Basics of Manufacturing Printed Circuit Boards

by Jim Usery

In electronics, printed circuit boards, or PCBs, are used to mechanically support electronic components which have their connection leads soldered onto copper pads in surface mount applications or through drilled holes in the board and copper pads for soldering the component leads in thru-hole applications. A board design may have all thru-hole components on the top or component side, a mix of thru-hole and surface mount on the top side only, a mix of thru-hole and surface mount components on the top side and surface mount components on the bottom or circuit side, or surface mount components on the top and bottom sides of the board.

The boards are also used to electrically connect the required leads for each component using conductive copper traces. The component pads and connection traces are etched from copper sheets laminated onto a non-conductive substrate. Printed circuit boards are designed as single sided with copper pads and traces on one side of the board only, double sided with copper pads and traces on the top and bottom sides of the board, or multilayer designs with copper pads and traces on top and bottom of board with a variable number of internal copper layers with traces and connections.

Single or double sided boards consist of a core dielectric material, such as FR-4 epoxy fiberglass, with copper plating on one or both sides. This copper plating is etched away to form the actual copper pads and connection traces on the board surfaces as part of the board manufacturing process. A multilayer board consists of a number of layers of dielectric material that has been impregnated with adhesives, and these layers are used to separate the layers of copper plating. All of these layers are aligned and then bonded into a single board structure under heat and pressure. Multilayer boards with 48 or more layers can be produced with today's technologies.

In a typical four layer board design, the internal layers are often used to provide power and ground connections, such as a +5V plane layer and a Ground plane layer as the two internal layers, with all other circuit and component connections made on the top and bottom layers of the board. Very complex board designs may have a large number of layers to make the various connections for different voltage levels, ground connections, or for connecting the many leads on ball grid array devices and other large integrated circuit package formats.

There are usually two types of material used to construct a multilayer board. Pre-preg material is thin layers of fiberglass pre-impregnated with an adhesive, and is in sheet form, usually about .002 inches thick. Core material is similar to a very thin double sided board in that it has a dielectric material, such as epoxy fiberglass, with a copper layer deposited on each side, usually .030 thickness dielectric material with 1 ounce copper layer on each side. In a multilayer board design, there are two methods used to build up the desired number of layers. The core stack-up method, which is an older technology, uses a center layer of pre-preg material with a layer of core material above and another layer of core material below. This combination of one pre-preg layer and two core layers would make a 4 layer board.

The film stack-up method, a newer technology, would have core material as the center layer followed by layers of pre-preg and copper material built up above and below to form the final number of layers required by the board design, sort of like Dagwood building a sandwich. This method allows the manufacturer flexibility in how the board layer thicknesses are combined to meet the finished product thickness requirements by varying the number of sheets of pre-preg in each layer. Once the material layers are completed, the entire stack is subjected to heat and pressure that causes the adhesive in the pre-preg to bond the core and pre-preg layers together into a single entity.

The process of manufacturing printed circuit boards follows the steps below for most applications:

Basic Steps for Manufacturing Printed Circuit Boards:

1. Setup - the process of determining materials, processes, and requirements to meet the customer's specifications for the board design based on the Gerber file information provided with the purchase order.

2. Imaging - the process of transferring the Gerber file data for a layer onto an etch resist film that is placed on the conductive copper layer.

3. Etching - the traditional process of exposing the copper and other areas unprotected by the etch resist film to a chemical that removes the unprotected copper, leaving the protected copper pads and traces in place; newer processes use plasma/laser etching instead of chemicals to remove the copper material, allowing finer line definitions.

4. Multilayer Pressing - the process of aligning the conductive copper and insulating dielectric layers and pressing them under heat to activate the adhesive in the dielectric layers to form a solid board material.

5. Drilling - the process of drilling all of the holes for plated through applications; a second drilling process is used for holes that are not to be plated through. Information on hole location and size is contained in the drill drawing file.

6. Plating - the process of applying copper plating to the pads, traces, and drilled through holes that are to be plated through; boards are placed in an electrically charged bath of copper.

7. Second Drilling - this is required when holes are to be drilled through a copper area but the hole is not to be plated through. Avoid this process if possible because it adds cost to the finished board.

8. Masking - the process of applying a protective masking material, a solder mask, over the bare copper traces or over the copper that has had a thin layer of solder applied; the solder mask protects against environmental damage, provides insulation, protects against solder shorts, and protects traces that run between pads.

9. Finishing - the process of coating the pad areas with a thin layer of solder to prepare the board for the eventual wave soldering or reflow soldering process that will occur at a later date after the components have been placed.

10. Silk Screening - the process of applying the markings for component designations and component outlines to the board. May be applied to just the top side or to both sides if components are mounted on both top and bottom sides.

11. Routing - the process of separating multiple boards from a panel of identical boards; this process also allows cutting notches or slots into the board if required.

12. Quality Control - a visual inspection of the boards; also can be the process of inspecting wall quality for plated through holes in multilayer boards by cross-sectioning or other methods.

13. Electrical Testing - the process of checking for continuity or shorted connections on the boards by means applying a voltage between various points on the board and determining if a current flow occurs. Depending upon the board complexity, this process may require a specially designed test fixture and test program to integrate with the electrical test system used by the board manufacturer.

At Innovative Circuits, Inc., we assemble a variety of surface mount and thru-hole boards each day for our customers. We also provide printed circuit board layout design services along with conversions from thru-hole designs to surface mount designs for our customers. We do not manufacture bare printed circuit boards ourselves but we do work with a number of board houses to produce the bare boards that we use every day to assemble our customers' products. As a result, we have become very familiar with the board manufacturing processes and we wanted to share our knowledge with others who may not have our level of exposure to printed circuit boards.

Jim Usery
Sales and Marketing Director
Innovative Circuits, Inc.
311A S Parkway St
Corinth, MS 38834
office 662-287-2007
toll free 866-887-7381
fax 662-665-9275
email jusery@icimfg.com

Visit our website at www.icimfg.com for more information about Innovative Circuits, Inc. and the services and expertise that we provide to our customers, or for additional technical articles. You can also sign up for our free Ezine, “PCB Information”, at the bottom of the Home Page at www.icimfg.com.