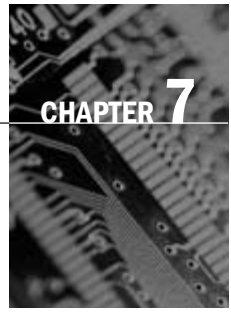


Terms and Definitions

Note that the terms capitalized within the definitions are also defined separately.

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

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

ANNULAR RING: That portion of conductive material completely surrounding a hole.

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

ARTWORK: 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: A ratio of the PCB thickness to the diameter of the smallest hole.

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

AUTOMATED OPTICAL INSPECTION (AOI): Visual inspection of the circuit board using a scanning technique to assess workmanship quality.

AUTOMATIC TEST EQUIPMENT (ATE): Equipment that automatically analyzes functional or static parameters in order to evaluate performance.

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.

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

BASE COPPER: The thin copper foil portion of a copper-clad laminate for PCBs. It can be present on one or both sides of the board.

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.

BASE MATERIAL THICKNESS: The thickness of the base material excluding metal foil or material deposited on the surface.

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 conductive surface hole that connects an outer layer with an innerlayer of a multilayer board without penetrating the entire board

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

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: 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.

B-STAGE MATERIAL: Sheet material impregnated with a resin cured to an intermediate stage (B-stage resin). Prepreg is the

popular term.

B-STAGE RESIN: A thermosetting resin that is in an intermediate state of cure.

BURIED VIA: A via hole that does not extend to the surface of a printed board.

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

CAD: See Computer-Aided Design.

CAM: See Computer-Aided Manufacturing.

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 in a solid state with high molecular weight. Being insoluble and infusion. [DAN: WHAT DO YOU MEAN BY THE SECOND SENTENCE?]

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

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

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

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 circuits.

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

COMPONENT: An electronics 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, including pin or wire to the circuit board.

COMPONENT SIDE: The side of the circuit board on which most of the components will be located. 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 CAM/CAD 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 conductive area on a PCB surface or internal layer usually composed of lands (to which component leads are connected) and paths (traces).

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

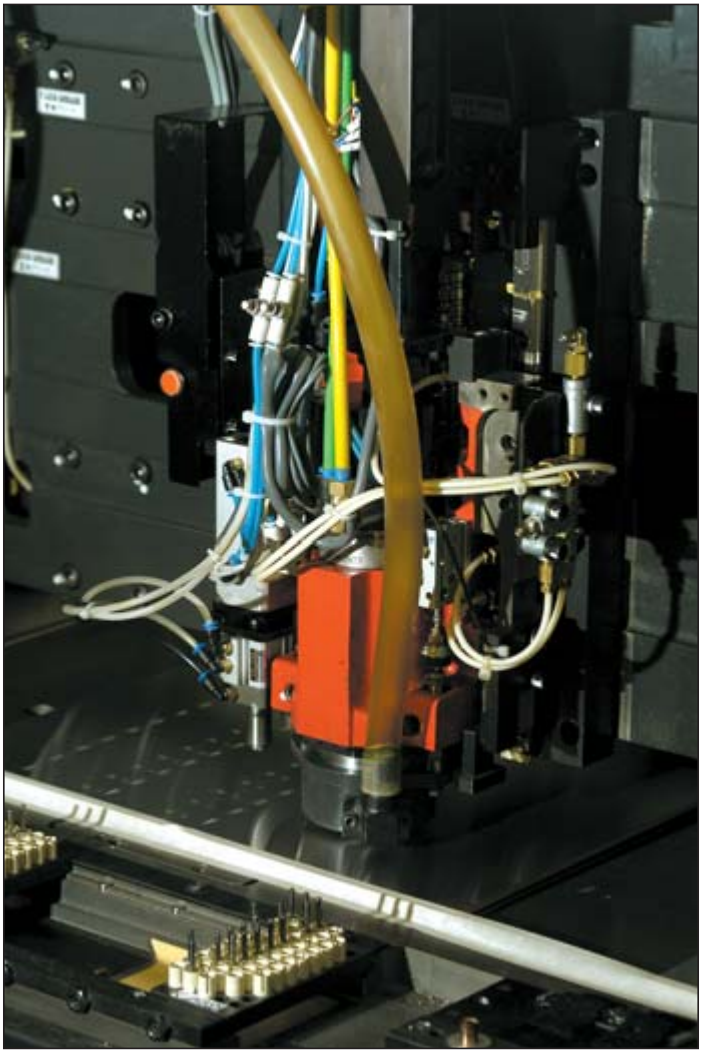
CONDUCTOR THICKNESS: The thickness of the conductor including all metallic coatings.

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.

CONTROLLED IMPEDANCE: The matching of substrate material properties with trace dimensions and locations to create specific electric impedance as seen by a signal on the trace.

CORE THICKNESS: The thickness of the laminate base without copper.



COURTESY PHOTOCIRCUITS

Closeup of drill spindle

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 to a material in order to produce a bond.

DEBURRING: Process of removing burrs after PCB drilling.

DEFECT: Any nonconformance to specified requirements by a

COURTESY PHOTOCIRCUITS

Electrolytic gold plating line

unit or product.

DEFINITION: The fidelity of reproduction of pattern edges, especially in a printed circuit relative to the original master 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 behavior 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.

DEWETTING: A condition that results when molten solder has coated a surface and then receded, leaving irregularly shaped mounds separated by areas covered with a thin solder film and with the base material not exposed.

DIELECTRIC: An insulating medium that occupies the region between two conductors.

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.

DOUBLE-SIDED BOARD: A printed board with a conductive pattern on both sides.

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

DRY-FILM RESISTS: Coating material specifically designed for use in the manufacture of printed circuit boards and chemically machined parts. They are suitable for all photomechanical operations and are resistant to various electroplating and etching processes.

DRY-FILM SOLDERMASK: Coating material (dry-film resist) applied to the printed circuit board via a lamination process to protect the board from solder or plating.

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).

ELECTRODEPOSTION: See Electroplating.

ELECTROPLATING: The electrodeposition of an adherent 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.

ENTRY MATERIAL: A thin layer of material composed of phenolic, aluminum foil, or paper that is placed on top of 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.

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 chemical, or chemical and electrolytic, removal of unwanted portions of conductive materials.

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

FLYING PROBE TEST: 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 130ºC.

FR-2: A paper material with phenolic resin binder similar to FR-1 but with a Tg of about 105ºC.

FR-3: A paper material that 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 fire-retardance. FR-4 is the most common dielectric material used in the construction of PCBs.

GERBER: A software format used by the photoplotter to describe the printed circuit board design.

GOLDEN BOARD: See Known Good Board.

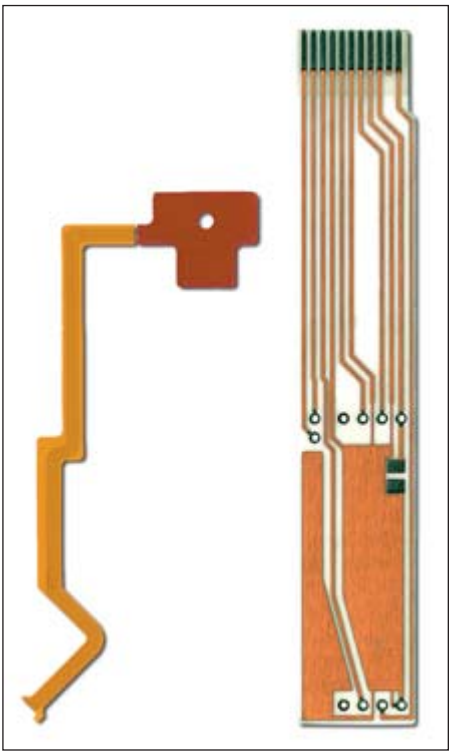
GROUND PLANE: A conductor layer, or portion of a conductor layer, used as a common reference point for circuit returns, shielding, or heat sinking.

HDI (HIGH DENSITY INTERCONNECT): Ultra fine-geometry multilayer PCB constructed with conductive surface microvia connections between layers. These boards also usually include buried and/or blind vias are made by sequential lamination.

HOLE BREAKOUT: A condition in which a hole is partially surrounded by the land.

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

HOT AIR SOLDER LEVELING (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.



Flexible circuit boards, made from polyimide material

IMAGING: The process by which panelization data are transferred to the photoplotter, which in turn uses light to transfer a negative 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.

INNERLAYERS: The internal layers of Laminate and Metal Foil within a Multilayer Board.

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.

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

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.

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

LAYUP: The process in which treated Prepregs and Copper Foils are assembled for pressing.

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

LIQUID PHOTOIMAGEABLE SOLDERMASK (LPI): A Mask sprayed on using photographic Imaging techniques to control deposition.

LINE: See Conductor.

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

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 or rolls.

MICROSECTIONING: The preparation of a specimen of a material, or materials, that is to be used in metallographic examination. This usually consists of cutting out a cross-section fol-

lowed 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 multilayer PCB. Often used to refer to any small geometry connecting holes created by laser drilling

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.

MULTILAYER BOARD: Printed boards consisting of a number (four or greater) 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.

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

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

PAD: See 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.

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.

PHOTOGRAPHIC IMAGE: An image in a photo mask or in an emulsion that is on a film or plate.

PHOTOPLOTTING: A photographic process whereby an image is generated by a controlled light beam that directly exposes a light-sensitive material.

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

A six-opening multilayer lamination press

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

PLATED THROUGH-HOLE: A hole with 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 VOID: The area of absence of s specific metal from a specific cross-sectional area.

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

PREPREG: Sheet material (e.g., glass fabric) impregnated with a resin cured to an intermediate stage (B-stage resin).

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.

PULSE PLATING: A method of plating that uses pulses instead of a direct current. **REFLOW:** The melting of an electrodeposited tin/lead followed by solidification. The surface has the appearance and physical characteristics of being hot-dipped.

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.

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

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.

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.

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

ROUTER: A machine that cuts away portions of the laminate to form the desired shape and size of the printed board.

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.

SINGLE-SIDED BOARD: A printed board with conductive pattern on one side only.

SMOBC: See Soldermask Over Bare Copper.

SOLDERMASK OVER BARE COPPER (SMOBC): 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 eliminates tin lead under the pads.

Surface mount technology (SMT): 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 LEVELING: The process by which the board is exposed to hot oil or hot air to remove any excess solder from holes and lands.

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

STEP-AND-REPEAT: A method by which successive exposures of a single image are made to produce a multiple image production master.

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:

TEST COUPON: A portion of a printed board or of a panel containing printed coupons used to determine the acceptability of such a board.

Tg: Glass transition temperature, the point at which rising temperatures causes the solid base laminate to start to exhib-

it 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 PCB or a panel of PCBs for registration and hold-down purposes during the manufacturing process.

TOP SIDE: See Component Side.

TRACE: A common term for Conductor.

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.

UL: Underwriters Laboratories Inc., an independent product safety testing and certification organization.

UNDERWRITERS SYMBOL: A logotype denoting that a product has been recognized (accepted) by Underwriters Laboratories Inc. (UL).

UV CURING: Polymerizing, hardening, or cross linking a low molecular weight resinous material in a wet coating ink using ultra violet light as an energy source.

VIA: A plated through hole that is used as an interlayer connection, but does not have component lead or other reinforcing material inserted in it.

VOID: The absence of any substances in a localized area.

WAVE SOLDERING: A process wherein assembled printed boards are brought in contact with a continuously flowing and circulating mass of solder.

WICKING: Migration of copper salts into the glass fibers of the insulating material.

WIP: An acronym for work in progress.