A Guide To The PCB Manufacturing Process

The printed circuit board (PCB) acts as the linchpin for almost all of today's modern electronics. If the device needs to do some sort of computation — such as is the case even with simple items like a digital clock — chances are there's a PCB inside of it. PCBs bring electronics to life by routing electrical signals where they need to go to satisfy all of the device's electronic requirements. For this to happen, PCBs are laid with a network of paths outlined in copper wire. It's these copper pathways that allow PCBs to direct electrical currents around their surface.

There are three main types of circuit board that get manufactured on a consistent basis, and it's important to understand the differences between each so you can decide the right circuit board for your requirements. The three main types of circuit boards in current manufacture are:

- **Single-Sided Circuit Boards:** These boards have rigid laminate of woven glass epoxy material, which is then covered on one side with a copper coating that is applied in varying thicknesses depending on the application

- **Double-Sided Circuit Boards:** Double-sided boards have the same woven glass epoxy base as single-sided boards — however, in the case of a double-sided board, there is copper coating on both sides of the board, also to varying thicknesses depending on the application

- **Multi-Layer Boards:** These use the same base material as single and double-sided boards, but are made with copper foil instead of copper coating — the copper foil is used to make "layers," alternating between base material and copper foil until the number of desired layers is reached.

Now that you know a little bit about the basic types of PCB available, we'll walk through the basic anatomy of a PCB to better familiarize you with it before moving on to how to build a PCB.

The Parts of a PCB: Basic Makeup of a Printed Circuit Board

There are three main parts to a PCB:

- **Substrate Board:** The first, and most important, is the substrate board, usually made of fiberglass. Fiberglass is used because it provides a core strength to the PCB and helps resist breakage. Think of the substrate board as the PCB's "skeleton".

- **Copper Layer:** Depending on the board type, this layer can either be copper foil or a full-on copper coating. Regardless of which approach is used, the point of the copper is still the same — to carry electrical signals to and from the PCB, much like your nervous system carries signals between your brain and your muscles.

- **Solder Mask:** The third and final piece of the PCB is the solder mask, which is a layer of polymer that helps protect the copper so that it doesn't short-circuit from coming into contact with the environment. In this way, the substrate acts as the PCB's "skin".

Now that we've gone over the basics of PCBs and PCB anatomy, we'll walk through the whole process of how to build a PCB.

How Is a PCB Manufactured?

The steps of the PCB manufacturing process are as follows:

Step One: Designing the PCB

The beginning step of any PCB manufacture is, of course, the design. PCB manufacture and design always starts with a plan: the designer lays out a blueprint for the PCB that fulfills all the requirements as outlined. The most commonly-used design software used by PCB designers is a software called Extended Gerber — also known as IX274X.

When it comes to PCB design, Extended Gerber is an excellent piece of software because it also works as an output format. Extended Gerber encodes all the information that the designer needs, such as the number of copper layers, the amount of solder masks needed and the other pieces of component notation. Once a design blueprint for the PCB is encoded by the Gerber Extended software, all the different parts and aspects of the design are checked over to make sure that there are no errors.

Once the examination by the designer is complete, the finished PCB design is sent off to a PCB fabrication house so that the PCB can be built. On arrival, the PCB design plan undergoes a second check by the fabricator, known as a Design for Manufacture (DFM) check. A proper DFM check ensures that the PCB design fulfills, at minimum, the tolerances required for manufacture.

Step Two: Printing the PCB Design

After all the checks are complete, the PCB design can be printed. Unlike other plans, like architectural drawings, PCB plans don't print out on a regular 8.5 x 11 sheet of paper. Instead, a special kind of printer, known as a plotter printer, is used. A plotter printer makes a "film" of the PCB. The final product of this "film" looks much like the transparencies that used to be used in schools — it's essentially a photo negative of the board itself.

The inside layers of the PCB are represented in two ink colors:

- **Black Ink:** Used for the copper traceries and circuits of the PCB
- **Clear Ink:** Denotes the non-conductive areas of the PCB, like the fiberglass board

On the outer layers of the PCB design, this trend is reversed — clear ink refers to the line of copper pathways, but black ink also refers to areas where copper will be removed.

Each PCB layer and the accompanying solder mask gets its own sheet, so a simple two-layer PCB needs four sheets — one for each layer and one each for the accompanying solder mask.

After the film sheets are printed, they're lined up and a hole, known as a registration hole, is punched through them. The registration hole is used as guide to align the films later on in the process.

Step Three: Printing the Copper for the Interior Layers

Step three is the first step in the process where the manufacturer starts to make the PCB. After the PCB design is printed onto a piece of laminate, copper is then pre-bonded to that same piece of laminate, which serves as the structure for the PCB. The copper is then whittled away to reveal the blueprint from earlier.

Next, the laminate panel is covered by a type of photo-sensitive film called the resist. The resist is made of a layer of photo-reactive chemicals that harden after they're exposed to ultraviolet light. The resist allows technicians to get an perfect match between the photos of the blueprint and what's printed to the photo resist.

Once the resist and the laminate board are lined up — using the holes from earlier — they receive a blast of ultraviolet light. The ultraviolet light passes through the translucent parts of the film, hardening the photo resist. This indicates areas of copper that are meant to be kept as pathways. In contrast, the black ink prevents any light from getting to the areas that aren't meant to harden so that they can later be removed.

Once the board has been prepared, it is washed with an alkaline solution to remove any of the leftover photo resist. The board is then pressure-washed to remove anything left on the surface and left to dry.

After drying, the only resist that should be left on the PCB is on top the copper that remains as part of the PCB when it's finally popped free. A technician looks over the PCBs to make that there are no errors. If no errors are present, then it's on to the next step.

Step Four: Getting Rid of the Unneeded Copper

The next stage in the process is that of removing the unwanted copper. Much like the alkaline solution from earlier, another powerful chemical is used to eat away at the copper that is not covered by photo resist. Once the unprotected copper is removed, the hardened photo resist from earlier needs to be removed, as well. Another solvent is used, leaving only the copper necessary for the PCB.

Note that when it comes to removing the unwanted copper from your PCB, heavier boards may require more copper solvent or more exposure to the solvent.

Step Five: Inspection and Layer Alignment

After each of the PCB's layers have been cleaned, they're ready for layer alignment and an optical inspection. The holes from earlier are used to align the inner and outer layers. To align the layers, a technician places them on a type of punch machine known as an optical punch. The optical punch drives a pin down through the holes to line up the layers of the PCB.

Following the optical punch, another machine performs an optical inspection to make sure there are no defects. This optical inspect is incredibly important because once the layers are placed together, any errors that exist can't be corrected. To confirm that there are no defects, the AOI machine compares the PCB to be inspected with the Extended Gerber design, which serves as the manufacturer's model.

After the PCB has passed inspection — that is, neither the technician nor the AOI machine found any defects — it moves onto the last couple steps of PCB manufacture and production.

Step Six: Bonding the PCB Layers

At step six in the process, the PCB layers are all together, waiting to be bonded. Once the layers have been confirmed as being defect-free, they're ready to be fused together. The PCB bonding process is done in two steps: the lay-up step and the bonding step.

The outside of the PCB is made of pieces of fiberglass that have been pre-soaked/pre-coated with an epoxy resin. The original piece of substrate is also covered in a layer of thin copper foil that now contains the etchings for the copper traces. Once the outer and inner layers are ready, it's time to push them together.

The sandwiching of these layers is done using metal clamps on a special press table. Each layer fits onto the table using a specialized pin. The technician doing the bonding process starts by placing a layer of pre-coated epoxy resin — known as pre-impregnated or prepreg — on the alignment basin of the table. A layer of substrate is placed over the pre-impregnated resin, followed by a layer of copper foil. The copper foil is in turn followed by more sheets of pre-impregnated resin, which are then finished off with a piece of aluminum and one last piece of copper known as a press plate.

Once the copper press plate is in place, the stack is ready to be pressed. The technician takes it over to a mechanical press and presses the layers down and together. As part of this process, pins are then punched down through the stack of layers to ensure that they're fixed properly.

If the layers are fixed properly, the PCB stack is taken to the next press, a bonding press. The bonding press uses a pair of heated plates to apply both heat and pressure to the stack of layers. The heat of the plates melts the epoxy inside of the prepeg — it and the pressure from the press combine to fuse the stack of PCB layers together.

Once the PCB layers are pressed together, there's a little bit of unpacking that needs to be done. The technician needs to remove the top press plate and the pins from earlier, which then allows them to pull the actual PCB free.

Step Seven: Drilling

After the PCB is pulled free, holes can be drilled into the board. Any components that need to be attached to the PCB later rely on these holes, so they need to be precise. A microscopic drill is used to put the holes in, because each hole is less than a hair's breadth in size.

Before drilling, an X-ray machine is used to locate the drill spots. Then, registration/guiding holes are drilled so that the PCB stack can be secured before the more specific holes are drilled. When it comes time to drill these holes, a computer-guided drill is used to make the holes themselves, using the file from the Extended Gerber design as a guide.

Once the drilling is complete, any additional copper that's left over at the edges is filed off.

Step Eight: PCB Plating

After the panel has been drilled, it's ready to be plated. The plating process uses a chemical to fuse all of the different layers of the PCB together. After being cleaned thoroughly, the PCB is bathed in a series of chemicals. Part of this bathing process coats the panel in a micron-thick layer of copper, which is deposited over the top-most layer and into the holes that have just been drilled.

Before the holes are filled with copper, they simply serve to expose the fiber glass substrate that makes up the panel's insides. Bathing those holes in copper covers the walls of the previously-drilled holes.

Steps Nine and Ten: Imaging and Plating the Outer Layer

Earlier in the process (Step 3) a photo resist was applied to the PCB panel. In Step 9, it's time to apply another layer of photo resist. However, this time the photo resist is only applied to the outside layer, since it still needs to be imaged. Once the outer layers have been coated in photo resist and imaged, they're plated in the exact same way the interior layers of the PCB were plated in the previous step. However, while the process is the same, the outer layers get a plating of tin to help guard the copper of the outside layer.

Step Eleven: The Last Etching

When it comes time to etch the outside layer for the last time, the tin guard is used to help protect the copper during the etching process. Any unwanted copper is removed using the same copper solvent from earlier, with the tin protecting the valued copper of the etching area.

Once all the unwanted copper is removed, the PCB's connections have been properly established and it's ready for solder masking.

Step Twelve: Applying the Solder Mask

To make the panels ready for solder mask application, they're cleaned. Once the PCB panels have been cleaned, an ink epoxy is applied, along with a solder mask film. The boards are blasted with ultraviolet light to mark out certain portions of the solder mask for removal.

After the unwanted pieces of solder mask have been removed, the PCB is placed into an oven and baked so that the solder mask will cure.

Steps Thirteen and Fourteen: Finishing the PCB and Silkscreening

As part of the finishing process, the PCB is plated with gold and/or silver to increase the ability for solder to stick to it. After the PCB has been gold or silver-plated, as necessary, it is silk-screened. The silk-screening process prints all of the vital information on the PCB, such as manufacturer marks, company ID numbers and warning labels.

Once the PCB has been plated and silk-screened with the correct information, it can be sent along to the final curing stage.

Step Fifteen: Electrical Reliability Testing

After the PCB has been coated and cured (if necessary), a technician performs a battery of electrical tests on the different areas of the PCB to ensure functionality. The main tests that are performed are the circuit continuity and isolation tests. The circuit continuity test checks for any disconnections in the PCB, known as "opens." The circuit isolation test, on the other hand, checks the isolation values of the PCB's various parts in order to check if there are any shorts. While the electrical tests mainly exist to ensure functionality, they also work as test of how well the initial PCB design stood up to the manufacturing process.

In addition to basic electrical reliability testing, there are other tests that can be used to determine if a PCB is functional. One of the main tests used to do this is known as the "bed of nails" test. During this text, several spring fixtures are attached to the test points on the circuit board. The spring fixtures then subject the test points on the circuit board with up to 200g of pressure to see how well the PCB stands up to high-pressure contact at its test points.

If the PCB has passed its electrical reliability testing — and any other testing the manufacturer chooses to implement — it can be moved on to the next step: cutting.

Step Sixteen: Cutting and Profiling

The last step of the PCB manufacturing process is the cutting and scoring of the PCB. This involves cutting out the different PCBs from original panel. There are two ways that PCBs can be cut from their original panels:

- Using a CNC machine or router, which cuts out small tabs around the edges of the PCB

- Using a V-groove, which cuts a diagonal channel along the sides of the board

No matter which way you use, your PCB will be able to easily cime free of the construction paneling.
Usually, PCB panels have the individual boards — or larger arrays, if applicable — routed out and scored so they can be broken off of the construction board after they've been assembled.

Once the boards are broken off the construction board, there comes a final inspection phase:

- The boards are checked over for general cleanliness to ensure that there are no sharp edges, burrs or other manufacturing hazards

- Slots, chamfers, bevels and countersinks are added during the routing and fabrication process, as necessary

- If possible, any shorts are repaired — the shorted boards are then re-tested using the same electrical reliability tests from above

- A visual inspection can be conducted, if necessary, to ensure that boards meet industry specifications and match up to the details laid out in Gerber data: the technician can also use the visual inspection to verify the hole sizes and the physical dimensions of the PCB, if required.