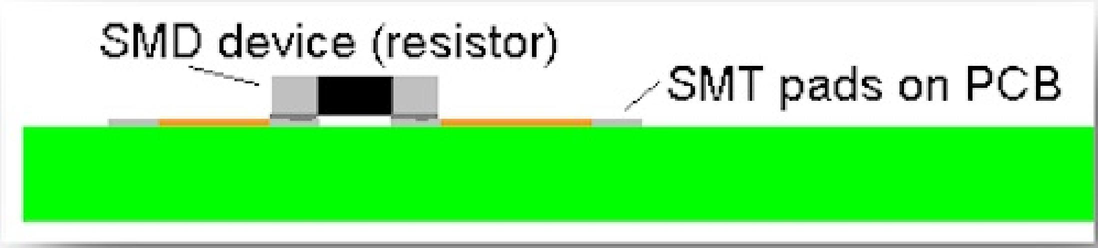


Figure 7: SMT PCB with four SMT pads

When considering PTH or SMT technology for the PCB, be careful to select the correct technology and components for the application. Prototyping will generally work better with PTH or larger SMD style components; however the larger technology may not meet final size or density constraints on a final production design. Conversely, SMT style PCBs can be packed very tightly together and can typically yield lower manufacturing costs (per board in larger volumes) through automated assembly. However, hand soldering small SMDs or components with ball-grid-array (BGA) technology can be very tricky for all but those with very experienced soldering skills.   Consider these important tradeoffs when selecting components and designing the PCB. Some components will be available in both through-hole and SMD styles, whereas others will be available in only one style.

Solder Mask – This is a shielding or insulating layer (this is also what makes common PCBs green) installed over the top and bottom layer to cover copper traces. The layer usually is not installed over the top of pads since this exposes the copper area to be soldered. This layer protects these covered conductors so that solder will not inadvertently adhere to them (creating shorts) when connecting nearby parts to pads or through-hole connections.

Silkscreen (also called Screen Print) - This is the final layer applied over the top solder mask and also over the bottom solder mask (if required) to display part outline and reference designator information on the board. Besides displaying part outline and reference designator information, the silkscreen can display orientation information (such as anode/cathode direction) and any other symbol or textual information the user wants to be displayed on the board (such as internal board model numbering, logos, PCB revision numbering and other manufacturing or standards marking).

Footprint – The mechanical and electrical pattern of the part. The information included in the part typically includes the copper landpattern, mechanical outline and any important dimensional information of the component such as the body information, height and size. The footprint for a component will typically include a silkscreen layer displaying part outline and orientation information and the copper pads or holes (landpattern) that are associated with the parts. As in the case of connectors, the footprint of the part may extend beyond the board outline, so the detail in the footprint should give the designer adequate information for part placement, orientation, and copper connections.

Landpattern – This is the pattern of pads or holes on the circuit board, sometimes including the part body information. Standards from the IPC, a global trade association, are often used to accurately specify the required copper size and shape required to ensure the proper bonding and adhering of the component's pads or wires to the PCB. The landpattern can include non-electrical pad information such as thermal relief pads or heat sink patterns that may be required.

Part 2 of this article will discuss an example design flow.

If you liked this feature, and would like to see a collection of related features delivered directly to your inbox, sign up for the PCB Design Center newsletter here.

Also See:
Successful PCB grounding with mixed-signal chips