Glossary of Terms

A

Activating: A treatment that renders nonconductive material receptive to electroless deposition.

Active Components: Semiconductor devices, such as transistors and diodes that can change its basic characteristics in a powered electrical circuit, such as amplifiers and rectifiers.

Additive Process: A process for obtaining conductive patterns by the selective deposition of conductive material on clad or unclad base material.

Analog Circuit: An electrical circuit that provides a continuous quantitative output as a response from its input.

Annular Ring: The width of the conductor pad surrounding a drilled hole.

Aperture Information: This is a text file describing the size and shape of each element on the board. These are also known as the D-code list. These lists are not necessary if your files are saved as Extended Gerber with embedded Apertures (RS274X)

Array: A group of elements or circuits (or circuit boards) arranged in rows and columns on a base material.

Artwork: Printed circuit design. An accurately scaled configuration used to produce the artwork master or production master.

Artwork Master: The photographic film or glass plate that embodies the image of the PCB pattern, usually on a 1:1 scale.

Aspect Ratio: The ratio of the board thickness to the smallest-hole diameter of the printed circuit board.

Assembly: A number of parts, subassemblies, or any combination thereof joined together.

Assembly File: A drawing describing the locations of components on a PCB.

Automated Optical Inspection (AOI): Visual inspection of the circuit board using a machine scanner to assess workmanship quality.

Automated Test Equipment (ATE): Equipment that automatically tests and analyzes functional parameters to evaluate performance of the tested electronic devices.

B

BBT: Bare Board Test.

B-Stage Material: Sheet material impregnated with a resin cure to an intermediate stage (B stage resin) Prepreg is the preferred term.

B-Stage Resin: A thermosetting resin that is in an intermediate state of cure.

Backplanes & Panels: Interconnection panels onto which printed circuits, other panels, or integrated circuit packages can be plugged or mounted. Typical thickness is 0.125" – 0.300".

Backup Material: A layer composed of phenolic, paper composite, or aluminum foil-clad fiber composite used during fabrication to prevent Burrs and to protect the drill table.

Ball Grid Array (BGA): A SMD package in which solder ball interconnects cover the bottom surface of the package.

Bare Board: An unpopulated PCB.

Barrel: The cylinder formed by plating through a drilled hole.

Base Copper: The thin copper foil portion of a copper-clad laminate for PCB's. It can be present on one or both sides of the board.

Base Copper Weight: see Copper Foil

Base Laminate: The dielectric material upon which the conductive pattern may be formed. The base material may be rigid or flexible.

Base Material: The insulating material upon which a conductive pattern may be formed. It may be rigid or flexible or both. It may be a dielectric or insulated metal sheet.

Bed-of-nails Fixture: A test fixture consisting of a frame and a holder containing a field of spring-loaded pins that make electrical contact with a planar test object (i.e. a PCB).

Bevel: An angled edge of a printed board.

Bleeding: A condition in which a plated hole discharges process materials of solutions from voids and crevices.

Blind Via: A via hole from an external layer to an internal layer. It is copper plated to enable it to conduct current, but it does not penetrate the board from top to bottom.

Blister: A localized swelling and separation between any of the layers of a laminated base material, or between base material and conductive foil. It is a form of delamination.

Blow Hole: A solder joint void caused by outgassing of process solutions during thermal cycling.

Board Thickness: Standard base thickness is 1/16 which is also called out as .062 or .059. Other standard sizes are 0.031, 0.093, 0.125 and 0.256. Typical tolerance is 10% of the given thickness. We can generally make any other thickness.

Book: A specified number of stacks of Prepreg plies which are assembled for Curing in a lamination press.

Bond Strength: The force per unit area required to separate two adjacent layers of a board by a force perpendicular to the board surface.

Bow and Twist: The deviation from flatness of a board characterized by a roughly cylindrical or spherical curvature such that if the board is rectangular its four corners are in the same plane.

Breakdown Voltage: The voltage at which an insulator or dielectric ruptures, or at which ionization and conduction take place in a gas or vapor.

Bridging, Electrical: The formation of a conductive path between two insulated conductors such as adjacent traces on a circuit board.

Built-In Self Test: An electrical testing method that allows the tested devices to test itself with specific added-on hardware.

Burried Via: A via hole between internal layers that electrically conducts (by a copper plating process) a current from layer to layer, it does not extend to the surface of the printed board.

Burr: A ridge left on the outside copper surface after drilling.

C

CAD: See Computer Aided Design.

CAM: See Computer Aided Manufacturing.

CAM Files: The files used for manufacturing PCB including Gerber file, NC Drill file and Assembly Drawings.

CEM: A punchable material (paper) used in single-sided boards but not suited for plated through-holes. CEM stands for composite epoxy material.

C-Stage: The condition of a resin polymer when it is solid state with high molecular weight.

Capacitance: The property of a system of conductors and dielectrics that permits storage of electricity when potential difference exists between conductors.

Centre-to-Centre Spacing: The nominal distance between the centres of adjacent features or traces on any layer of a printed circuit board. Also known as “pitch”.

Ceramic Ball Grid Array (CBGA): A ball grid array package with a ceramic substrate.

Chamfer: A broken corner to eliminate an otherwise sharp edge.

Characteristic Impedance: A compound measurement of the resistance, inductance, conductance and capacitance of a transmission line expressed in ohms. In printing wiring its value depends on the width and the thickness of the conductor, the distance from the conductor to ground plane(s), and the dielectric constant of the insulating media.

Chase: The aluminum frame used in screening inks onto the board surface.

Check Plots: A scaled paper or mylar image of the database after placement and routing have been completed.

Chip-on-Board (COB): A configuration in which a chip is directly attached to a printed circuit board or substrate by solder or conductive adhesives.

Chip: The individual circuit or component of a silicon wafer, the leadless form of an electronic component.

Circuit: The interconnection of a number of devices in one or more closed paths to perform a desired electrical or electronic function.

Circuit Board (PCB/ECB): The general term for a printed or etched circuit board. It includes single, double, or multiple layer boards, both rigid and flexible.

Circuitry Layer: A layer of a printed board containing conductors, including ground and voltage planes.

Clad or Cladding: A relatively thin layer or sheet of metal foil that is bonded to a laminate core to form the base material for printed circuit boards.

Cleanroom: A room in which the concentration of airborne particles is controlled to specified limits.

Clearances: A clearance (or isolation) is a term to describe the space from power/ground layer copper to through hole. To prevent shorting, ground and power layer clearances need to be .023"larger then the finish hole size for the inner layers. This allows for registration, drilling, and plating tolerances.

Coating: A thin layer of material, conductive, magnetic or dielectric, deposited on a substance surface.

Coefficient of Thermal Expansion (CTE): The ratio of dimensional change of an object to the original dimension when temperature changes, expressed in %/ºC or ppm/ºC.

Component: An electronic device, typically a resistor, capacitor, inductor, or integrated circuit (IC), that is mounted to the circuit board and performs a specific electrical function.

Component Hole: A hole used for the attachment and electrical connection of a component termination, such as a pin or wire to the circuit board.

Component Side: The Side of a PCB on which most of components are mounted. Also called the "top side".

Computer-Aided Design (CAD): A software program with algorithms for drafting and modeling, providing a graphical representation of a printed board's conductor layout and signal routes.

Computer-Aided Manufacturing (CAM): The use of computers to analyze and transfer an electronic design (CAD) to the manufacturing floor.

Computer-Integrated Manufacturing (CIM): Software that takes assembly data from a CAD or CAM package and, using a pre-defined factory modeling system, outputs routing of components to machine programming points and assembly and inspection documentation.

Conductor: A thin layer conductive area on a PCB surface or internal layer usually composed of lands (to which component leads are connected) and paths (traces).

Conductor Base Width: The conductor width at the plane of the surface of the base material. Also see Conductor Width.

Conductor-to-Hole Spacing: The distance between the edge of a conductor and the edge of hole.

Conductor Spacing: The distance between adjacent edges (not centerline to centerline) of isolated conductive patterns in a conductor layer.

Conductor Thickness: The thickness of the conductor including all metallic coatings.

Conductor Width: The observable width of the pertinent conductor at any point chosen at random on the printed circuit board.

Conformal Coating: An insulating protective coating that conforms to the configuration of the object coated and is applied on the completed board assembly.

Connector Area: The portion of the circuit board that is used for providing electrical connections.

Contaminant: An impurity or foreign substance whose presence on printed wiring assemblies could electrolytically, chemically, or galvanically corrode the system.

Continuity: An uninterrupted flow of electrical current in a circuit.

Controlled Impedance: See Characteristic Impedance

Coordinate Tolerancing: A method of tolerancing hole locations in which the tolerance is applied directly to linear and angular dimensions, usually forming a rectangular area of allowable variation. Also see, Positional Limitation Tolerancing and True Position Tolerance.

Copper Foil (Base Copper Weight): Coated copper layer on the board. It can be either characterized by weight or thickness of the coated copper layer. For instance, 0.5, 1 and 2 ounces per square foot are equivalent to 0.0007, 0.0014 and 0.0028 inch thick copper layers.

Copper (finished copper): This is how much copper your board will have on its surface. It is the copper foil thickness, plus plated copper, minus surface preparation. It is given in oz/per sq foot. 1 oz=a minimum of .0012-.0014” thickness

Copper Invar Copper: A multilayer metal alloy of a specific proportion, laminated together without the use of an insulating adhesive, thereby retaining a thermal and electrical conductive property.

Core Thickness: The thickness of the laminate base without copper.

Corrosive Flux: A flux that contains corrosive chemicals such as halides, amines, inorganic or organic acids that can cause oxidation of copper or tin conductors.

Cosmetic Defect: A defect such as a slight change in its usual colour that doesn’t affect a board’s functionality.

Cover Lay, Cover Layer, Cover Coat: Outer layer(s) of insulation material applied over the conductive pattern on the surface of a printed circuit board.

Crazing: A condition existing in the base material in the form of connected white spots or "crosses" on or below the surface of the base material, reflecting the separation of fibers in the glass cloth and resin material.

CTE: Coefficient of thermal expansion. The measure of the amount a material changes in any axis per degree of temperature change.

Curing: The irreversible process of polymerizing a thermosetting epoxy in a temperature-time profile.

Curing Time: The time needed to complete curing of resin at a certain temperature.

Current Carrying Capacity: The maximum current which can be carried continuously, under specified conditions, by a conductor without causing degradation of electrical or mechanical properties of the printed circuit board.

Cut lines: The cut line is going to be used to program the router path and it represents the board outside edge.

D

DFSM: Dry Film Solder Mask. Coating material (dry-film resist) applied to the printed circuit board via a lamination process to protect the board from solder or plating.

Date Code: This will have the year and week of manufacture of the board. It can be etched on the board or part of the silkscreen.

Datum Reference: A defined point, line, or plane used to locate the pattern or layer for manufacturing, inspection, or for both purposes.

Deburring: Process of removing burrs after drilling.

Defect: Any nonconformance to specified requirements by a unit or product.

Definition: The accuracy of reproduction of pattern edges, especially in a printed circuit relative to the original mater pattern.

Delamination: A separation between any of the layers of the base of laminate or between the laminate and the metal cladding originating from or extending to the edges of a hole or edge of board.

Design Rule: Guidelines that determine automatic conductor routing behaviour with respect to specified design parameters.

Design Rule Checking: The use of a computer program to perform continuity verification of all conductor routing in accordance with appropriate design rules.

Desmear: The removal of friction-melted resin and drilling debris from a hole wall.

Develop: An imaging operation in which unpolymerized (unexposed) photoresist is dissolved or washed away to produce a copper board with a photoresist pattern for etching or plating.

Dewetting: A condition which occurs when molton solder has coated a surface and then recedes, leaving irregular shape mounds of solder separated by areas covered with thin solder film; base material is not exposed.

Die: Integrated circuit chip as diced or cut from a finished wafer.

Die Bonder: The placement machine bonding IC chips onto a chip-on-board substrate.

Die Bonding: The attachment of an IC chip to a substrate.

Dielectric: An insulating medium between conductors.

Dielectric Constant: Is the ratio of permittivity of the material to that of a vacuum (referred to as a relative permittivity).

Differential Impedance: Refers to the impedance of a pair of conductors when driven in a differential mode, that is, when the conductors are driven by signals that have opposite polarity edges.

Digitizing: The converting of feature locations on a flat plane to a digital representation in X-Y coordinates.

Dimensional Stability: A measure of the dimensional change of a material that is caused by factors such as temperature changes, humidity changes, chemical treatment, and stress exposure.

DIP: Dual in-line package with two rows of leads from the base in standard spacing between the leads and row.

Double-Sided Assembly: PCB assembly with components on both sides of the substrate.

Double-Sided Board: A circuit board with conductive patterns on both sides.

Drawing or Print: This usually includes a drawing of the board outline with symbols marking individual drill sizes and corresponding board locations, dimensions, and any other pertinent manufacturing information unique to the board (copper weight, board thickness, stack-up specifics, etc.)

Drill file (Excellon Drill File): It will have x and y coordinates with tool sizes viewable in any text editor. It is the file that governs your finished hole sizes.

Drilling: The act of forming holes (vias) in a substrate by mechanical or laser means.

Drills, Circuit Board: Solid carbide cutting tools with four facet points and two helical flutes designed specifically for the fast removal of chips in extremely abrasive materials.

Dry - Film Resists: Coated photosensitive film on the copper foil of PCB using photographic methods. They are resistant to electroplating and etching processes in the manufacturing process of PCB.

Dry Film Solder Mask: A solder coating material applied to the PCB, through a lamination process to protect the board from solder or plating.

DRC: Design rule check.

E

Edge Bevel: A bevel operation performed on edge connectors to improve their wear and ease of installation.

Edge-Board Connector: A connector designed specifically for making removable and reliable interconnection between the edge board contacts on the edge of a printed board and external wiring.

Edge Connector: A connector on the circuit-board edge in the form of gold plated pads or lines of coated holes used to connect other circuit board or electronic device.

Edge Clearance: The smallest distance from any conductors or components to the edge of the PCB.

Edge Dip Solderability Test: A solderability test performed by taking a specially-prepared specimen, fluxing it with a nonactivated rosin flux, and then immersing it into a pot of molten solder at a pre-determined rate of immersion for a pre-determined dwell time, and then withdrawing it at a pre-determined rate.

Electrical Test (1 sided / 2/sided): Testing is primarily to test for opens and shorts.

Electroless deposition: The chemical coating of a conductive material onto a base material surface by reduction of metal ions in a chemical solution without using electrodes compared to electroplating.

Electroless Copper: A thin layer of copper deposited on the plastic or metallic surface of a PCB from an autocatalytic plating solution (without the application of electrical current).

Electroplating: The electrochemical deposition of reduced metal ions from an electrolytic solution onto the cathode by applying a DC current through the electrolytic solution between two electrodes, cathode and anode, respectively.

Entry Material: A thin layer of material composed of phenolic, aluminum foil, or paper that is placed on top to the panel prior to drilling, to improve drill accuracy and prevent burrs and dents.

Epoxy: A family of thermosetting resins. Epoxies form a chemical bond to many metal surfaces.

Epoxy Smear: Epoxy resin that has been deposited on edges of copper in holes during drilling either as uniform coating or in scattered patches. It is undesirable because it can electrically isolate the conductive layers from the plated-through-hole interconnections.

ESR: Electro-statically applied Solder Resist.

Etch: Chemical removal of metal (copper) to achieve a desired circuit pattern.

Etch Factor: The ratio of the depth of etch (conductor thickness) to the amount of lateral etch (undercut).

Etchback: The controlled removal of all components of the base material by a chemical process acting on the sidewalls of plated-through holes to expose additional internal conductor areas.

Etching: The process of removing unwanted metallic substances via chemical means.

Even Mode Impedance: The impedance of a single line in a coupled-line pair when a common-mode signal drives both conductors within the pair

F

Fiducial: Etched features or drilled hole used for optical alignment during assembly operations.

Film Artwork: A positive or negative piece of film containing a circuit, soldermask, or nomenclature pattern.

Fine Pitch: Fine pitch is more commonly referred to surface-mount components with a lead pitch of 25 mils or less.

Finger: A gold-plated terminal of a card-edge connector. Also see Gold Finger.

Files Gerber: Industry standard format for files used to generate artwork necessary for circuit board imaging.

First Article: A sample part or assembly manufactured prior to the start of production for the purpose of ensuring that the manufacturer is capable of producing a product that will meet specified requirements.

Fixture: A device that enables interfacing a printed circuit board with a spring-contact probe test pattern.

Flat: A standard size sheet of laminate material which is processed into one or more circuit boards.

Flux: The material used to remove oxides from metal surfaces and enable wetting of the metal with solder.

Flying Probe: A testing device that uses multiple moving pins to make contact with two spots on the electrical circuit and send a signal between them, a procedure that determines whether faults exist.

FR-1: A paper material with a phenolic resin binder. FR-1 has a Tg of about 130°C.

FR-2: A paper material with phenolic resin binder similar to FR-1 but with a Tg of about 150°C.

FR-3: A paper material that similar to FR-2 except that an epoxy resin is used instead of phenolic resin as a binder.

FR-4: Flame Retardent laminate made from woven glass fiber material impregnated with epoxy resin.

Functional Test: The electrical testing of an assembled electronic device with simulated function generated by the test hardware and software.

Fused Coating: A metallic coating (usually tin or solder alloy) which has been melted and solidified, forming a metallurgical bond to the base material.

G

G10: A laminate consisting of woven epoxy-glass cloth impregnated with epoxy resin under pressure and heat. G10 lacks the anti-flammability properties of FR-4. Used mainly for thin circuits such as in watches.

Gerber File: Data file used to control a photoplotter.

Glass Transition Temperature: The temperature at which an amorphous polymer (or the amorphous regions in a partially crystalline polymer) changes from a hard and relatively brittle condition to a viscous or rubbery condition. When this transition occurs, many physical properties undergo significant changes. Some of those properties are hardness, brittleness, coefficient of thermal expansion, and specific heat.

Golden Board: See Known Good Board.

Ground Plane: A conductor layer, or portion of a conductor layer, used as a common reference point for circuit returns, shielding, or heat sinking.

GI: The woven glass fiber laminate impregnated with polyimide resin.

Gold Finger: The gold-plated terminal of a card-edge connector. Also see Finger. Approximately 150-200 micro-inches minimum of nickel under a minimum 30micro-inches of gold on top.

Grid: An orthogonal network of two sets of parallel, equidistant lines used for locating points on a printed circuit board.

H

HASL: A method of coating exposed copper with solder by inserting a panel into a bath of molten solder, then passing the panel rapidly past jets of hot air.

HDI (High Density Interconnect): Ultra fine-geometry multi-layer PCB constructed with conductive microvia connections. These boards also usually include buried and/or blind vias and are made by sequential lamination.

Haloing: Mechanically-induced fracturing delamination on or below the surface of the base material; it is usually exhibited by a light area around holes, or other machines areas, or both.

Heat Sinks: Devices used to absorb or transfer heat away from heat-generating parts/components on a PCB.

Hermetic: Airtight sealing of an object.

Hole Breakout: A condition in which a hole is partially surrounded by the land.

Hole Density: The number of holes per unit area on a PCB.

Hole Pattern: The arrangement of all holes in a printed board with respect to a reference point.

Hole Void: A void in the metallic deposit of a plated-through hole exposing the base material.

I

In-Circuit Test: Electrical test of individual component or part of the circuit in a PCB assembly instead of testing the whole circuit.

Image: That portion on artwork masters, working tools, silkscreens or photomasks that would be considered the photographic image. Also would include images created with photo-resists or silk-screening techniques. Generally, “one image” refers to a single circuit board image, thus there may be several images per array.

Imaging: The process by which panelization data are transferred to the photoplotter, which in turn uses light to transfer an image circuitry pattern onto the panel.

Impedance: The total passive opposition offered to the flow of electric current. This term is generally used to describe high-frequency circuit boards.

Ink: Common term for screen resist.

Inkjetting: The dispersal of well-defined ink “dots” onto a PCB. Inkjet equipment uses heat to liquefy a solid ink pellet and change the ink into a liquid, which is then dropped via a nozzle onto the printed surface, where it quickly dries.

Innerlayers: The internal layers of laminate and metal foil within a multilayer board.

Inspection Overlay: A positive or negative transparency made from the production master and used as an inspection aid.

Insulation Resistance: The electrical resistance of an insulating material that is determined under specific conditions between any pair of contacts, conductors, or grounding devices in various combinations.

Interstitial Via Hole: An embedded through-hole with connection of two or more conductor layers in a multilayer PCB.

J

Jumper Wire: An electrical connection formed by wire between two points on a printed board adder after the intended conductive pattern is formed.

DYNAMIC & PROTO CIRCUITS INC.

K

Kerf: A widening of the rout path as may be called out on the blueprint. Allows extra space for hardware to be attached to the board.

Keying Slot: A slot in a printed circuit board that polarizes it, thereby permitting it to be plugged into its mating receptacle with pins properly aligned, but preventing it from being reversed or plugged into any other receptacle.

Known Good Board (KGB): A board or assembly that is verified to be free of defects. Also known as a Golden Board.

L

Laminate: The plastic material usually reinforced by glass or paper that supports the copper cladding from which circuit traces are created.

Laminate Thickness: Thickness of the metal-clad base material, single- or double-sided, prior to any subsequent processing.

Laminate Void: An absence of epoxy resin in any cross-sectional area that should normally contain epoxy resin.

Laminating Presses, Multilayer: Equipment that applies both pressure and heat to laminate and prepreg to make multilayer boards.

Lamination: The process manufacturing a laminate or PCB using pressure and heat.

Land: The portion of the conductive pattern on printed circuits designated for the mounting or attachment of components. Also called a pad.

Landless Hole: A plated-through hole without land(s). Also referred to as padless plated holes.

Layer-to-Layer Spacing: The thickness of dielectric material between adjacent layers or conductive circuitry in a multilayer printed circuit board.

Layers Sequence: Indicates a layer sequence so that we are able to build your order with the correct stack up.

Layup: The process in which treated prepregs and copper foils are prior to lamination.

Leakage Current: A small amount of current that flows across a dielectric area between two adjacent conductors.

Legend: A format of printed letters or symbols on the PCB, such as part numbers and product number or logos.

Line: See Conductor

Liquid Photoimaginable Soldermask (LPI): Liquid Photo-Imageable solder mask that uses photographic imaging to control a thinner mask deposition than the dry film solder mask.

Lot: A quantity of circuit boards that share a common design.

M

Major Defect: A defect that is likely to result in failure of a unit or product by materially reducing its usability for its intended purpose.

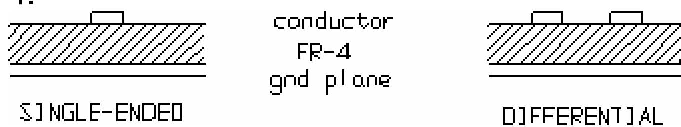
Mask: A material applied to enable selective etching, plating, or the application of solder to a PCB. Also called soldermask or resist.

Measling: Discrete white spots or crosses below the surface of the base laminate that reflect a separation of fibers in the glass cloth at the weave intersection.

Metal Foil: The plane of conductive material of a printed board from which circuits are formed. Metal foil is generally copper and is provided in sheets and rolls.

Microsectioning: The preparation of a specimen of a material, or materials, that is used in metallographic examination. This usually consists of cutting out a cross-section followed by encapsulation, polishing, etching, and staining.

Microstrip – a specific transmission line on a PCB where the signal trace is on an outside surface of the PCB and is spaced above a ground plane by the dielectric material, such as FR-4.



Microvia: Usually defined as a conductive hole with a diameter of 0.005” or less that connects layers of a multilayer PCB. Often used to refer to any small geometry connecting holes created by laser drilling.

Mil: One-thousandth of an inch (0.001”).

Minimum Annular Ring: The minimum metal width, at the narrowest point, between the circumference of the hole and the outer circumference of the land. This measurement is made to the drilled hole on internal layers of multilayer printed circuit boards and to the edge of the plating on outside layers of multilayer boards and double-sided boards.

Minimum Conductor Width: The smallest width of any conductors, such as traces, on a PCB.

Minimum Conductor Space: The smallest distance between any two adjacent conductors, such as traces, in a PCB.

Minimum Electrical Spacing: The minimum allowable distance between adjacent conductors that is sufficient to prevent dielectric breakdown, corona, or both, between the semiconductors at any given voltage and altitude.

Minor Defect: A defect that is not likely to result in the failure of a unit or product or that does not reduce its usability for its intended purpose.

Mis-Registration: The lack of dimensional conformity between successively-produced features or patterns.

Multilayer PCB: Circuit boards consisting three or more layers of printed circuits separated by laminate layers and bonded together with internal and external interconnections.

N

Nail Heading: The flared condition of copper on the inner conductor layers of a multilayer board caused by hole drilling.

NC Drill: Numeric Control drill machine used to drill holes at exact locations of a PCB specified in NC Drill File.

Negative: An artwork master or production master in which the intended conductive pattern is transparent to light, and the areas to be free from conductive material are opaque.

Netlist: List of parts and their connection points which are connected in each net of a circuit.

Node: A pin or lead to which at least two components are connected through conductors.

Nomenclature: Identification symbols applied to the board by means of screen printing, inkjetting, or laser processes.

Nonfunctional Land: A land on internal or external layers, not connected to the conductive pattern on its layer.

NPTH: Non-plated trough-hole.

Number of Holes: This is the total number of holes in the board. It has an influence on the price if it increases drilling time. Smallest hole we can mechanically drill is .008

O

Odd Mode Impedance: The impedance of a single line in a coupled-line pair when a differential signal drives both conductors within the pair.

Open: A circuit interruption (such as a broken track) that results in an incomplete path for current flow.

Outerlayer: The top and bottom sides of any type of circuit board.

Outgassing: De-aeration or other gaseous emission from a printed circuit board when exposed to the soldering operation.

Overhang: Increase in printed circuit conductor width caused by plating build-up or by undercutting during etching.

Oxide: An oxide coating on copper layers to enhance the bond or peel strength of internal layers when laminating a multilayer board, by ensuring that resin will have extra surface to adhere to

P

Pad: The portion of a conductive pattern for connection and attachment of electronic components on the PCB. Also called Land.

Panel: A rectangular sheet of base material or metal-clad material of predetermined size that is used for the processing of printed boards, and when required, one or more test coupons.

Panel Plating: The electrolytic plating of the entire surface of a panel (including holes).

Pattern: The configuration of conductive and nonconductive materials on a panel or printed board. Also, the circuit configuration on related tools, drawings, and masters.

Pattern Plating: The selective plating of a conductive pattern.

Peel Strength: The force per unit width required to peed the conductor or foil from the base material.

Permittivity: Is the measure of the ability of a material to store electrical energy when exposed to an electrical field.

PCB: Printed Circuit Board. Also called Printed Wiring Board (PWB).

PCMCIA: Personal Computer Memory Card International Association.

PEC: Printed Electronic Component.

Photomask: A silver halide or diazo image on a transparent substrate that is used to either block or pass light.

Photoplotter: Device used to generate artwork photographically by plotting objects (as opposed to copying an entire image at once as with a camera) onto film for use in manufacturing printed circuit boards.

Photo Print: The process of forming a circuit pattern image by hardening a photosensitive polymeric material by passing light through a photographic film.

Photoresist: A light-sensitive material that is used to establish an image by exposure to light and chemical development.

Phototool: A transparent film that contains the circuit pattern, which is represented by a series of lines of dots at a high resolution.

Pick-and-Place: A manufacturing operation of assembly process in which components are selected and placed onto specific locations according to the assembly file of the circuit.

Pilot Order: First production order going through process.

Pinhole: A minute hole through a layer or pattern.

Pitch: The center-to-center spacing between conductors, such as pads and pins, on a PCB.

Plastic Leaded Chip Carrier (PLCC): A component package with J-leads.

PTH (Plated Through-Hole): A hole plating on its walls that makes an electrical connection between conductive layers, external layers, or both, of a printed board.

Platen: A flat plate of metal within the lamination press in between which stacks are placed during pressing.

Plating: Chemical or electromechanical deposition of metal on a pattern.

Plating, Electroless: A method of metal deposition employing a chemical reducing agent present in the processing solution. The process is further characterized by the catalytic mature of the surface which enables the metal to be plated to any thickness.

Plating Resist: Material deposited as a covering film on an area to prevent plating on this area.

Plating Void: The area of absence of a specific metal from a specific cross-sectional area.

Plotting: The mechanical coverting of X-Y positional information into a visual pattern such as artwork.

Polyimide Resins: High temperature thermoplastics used with glass to produce printed circuit laminates for multilayer and other circuit applications requiring high temperature performance.

Polymerize: To unite chemically two or more monomers or polymers to form a molecule with a higher molecular weight.

Positional Limitation Tolerancing: Defines a zone within which the axis or centre plane of a feature is permitted to vary from true (theoretically exact) position.

Preclean: Cleaning steps taken prior to an operation to ensure success of the operation.

Prepreg: Sheet material consisting of glass cloth impregnated with a synthetic resin, such as epoxy or polyimide, partially cured to the B stage.

Press-Fit Contact: An electrical contact which can be pressed into a hole in an insulator, printed board (with or without plated-through holes), or a metal plate.

Pressing: The process by which a combination of heat and pressure are applied to a book, thereby producing fully cured laminated sheets.

Printed Board: The general term for completely processed printed circuit or printed wiring configurations. It includes single, double-sided, and multilayer boards, both rigid and flexible.

Printed Circuit: A conductive pattern that comprises printed components, printed wiring, or a combination thereof, all formed in a predetermined design and intended to be attached to a common base. (In addition, this is a generic term used to describe a printed board produced by any of a number of techniques).

Printed Wiring Board: A part manufactured from rigid base material upon which completely processed printed wiring has been formed.

Production Master: A 1:1 scale pattern which is used to produce one or more printed boards (rigid or flexible) within the accuracy specified on the master drawing. (a) Single-Image Product Master – A production master used in the process of making a single printed circuit board. (b) Multiple-Image Production Master – A production master used in the process of making two or more printed circuit boards simultaneously.

Pulse Plating: A method of plating that uses pulses instead of a direct current.

R

Readme Files: A text file included in the zip file. It provides a reference for your order. If you have a specific need or important point it is generally put in the readme file.

Reflowing: The melting of an electro-deposit followed by solidification. The surface has the appearance and physical characteristics of being hot-dipped.

Reflow Soldering: Melting, joining and solidification of two coated metal layers by application of heat to the surface and predeposited solder paste.

Registration: The degree of conformity to the position of a pattern, or a portion thereof, a hole, or other, feature to its intended position on a product.

Residue: An undesirable substance remaining on a substrate after a process step.

Resin (Epoxy) Smear: Resin transferred from the base material onto the surface of the conductive pattern in the wall of a drilled hole.

Resin-Starved Area: A region in a printed circuit board that has an insufficient amount of resin to wet out the reinforcement completely, evidenced by low gloss, dry spots or exposed fibers.

Resist: Coating material used to mask or to protect selected areas of a pattern from the action of an etchant, solder, or plating. Also called soldermask or mask.

Resistivity: The ability of a material to resist the passage of electrical current through it.

Reverse Image: The resist pattern on a printed circuit board enabling the exposure of conductive areas for subsequent plating.

Rework: Reprocessing that makes articles conform to specifications.

Rigid-Flex: A PCB construction combining flexible circuits and rigid multilayers usually to provide a built-in connection or to make a three-dimensional form that includes components.

Robber: An exposed area generally attached to a rack used in electroplating, usually to provide a more uniform current density on plated parts. Thieves are intended to absorb the unevenly distributed current on parts, thereby assured that the parts will receive a uniform electroplated coating.

Rough Holes: Holes with a copper burr around either the entry or exit hole and that lack a smooth barrel.

Route (or Track): A layout or wiring of an electrical connection.

Routing options: They can be Tab routing, Tab routing with Perferation Holds and Vscore.

S

Schematic Diagram: A drawing which shows, by means of graphic symbols, the electrical connections, components, and functions of an electronic circuit.

Scoring: A technique in which grooves are machined on opposite sides of a panel to a depth that permits individual boards to be separated from the panel after component assembly.

Screen: A cloth material (usually polyester or stainless steel for circuit boards) coated with a pattern which determines the flow and location of coatings forced through its openings.

Screen Printing: A process for transferring an image to a surface by forcing suitable media through a stencil screen with a squeegee.

Selective Plate: A process for plating unique features with a different metal than the remaining features will have. Created by imaging, exposing and plating selected area then repeating the process for the remainder of the board.

Shadowing: A condition occurring during etchback in which the dielectric material, in contact with the foil, is incompletely removed although acceptable etchback may have been achieved elsewhere.

Silk Screen (Silk Legend): Epoxy-ink Legend printed on PCB. The most common colors used are white and yellow.

Single-sided Board: A printed board with conductive pattern on one side only.

Small Outline Integrated Circuit (SOIC): An integrated circuit with two parallel rows of pins in surface mount package.

Smallest Hole: This is the smallest hole contained in your drill file.

SMOBC: Solder mask over bare copper. A method of fabricating a printed circuit board which results in final metallization being copper with no protective metal. The non-coated areas are coated by solder resist, exposing only the component terminal areas. This eliminated tin lead under the pads.

SMD: Surface Mount Device.

SMT: Surface Mount Technology. Defines the entire body of processes and components that create printed circuit board assemblies with leadless components.

Solder: An alloy that melts at relatively low temperatures and is used to join or seal metals with higher melting points. A metal alloy with a melting temperature below 427°C (800°F).

Solder Bridging: Solder connecting, in most cases, misconnecting, two or more adjacent pads that come into contact to form a conductive path.

Solder Bumps: Round solder balls bonded to the pads of components used in face-down bonding techniques.

Solder Leveling: The process by which the board is exposed to hot oil or hot air to remove any excess solder from holes and lands.

Solder Mask: Used to protect the board and circuitry during the assembly and packaging operations.

Solder Mask or Solder resist: Coating to prevent solder to deposit on.

Solder Wick: A band of wire removes molten solder away from a solder joining or a solder bridge or just for desoldering.

Solderability Testing: The evaluation of a metal to determine its ability to be wetted by solder.

Squeegee: The tool used in silkscreening which forces the resist or ink through the mesh.

Starvation, Resin: A deficiency of resin in base material which is apparent after lamination by the presence of weave texture, low gloss, or dry spots.

Step-and-Repeat: A method by which successive exposures of a single image are made to produce a multiple image production master.

Stripline: A specific transmission line on a PCB where the signal trace is buried within the PCB and is spaced above and below a ground plane by the dielectric material, such as FR-4.

Stripping: The process by which imaging material (resist) is chemically removed from a panel during fabrication.

Substrate: A material on whose surface adhesive substance is spread for bonding or coating. Also, any material that provides a supporting surface for other materials used to support printed circuit patterns.

Subtractive Processing: The method of selectively removing copper from a board to form a circuit. In this case, "substractive" refers to the method of image transfer from a phototool or image file to the copper circuit.

T

TDR: Time Domain Reflectometer, a device which a board house can use for measuring characteristic impedance of a conductor on a printed board, thus insuring an accurate build for controlled impedance

Tab Routing: Rather than completing the route path around the board edge, "Tabs" are left so as to leave boards attached in pallets for ease of assembly.

Temperature Coefficient (TC): The ratio of a quantity change of an electrical parameter, such as resistance or capacitance, of an electronic component to the original value when temperature changes, expressed in %/ºC or ppm/ºC.

Test Coupon: A portion of a printed circuit board containing printed circuit coupons used to determine the acceptability of such boards.

Test Point: A specific point in a circuit board used for specific testing for functional adjustment or quality test in the circuit-based device.

TG: Glass transition temperature, the point at which rising temperatures causes the solid base laminate to start to exhibit soft, plastic-like symptoms. This is expressed in degrees Celsius (°C).

Thief: An extra cathode placed as to divert to itself some of the current from portions of the board which otherwise would receive too high a current density.

Tooling Holes: The general term for holes placed on a printed circuit board for registration purposes during the manufacturing process

Top Side: See Component Side

Trace: A common term for conductor.

Transmission line: Commonly used to denote a controlled ***impedance*** conductor path that has a defined ***velocity of propagation***. Geometry and dielectric materials determine many of the properties of a transmission line. Examples are coaxial cable, twinax or parallel paired cable, twisted pairs, parallel traces on a PCB.

Traveler: The list of instructions describing the board, including any specific processing requirements. Also called a shop traveler, routing sheet, job order, or production order.

Twist: A laminate defect in which deviation from planarity results in a twisted arc.

U

UL: Underwriter's Laboratories. An independent product safety testing and certification organization.

UV (Ultraviolet) Curing: Polymerizing hardening, or cross-linking a low molecular weight resinous material in a wet coating or ink, using ultraviolet light as an energy source.

V

V-Scoring: The edges are "scored" to allow breaking boards apart after assembly

Via: A plated-through hole used for interconnection of conductors on different sides or layers of a PCB.

Void: A void in the metallic deposit of a plated-through hole exposing the base material.

W

Wave Soldering: A manufacturing operation in which solder joints are soldered simultaneously using a wave of molten solder.

Wicking: Migration of copper salts into the glass fibers of the insulation material.

Z

Zero Defects Sampling: A statistical based attribute sampling plan (c=o) where a given sample of parts are inspected and any defects found are cause for rejection of the entire lot.

Zip File: Zipping a file compresses one or more files into a smaller archive. It takes up less hard drive space and less time to transfer across a network or the internet.