Introduction to Printed Circuit Board Prototyping

An Online Continuing Education Course for Engineers

Course Number: E-2005

Credit: 2 Hours / 2 PDH / 2 CPD

Introduction to Printed Circuit Board Prototyping (2 PDH)

Author: Danny R. Graves, MSEE, PE

Introduction

A working electronic prototype is often required for marketing or testing purposes. This prototype part could have a circuit that is hand-built on perf-board or could utilize other quick, cheap, and dirty methods that may work but don't look professional or provide test results indicative of a "production" part. If you want your product to look the best (and often work the best), you should get a printed circuit board (PCB) designed and built. Even better, you should have the PCB designed such that it can be produced on a mass scale with automated machinery without major redesign. But, how do you design a PCB and get prototypes built? I will cover the basics of this process in this course.

The Printed Circuit Board

Basic Construction

A printed circuit Board (PCB) is basically a hunk of laminated insulator material with conductive traces on the surface and sometimes buried *inside* the insulator. The conductive traces interconnect the components (for example, transistors, diodes, resistors, LEDs, etc) that are placed on the surface(s) of the PCB. PCBs can have electronic components placed on both sides in the final assembly.

A PCB is made up of several components. These components are the laminate, conductive layers, lands, holes, solder mask, and silkscreen.

The main body of the PCB is made up of different layers laminated together**.** The laminate material is the foundation of the PCB. This can be various materials such as FR2, FR4, CEM-1. These materials are made of such things as epoxy and fiberglass. For

flexible PCBs, polyimide film laminates are used. Laminates are electrical insulator materials. The choice of substrate material greatly affects the cost of an individual PCB.

Solder mask

The solder mask is the material that is placed over the areas to which you do not want solder to adhere. Solder mask is the stuff that is usually green, although it can be a variety of colors such as red, blue, or black, and covers the surface of the PCB except the component pads. Figure 1 shows an example of solder mask.

There are a variety of different types of solder masks and processes by which to apply them. Screening the solder mask is an inexpensive but inexact way to apply the mask material. Liquid photo imageable (LPI) is a slightly more expensive but more precise way to apply the mask.

SMOBC stands for Solder Mask Over Bare Copper. SMOBC is important if the assembly will be wave-soldered. Wave-soldering is when the PCB assembly is soldered by a machine which passes the surface of the PCB assembly (the surface(s) where the component leads are located), via a conveyer belt, over a vat of molten solder. PCBs which are not SMOBC tend to result in wrinkled solder mask when wave soldered. This is because the solder under the solder mask melts, when the assembly travels through the wave solder, and "bubbles up" the solder mask.

Surface mount (SMT) components don't have wire leads that protrude *through* the PCB but instead component pads that sit on top of the PCB pads. SMT components are usually placed on a PCB by first, placing solder paste on the board pads, then setting each of the component's pads in this paste. The entire board assembly is then sent through a "reflow" oven that melts the paste and solders the component into position. Alternately, SMT components can be glued into position (the glue dot is placed on the component body not the electrical pads) then sent through a wave-solder. In either process (wave-solder or reflow oven), the PCB's solder mask must have openings where the component electrical connections are located.

It is often advantageous to plate the exposed surfaces of the copper conductors and component lead pads with a material, for example silver, to inhibit tarnishing and preserve solderability. In other words, plating ensures effective soldering of the components to the PCB after the PCBs have been stored in a warehouse for a while. Another option to preserve solderability is hot air solder leveling (HASL). HASL involves applying solder to the exposed copper (solder plating) and leveling (flattening it out) it by using hot air blown over the surface of the circuit board.

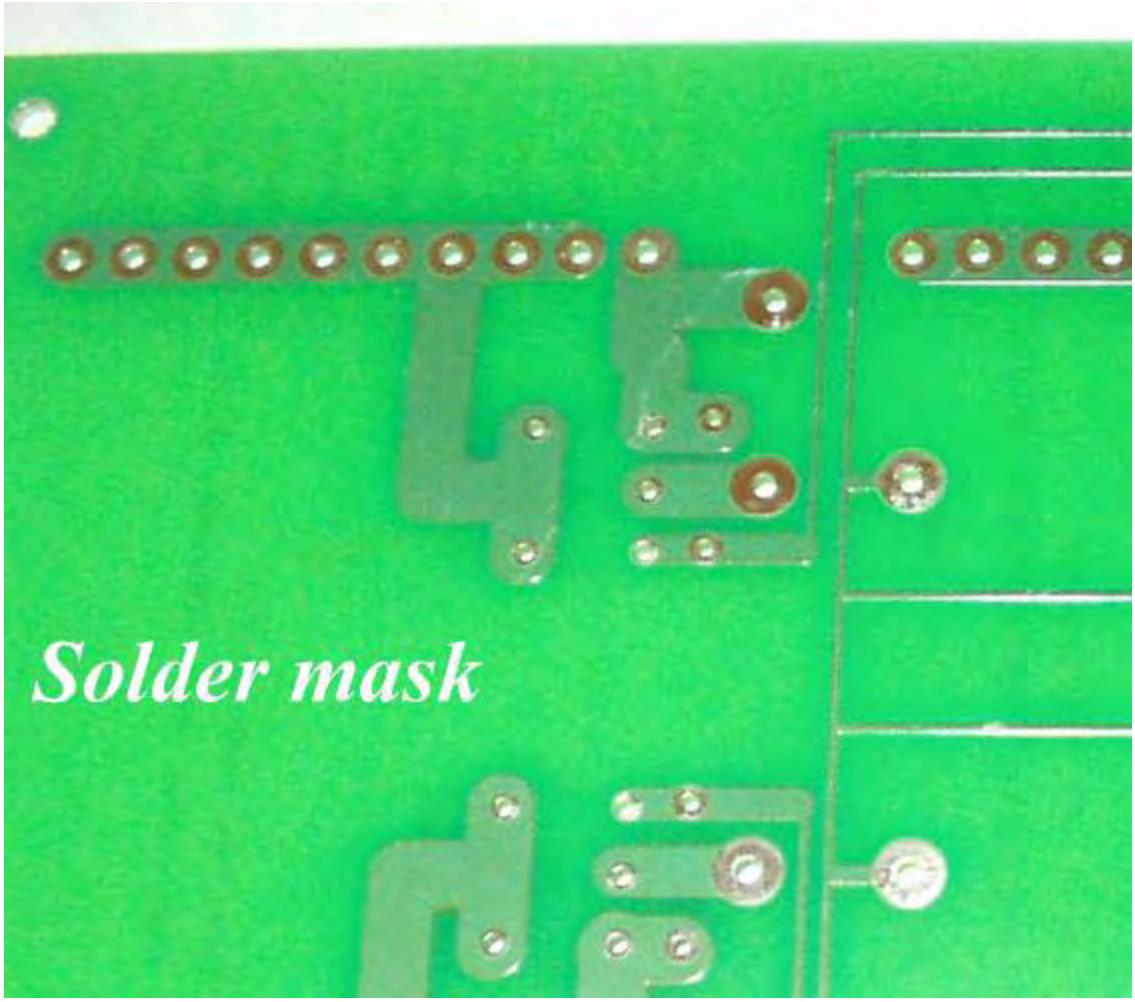


Figure 1. An example of printed circuit board solder mask.

Silkscreen

A silkscreen is a printing, *in ink,* of characters or graphics on the surface of the PCB. A manufacturing department may require a silkscreen to show the location and reference designator of the components for inspection and re-work purposes. It's possible to print characters in copper but it's often advantageous to use a non-conductive method of printing, especially in high-density designs. Figure 2 shows an example of silkscreen.

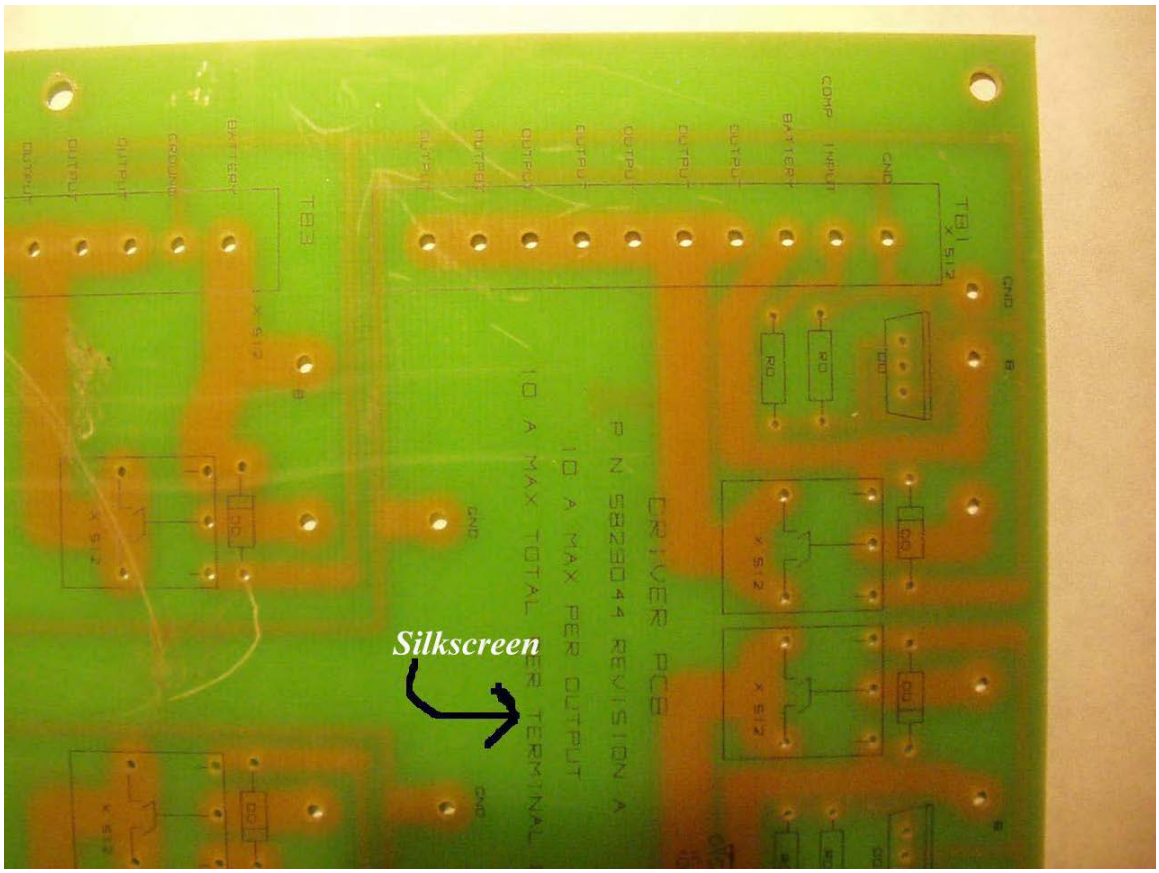


Figure 2. An example of silk-screening on a printed circuit board.

Traces

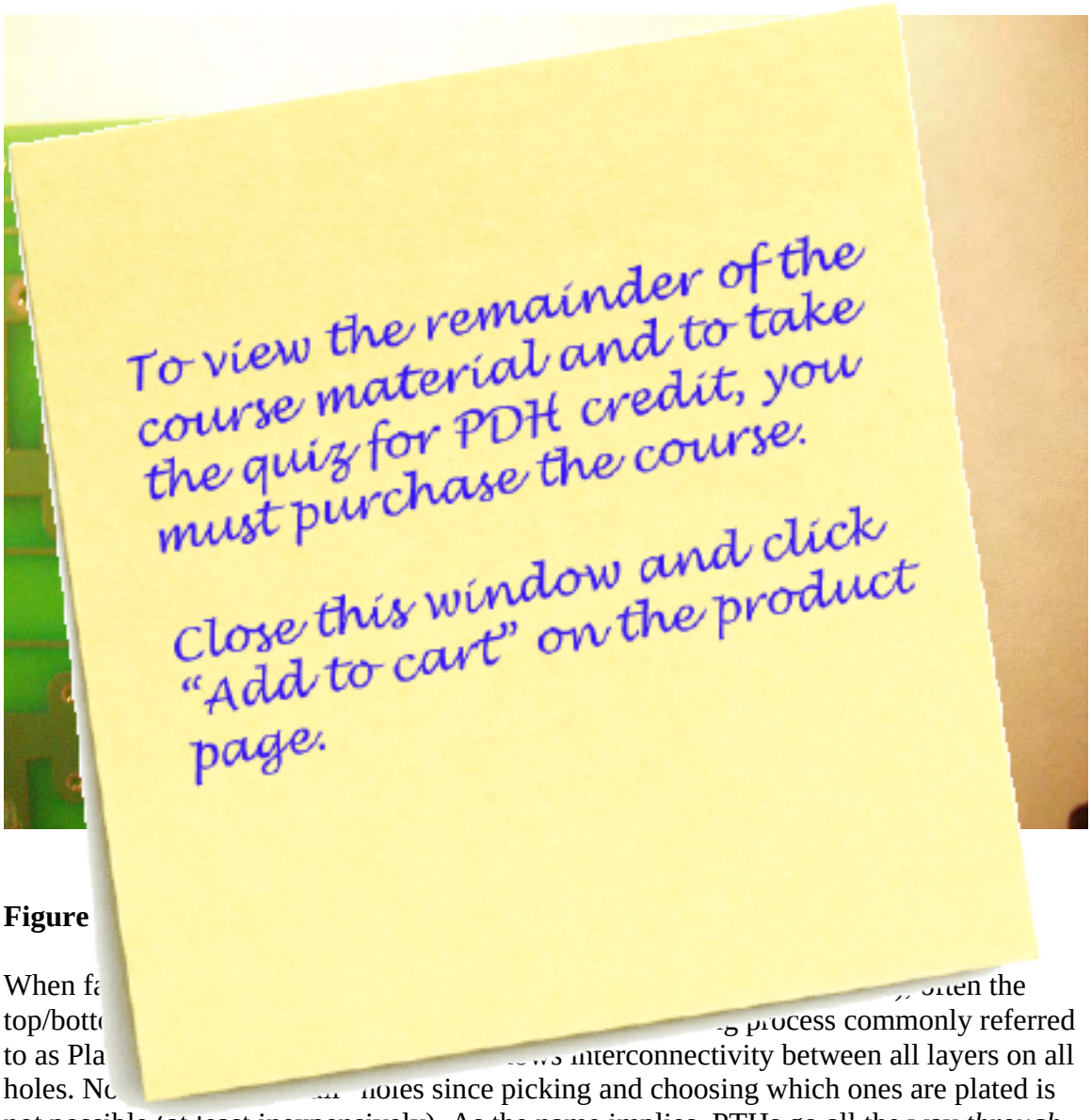
Copper is what makes the electrical connections on the PCB. The strips of copper connecting components are called traces or tracks. Figure 3 shows an example of a trace. A layer of the PCB that contains traces or a continuous sheet of copper (like a ground plane) is called a conductive layer. Continuous layers are often used for electromagnetic shielding or heat-sinking purposes. A PCB can have one or more conductive layers. When you hear someone say they have a 4-layer board it means they have a PCB with 4 conductive layers, usually one on each external surface and two in the middle of the insulator material. An odd number of conductive layers are generally not used because of warping effects on the PCB.

Holes

There are holes in most PCBs for mechanical mounting, mounting and soldering components, and interconnecting the conductive layers. A land is the area around a hole to which a component is soldered. Figure 3 shows an example of a land.

Via holes are what connects the different conductive layers together. Vias are conductively plated on their interior surface and traces on the appropriate layers are

connected to the vias. Vias can protrude all the way through a PCB or they can be buried. "Buried" means that the via interconnects internal layers and cannot be seen from the exterior of the PCB. A higher density of conductors is possible with buried via holes since some layers do not have space taken up by the via.



Figure

When fa… often the top/bott… process commonly referred to as Pla… interconnectivity between all layers on all holes. No… holes since picking and choosing which ones are plated is not possible (at least inexpensively). As the name implies, PTHs go all the way *through* the board. Note that a PTH and a non-buried via are essentially the same thing. Also, note that specifying PTH adds cost to a board.