A

Annular Ring – That portion of conductive material completely surrounding a hole. The width of the annular ring is important design and manufacturing consideration.

Array – A group of elements or circuits arranged in rows and columns on a base material.

Artwork – An accurately scaled configuration of electronic data used to produce the artwork master or production master.

AS9100 - A standardized quality management system developed for the aviation, aerospace and defense industry suppliers that incorporates ISO-9001:2008 and industry requirements in an effort to provide improved quality and performance.

B

B-Stage – An intermediate stage in the reaction of a thermosetting resin in which the material softens when heated and swells, but does not entirely fuse or dissolve, when it is in contact with certain liquids.

Barrel – The cylinder formed by plating the walls of a drilled hole.

Bare Board – An unassembled (unpopulated) printed board.

Base Material – The insulating material used to form the conductive pattern. It may be rigid or flexible or both. It may be a dielectric or insulated metal sheet.

Base Material Thickness – The thickness of the base material excluding metal foil or material deposited on the surface.

Bed of Nails – 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.

Blind Via – A via extending only to one surface of a printed board. Does not extend from the top to the bottom layer.

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

Board Thickness – The overall thickness of the base material and all conductive material deposited thereon.

Book – A specified number of Pre-preg plies which are assembled along with inner-layer cores in preparation for curing in a lamination press.

Bow – The deviation from flatness of a board characterized by a roughly cylindrical or spherical curvature such that, if the product is rectangle, its four corners are in the same plane.

Border Area – The region of a base material that is external to that of the end product being fabricated within it.

Buried Via – A via hole drilled through inner-layers of a multilayer board that does not extend to the surface layers.

Burr – A ridge surrounding the hole left on the outside copper surface after drilling.

C

CAD – (Computer Aided Design) – A system where engineers create a design and see the proposed product in front of them on a graphics screen or in the form of a computer printout or plot. In electronics, the result would be a printed circuit layout.

CAM – (Computer Aided Manufacturing) – The interactive use of computers systems, programs, and procedures in various phases of a manufacturing process wherein, the decision-making activity rests with the human operator and a computer provides the data manipulation functions.

CAM Files – The data files used directly in the manufacture of printed wiring. The file types are: (1) Gerber files, which control a photo-plotter. (2) NC Drill file, which controls an NC Drill machine. (3) Fabrication drawings in Gerber, HPGL or any other electronic format. CAM files represent the final product of PCB design. These files are given to the board house which further refines and manipulates CAM in their processes, for example in step- and-repeat panelization.

Capacitance - The property of a system of conductors and dielectrics that allows the storage of electricity when a potential difference exists between the conductors.

Castellated Holes - A via may be at the edge of the board so that it is cut in half when the board is separated; this is known as a castellated hole and is used for a variety of reasons, including allowing one PCB to be soldered to another in a stack.

Center to Center Spacing – The nominal distance between the centers of adjacent features on any single layer of a printed board, e.g.; gold fingers and surface mounts.

Clad – A copper object on a printed circuit board. Specifying certain text items for a board to be "in clad," means that the text should be made of copper, not silkscreen.

Class- This standard recognizes that electrical and electronic products are subject to classifications by intended end-item use. IPC is the standard use for acceptability of printed boards. Three general end-product classes have been established to reflect differences in producibility, complexity, functional performance requirements and verification (inspection/test) frequency.

Class 1- Includes limited life products suitable for applications where the requirement is function of the completed product.

Class 2- Includes products where continued performance and extended life is required, and for which uninterrupted service is desired but not critical.

Class 3- Includes products where continued high performance or performance-on-demand is critical, product down time cannot be tolerated, and the product must function when required.

Clearance Hole – A hole in the conductive pattern that is larger than, and coaxial with a hole in the base material of a printed board.

CNC (Computer Numerical Control) – A system that utilizes a computer and software as the primary numerical control technique.

Component – Any of the basic parts used in building electronic equipment, such as a resister, capacitor, DIP or connector, etc.

Component Hole – A hole that is used for the attachment and/or electrical connection of component terminations, including pins and wires, to a printed board.

Component Side - The side of the circuit board on which most of the components will be located. Also called the "top side."

Conductive Pattern – The configuration pattern or design of the conductive material on a base material. This includes conductors, lands, vias, heat sinks and passive components when those are integral parts of the printed board manufacturing process.

Conductor Spacing – The observable distance between adjacent edges (not center to center spacing) of isolated patterns in a conductor layer.

Continuity – An uninterrupted path for the flow of electrical current in a circuit.

Connection – One leg of a net.

Connector – A plug or receptacle, which can be easily joined to or be separated from its mate. Multiple-contact connectors join two or more conductors with others in one mechanical assembly.

Controlled Impedance – The matching of substrate material properties with trace dimensions and locations in an effort to create specific electric impedance for a signal moving along a trace.

Controlled Dielectric – Specified thickness of the insulating layers between a signal and power or ground planes.

Copper Weight (Inner) – Copper thickness required on inner layer cores, typically 1 oz.

Copper Weight (Outer) – Number of ounces of copper per square foot on outer layers. Specify this as "finished" copper weight.

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

Counterbore – A cylindrical recess, machined around a hole to allow a screw head to sit flush with a

Countersink – A beveled recess, machined around a hole to allow a screw head to sit flush with a surface.

CTE – (Coefficient of Thermal Expansion) – The measure of the amount a material changes in any axis per degree of temperature change.

Curing – The act of applying heat and pressure to the laminate materials in order to produce a bond.

D

Database – A collection of interrelated data items stored together without unnecessary redundancy, to serve one or more applications.

Date Code – Marking of products to indicate their date of manufacture.

Datum – The theoretically-exact point, axis or plane that is the origin from which the location of geometric characteristics of features of a part are established.

Delamination – A separation between plies within a base material, between a base material and a conductive foil, or any other planner separation with a printed board.

Design Rule Checking – The use of a computer-aided program to perform continuity verification of all conductors routing in accordance with appropriate design rules.

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

Dewetting – A condition that results when molten solder has coated a surface and then receded. It leaves irregularly shaped mounds separated by areas of thin solder. The base material is not exposed.

Dielectric – A material with a high resistance to the flow current, and which is capable of being polarized by an electrical field.

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.

Double-Sided Board – A printed board with a conductive pattern on both sides.

Dry-Film Resists – Coating material specifically designed for use in the manufacture of printed circuit boards and chemically machined parts. They are resistant to various electroplating and etching processes.

Dry Film Solder Mask – A solder mask film applied to a printed board using photographic methods. This method can manage the higher resolution required for fine line design and surface mount.

E

Electro-deposition – The deposition of a conductive material from a plating solution by the application of electrical current.

Electroplating – The electro-deposition of a metal coating on a conductive object. The object to be plated is placed in an electrolyte and connected to one terminal of a direct current (DC) voltage source. The metal to be deposited is similarly immersed and connected to the other terminal.

Etchback – The controlled removal by a chemical process, to a specific depth, of nonmetallic materials from the sidewalls of holes in order to remove resin smear and to expose additional internal conductor surfaces.

Etching – The chemical, or chemical and electrolytic, removal of unwanted portions of conductive or resistive material.

F

Fab - Fabrication.

Fabrication Drawing – A drawing used to aid the construction of a printed circuit board. It shows all of the locations of the holes to be drilled, their sizes and tolerances, dimensions of the board edges, and notes on the materials and methods to be used. Called "fab Drawing" for short.

Fiducial Mark – A printed board feature (or features) that is created in the same process as the conductive pattern and that provides a common measurable point for component mounting with respect to a land pattern or land patterns.

Fine Pitch – Refers to chip packages with lead pitches below .050. The largest pitch in this class of parts is about .031. Lead pitches as small as .020 are used.

Finger – A gold-plated terminal of a card-edge connector. (Derived from its shape.)

First Article – A sample part or assembly typically 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.

Flying Probe – A type of bare board electrical test machine that uses probes on the ends of mechanical arms to locate and touch the pads on the board. The probes move quickly across the board verifying continuity of each net as well as resistance to adjacent nets.

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 105°C.

FR-3 – A paper material that is similar to FR-2 - except that an epoxy resin is used instead of phenolic resin as a binder. Used mainly in Europe.

FR-4 – The UL-designated rating for a laminate composed of glass and epoxy that meets a specific standard for flammability. FR-4 is the most common dielectric material used in the construction of PCBs.

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 photo-plotter. Named after the Gerber Scientific Co., who made the original vector photo-plotter.

Ground – A common reference point for electrical circuits returns, shielding or heat sinking.

Ground Plane – A conductor layer, or portion thereof that serves as a common reference for electrical circuit returns, shielding or heat sinking.

H

HASL – (Hot Air Solder Leveling) – 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.

Hole Breakout – A condition in which a hole is not completely surrounded by the land.

I

Imaging – The process of transferring electronic data to the photo-plotter, which in turn uses light to transfer a negative image circuitry pattern onto the panel or film.

Immersion Plating – The chemical deposition of a thin metallic coating over certain basis metals that is achieved by a partial displacement of the basis metal.

Impedance – The resistance to the flow of current, represented by an electrical network of combined resistance, capacitance and inductance reaction, in a conductor as seen by an AC source of varying time voltage. The unit of measure is ohms.

Inclusions – Foreign particles, metallic or nonmetallic, that may be entrapped in an insulating material, conductive layer, plating, base material, or solder connection.

Inner-Layers – The internal layers of laminate and metal foil within a multi-layer board.

IPC – (The Institute for Interconnecting and Packaging Electronic Circuits) – The final American authority on how to design and manufacture printed wiring.

K

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

L

Laminate – A product made by bonding together two or more layers of materials.

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.

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

Laser Photo-Plotter – A plotter that uses a laser, which simulates a vector photo-plotter by using software to create a raster image of the individual objects in a CAD database, then plots the image as

a series of lines of dots at a very fine resolution. A laser photo-plotter is capable of more accurate and consistent plots than a vector plotter.

Lead – A terminal on a component.

Legend – A format of lettering or symbols on the printed circuit board: e.g. part number, serial number, component locations, and patterns.

LPI – (Liquid Photo-Imageable Solder Mask) – An ink that is developed off using photographic imaging techniques to control deposition. It is the most accurate method of mask application and results in a thinner mask than dry film solder mask. It is often preferred for dense SMT. Application can be spray, curtain coat or squeegee.

M

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

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

Microvia – Usually defined as a conductive hole with a diameter of 0.005" or less that connects layers of a multi-layer PCB. Often used to refer to any small geometry connection holes created by laser drilling.

Mil – One thousandth of an inch.

Mounting Hole – A hole that is used for the mechanical support of a printed board or for the mechanical attachment of components to a printed board.

Multi-Layer Board – Printed boards consisting of a number (three or more) of separate conducting circuit planes separated by insulating materials and bonded together into relatively thin homogeneous constructions with internal and external connections to each level of the circuitry as needed.

N

NC Drill – (Numeric Control drill machine) – A machine used to drill the holes in a printed board at exact locations, which are specified in a data file.

Negative – A reverse-image copy of a positive, useful for checking revisions of a PCB and is often used for representing inner layer planes. When a negative image is used for an inner-layer it would typically have clearances (solid circles) and thermals (segmented donuts) that either isolate holes from the plane or make thermally relieved connections respectively.

Net – A collection of terminals all of which are, or must be, connected electrically. Also known as signal.

Netlist – List of names of symbols or parts and their connection points which are logically connected in each net of a circuit. A netlist can be captured from properly prepared schematic-drawing files of an electrical CAE application.

Nomenclature – Identification symbols applied to the board.

O

Open – Open circuit. An unwanted break in the continuity of an electrical circuit which prevents current from flowing.

Outer-Layer – The top and bottom sides of any type of circuit board.

P

Pad-The portion of the conductive pattern on printed circuits designated for the mounting or attachment of components.

Panel – Material (most commonly an epoxy- copper laminate know as FR-4) sized for fabrication of printed circuit boards.

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

Pattern Plating – The selective plating of a conductive pattern.

PCB Array - Boards supplied in pallet form, sometimes called "panelized", "stepped", "palletized" and "rout and retain”.

PCB Prototype - A printed circuit board fabricated especially for use in testing. In some cases, a printed circuit board manufactured rapidly for a specific application.

Photo Plotter – Device used to generate photographically by plotting objects onto film for use in manufacturing printed wiring.

Photo Resist – A material that is sensitive to portions of the light spectrum and that, when properly exposed can mask portions of a base metal with a high degree of integrity.

Plating – The chemical or electrochemical deposited metal on a surface.

Plated-Through Hole – A hole in a PWB with metal plating added after it is drilled. Its purpose is to serve either as a contact point for a through-hole component or as a via.

Positive – A developed image of photo-plotted file, where the areas selectively exposed by the photo plotter appear black and unexposed areas are clear. For outer-layers, color will indicate copper. Positive inner-layers will have clear areas to indicate copper.

Prepreg – A sheet of material that has been impregnated with a resin cured to an intermediate stage. I.e. B-stage resin.

Probe Test – A spring-loaded metal device used to make electrical contact between test equipment and the unit under test.

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

Q

Quick Turn – Fast turnaround PCB manufacturing.

R

Reflow – The melting of electrodeposited tin/lead followed by solidification. The surface has the appearance and physical characteristics of being hot-dipped.

Reference Designator – The name of components on a printed circuit by convention beginning with one or two letters followed by a numeric value. The letter designates the class of component; eg. "Q" is commonly used as a prefix for transistors. Reference designators appear as usually white or yellow epoxy ink (the "silkscreen") on a circuit board. They are placed close to their respective components but not underneath them. So that they are visible on the assembled board.

Reference Dimension – A dimension without a tolerance that is used only for informational purposes that does not govern inspection or other manufacturing operations.

Resist – A coating material that is used to mask or protect select areas of a pattern during manufacturing or testing from the action of an etchant, plating, solder, etc.

RoHS – The RoHS Directive and similar legislative acts in other parts of the world are fostering changes in the fabrication and assembly of printed circuit boards.

Rout – A layout or wiring of a connection. The action of creating such a wiring. The term is also used for the actual milling of a PCB.

S

Schematic – A diagram which shows, by means of graphic symbols, the electrical connections and functions of a specific circuit arrangement.

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

Short – Short circuit. An abnormal connection of relatively low resistance between two points of a circuit. The result is excess (often damaging) current between these points. Such a connection is considered to have occurred in a printed wiring CAD database or artwork anytime conductors from

different nets either touch or come closer than the minimum spacing allowed for the design rules being used

Silkscreen –The decals and reference designators in epoxy ink on a printed wiring board so called because of the method of application – the ink is "squeegeed" through a silk screen, the same technique used in the printed of T-shirts. Minimum line width at ACI for silkscreen is .008. Also called "silkscreen legend".

SMOBC –(Solder Mask Over Bare Copper) -A method of fabricating a printed circuit board that 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 eliminates tin lead under the mask.

SMD – Surface mount device.

SMT – Surface Mount Technology.

Solder – An alloy that melts at relatively low temperatures and is used to join or seal metals with higher melting points.

Solder Coat – A layer of solder that is applied directly from a molten solder bath to a conductive pattern.

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

Solder Paste Stencils – Stencils ensure the right amount of solder paste is applied to achieve optimal electrical connections.

Solder Mask – A technique wherein everything on a circuit board is coated with a mask except 1) the contacts to be soldered, 2) the gold-plated terminals of any card-edge connectors and 3) fiducial marks.

Step–and–Repeat – The successive exposure of a single image on order to produce a multiple-image production master.

Stuff – Components are attached and soldered to a printed wiring board. Often done by an assembly house.

Sub–Panel – A group of printed circuits arrayed in a panel and handled by both the board house and the assembly house as though it were a single printed wiring board. The sub-panel is usually prepared at the board house by routing most of the material separating individual modules leaving small tabs.

Surface Mount – Surface mount technology. Components are soldered to the board without using holes. The result is higher component density, allowing smaller PWBs. Abbreviated SMT.

T

Tented Via – A via with dry film solder mask completely covering both its pad and its plated-thru hole. This completely insulates the via from foreign objects, thus protecting against accidental shorts, but it

also renders the via unusable as a test point. Sometimes vias are tented on the topside of the board and left uncovered on the bottom side to permit probing from that side only with a text fixture.

Tenting – The covering of holes in a printed board and the surrounding conductive pattern with a dry film resist.

Terminal – A point of connection for two or more conductors in an electrical circuit; one of the conductors is usually an electrical contact or lead of a component.

Test Board – A printed board that is deemed to be suitable for determining the acceptability of a group of boards that were. Or will be, produced with the same fabrication process.

Test Fixture – A device that interfaces between test equipment and the unit under test.

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

Through-Hole – Having pins designed to be inserted into holes and soldered to pads on a printed board. Also spelled "thru-hole".

Trace – Segment of a conductor route or net.

Track – Trace.

U

UL – (Underwriter's Laboratories, Inc.) – A corporation supported by some underwriters for the purpose of establishing safety standards on types of equipment or components.

V

Via – Feed through. A plated-through hole in a PWB used to route a trace vertically in the board, that is, from one layer to another.

Void – The absence of any substances in a localized area. (Ex: missing plating in a hole)

W

Wave Soldering – Assembled printed boards are brought in contact with a continuously flowing and circulating mass of solder, typically in a bath to connect the leads of components to through hole pads and barrels.

Wicking – Migration of copper salts into the glass fibers of the insulating material found in the barrel of a plated hole.

X

X-Axis – The horizontal or left-to-right direction in a two-dimensional system of coordinates.

Y

Y-Axis – The vertical or bottom-to-top direction in a two-dimensional system of coordinates.

Z

Z-Axis – The axis perpendicular to the plane formed by the X and Y datum reference. This axis usually represents the thickness of the boards.