**PCB Production Process Glossary**

|  |  |
| --- | --- |
| **4-Kelvin Test** | A testing method that measures very low electrical resistances accurately using four probes to eliminate contact resistance effects. |
| **Anode** | A copper part connected to the positive side of a power source. |
| **AOI (Automated Optical Inspection)** | A camera system that compares the circuitry on the board to the design and looks for ***shorts*** and ***opens***. |
| **Artwork** | A tool used to transfer the customer's design from the ***Gerber files*** directly onto the surface of the cores. |
| **Backlight Test** | An inspection method that shines a light through the panel to make sure that a thin, even layer of metal has been applied to the surface and inside the holes. |
| **Bed of Nails or Fixture Machine Testing** | A testing method that uses a custom-designed fixture with pins to simultaneously test specific points on a PCB for electrical continuity and other issues. |
| **Bonding** | "Gluing" the stacked layers together into one solid panel. |
| **Cathode** | When a panel is connected to the negative side of a power source, which makes the panel negative, too. |
| **Certificate Of Compliance** | An official document - like a report card - that confirms the customer's PCBs have passed all our quality checks and meet both the customer's, our own, and industry standards. |
| **Circuitry** | Conductors, or "roads", on a PCB that lets electricity travel so that components can communicate. These roads are made of copper and are essential for the PCB to function properly. |
| **Clean Room** | Rooms with very few particles flying around in the air. |
| **Conveyorized Oven** | An oven that moves the PCBs through different heat zones. |
| **Core** | A base material made of glass fiber and epoxy resin that has copper foil on both sides. Used to create the inner layers of a PCB and come in difference thicknesses, both in terms of material and copper. Also called *Copper Clad Laminate*(CCL). |
| **Deposit** | When copper ions attach to a surface, forming a smooth, even layer of copper. |
| **Developer** | A fluid that removes unpolymerized photosensitive film. |
| **Electroless Copper** | A plating process that adds a thin, smooth layer of copper to surfaces by using chemicals instead of electricity*.* |
| **Engineering Questions (EQ's)** | The factory's request to change or clarify something in the design.  Usually, to reduce risks for the customer, or to reduce costs. |
| **ENIG (Electroless Nickel Immersion Gold)** | A surface finish made of nickel and gold. |
| **Fiducials** | Reference markings on the cores used to properly align the layers during the manufacturing process. |
| **Flying Probe Test** | A testing method where robotic probes test PCB connections without needing a custom fixture. |
| **Galvanic / Electrolytic Plating** | A process where copper ions are electrically attracted to a surface. |
| **Gerber File** | A document with the customer's PCB design. |
| **HASL (Hot Air Soldering)** | A surface finish that coats the PCB with a layer of solder and then blows hot air over the panel to remove excessive solder. |
| **Image** | Creating a pattern on a solder mask by exposing certain areas to light (usually UV light) or a laser. |
| **Immersion Tin** | A surface finish made of tin. |
| **IPC** | The global standard for electronics manufacturing, including PCB production. |
| **IPC-SM-840** | A standard for solder mask inks that makes sure the solder mask meets specific criteria for durability and reliability, particularly in demanding environments. |
| **IPC TYPE 6 (VI) Plugging** (Described in IPC6012 Standard) | A plugging standard that says that vias must be filled with a non-conductive material like resin or solder mask and then covered with another protective layer. |
| **Lamination** | The process of pressing cores together under heat and pressure to turn them into one strong PCB. |
| **Lay-Up** | When cores and sheets of prepreg are stacked as specified. |
| **Laser Direct Imaging (LDI)** | Method used to directly write the circuit pattern onto the photosensitive film. |
| **Legend** | The administrative information printed directly on the PCB. |
| **Microsections** | An inspection method that involves taking small cut-outs of a PCB to examine the details inside. |
| **Multilayer Board** | A PCB with three or more layers of copper. |
| **Non-Plated Hole** | Hole that doesn't carry electricity. |
| **Open Circuit** | When the circuit is broken, so electrons can’t move. |
| **Oxide Layer** | A coating that prepares the board's surface before lamination to improve bonding. |
| **Pads** | Where components will later be soldered ("glued") to. |
| **Panel** | Several PCBs that go through production together, instead of one by one. |
| **PCB Specification** | Our own rulebook for building high-quality, reliable PCBs, every time. |
| **Photoresist** | A photosensitive film that hasn't been polymerized, so it’s soft and easy to remove. |
| **Photosensitive Film** | A film that polymerizes("hardens") when exposed to light. Used to create precise patterns on the board. |
| **Pins** | Things used in holes on the cores and prepreg to make sure they stay aligned during the process. |
| **Plating** | Coating hole walls with copper. |
| **Plating Currents** | The electrical power used to control how thick the copper deposits become during the plating process. |
| **Plating Flight Bars** | Tools used in the electrolytic plating process to hold the PCB panels in place and connect them to a power source. |
| **Prepreg** | A base material made of epoxy and fiberglass that isn't cured yet. Used to glue different layers of a PCB together. |
| **Profiling** | The process of cutting the customer's panels from the production panels and outlining individual PCBs to make it easier for the customer to separate them later. |
| **Registration** | The process factories use to find ***fiducials*** inside the bonded layers to ensure they're still properly aligned, so holes can be drilled in the correct places on the newly laminated, copper-covered stack. |
| **Resin** | The material in the prepreg that melts when it’s heated. |
| **Routing** | A method where PCBs are cut out or partly cut out from the customer panels. Ideal for round or irregular shapes. |
| **"Short" Circuit** | When two tracks connect when they shouldn’t, electrons take a shortcut by jumping from one track to another. This can happen when ***track separation*** (the distance between the tracks) is ***out of specification*** (meaning it doesn't meet the required standards), making them accidentally conductive. |
| **Solder Mask** | An ink used on the surface of the PCB. It's usually green and stops unwanted electrical connections by insulating the spaces between the circuits. It also protects the copper from getting damaged or corroded. |
| **Solvent** | A liquid (usually) that dissolves another material. |
| **Surface Finish** | A coating applied to the exposed copper on the surface of the PCB. |
| **Tooling Hole** | Non-plated hole that keeps the panel in place during production. |
| **Tracks** | A conductor that is part of a ***circuit***. |
| **Uv-Light** | Part of the spectrum of natural light and used to polymerize photosensitive films. |
| **V-Score Profiling** | A method where straight score lines are cut from both sides using rotating blades, but the PCBs are still attached to the panel. Ideal for straight cuts. |
| **Via Filling or Plugging Vias** | Filling holes with resin or solder mask ink. |
| **Via Hole** | Hole that usually go all the way through the board to electrically connect all the layers. |
| **Welding** | A method used on the edges of the layers to keep them aligned and hold them in place during the process until fully bonded together. |