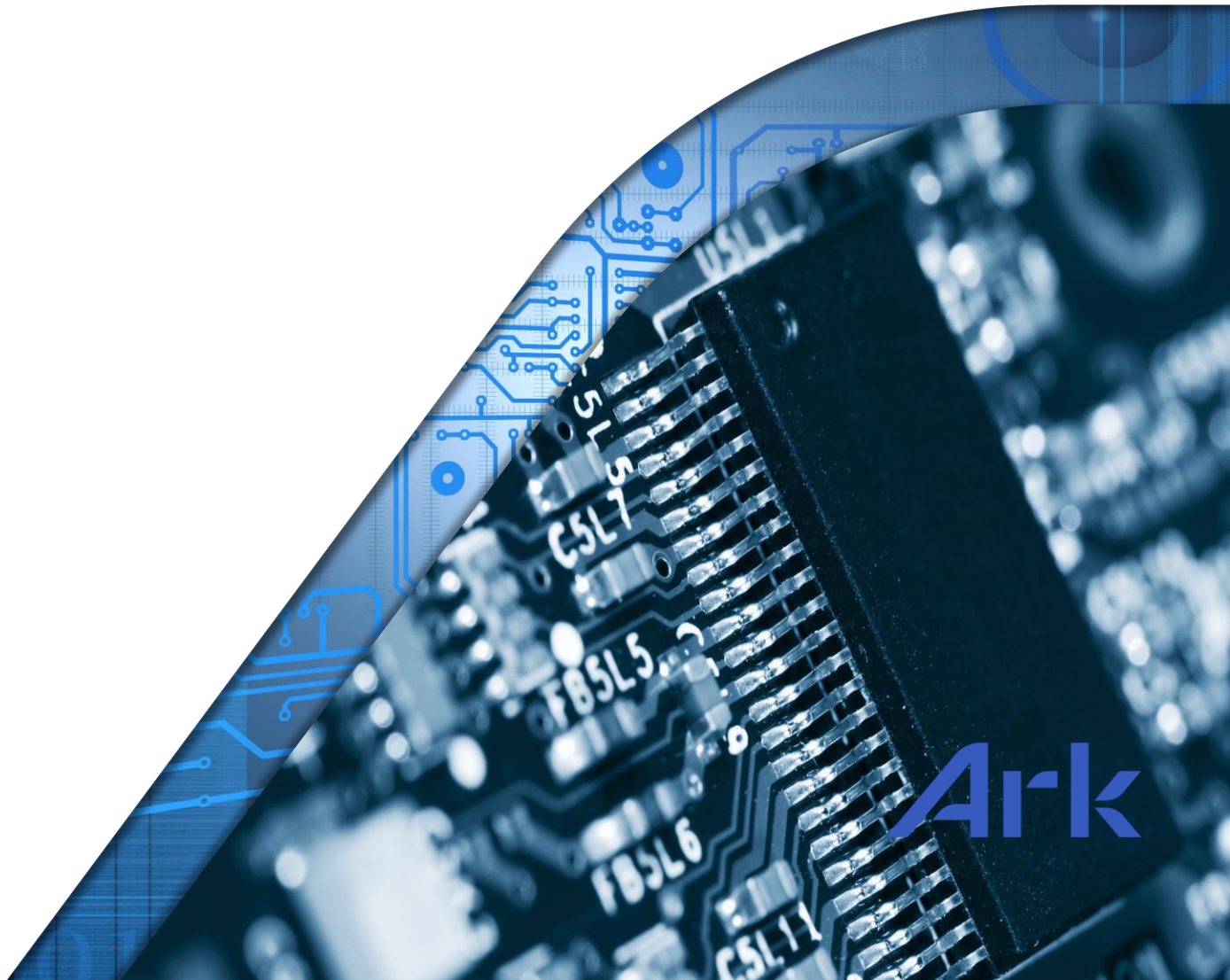


Ark Electronics

Contract Manufacturing Glossary

180 Terms About Electronic
Manufacturing Services



Ark

About Ark

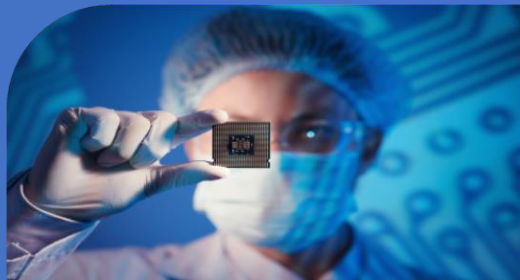
Delivering Intelligent Innovation with Integrity

Ark provides end-to-end, turnkey services for developing a wide range of products, from PCBA to finished box builds.

With niche Research & Development expertise into IoT (Internet of Things), PCBA Design, Advanced Voice Solutions, Hardware Development, Finished Products, Prototyping Development, Product Re-Engineering, Product Sustenance & Manufacturing across several industry verticals such as Consumer Electronics, Automotive, Industrial Automation, Smart Home Automation and Wearable Technologies.

Our Vision

Customizing technology for the benefit of human-kind and the environment – one customer at a time



Our Purpose

To Deliver Intelligent Innovation
with Integrity

"ARK has worked with [us] for over 7 years and won our 'Outstanding Suppliers' for two consecutive years. Ark's operational control, quality control and order fulfillment ability are among the best in the industry."

Hisense



March 2017:
ISO 9001 : 2015 (renewed)
Standard, Quality
Management System
Certification



March 2017:
ISO 14001 : 2015 (renewed)
Standard, Environmental
Management System
Certification



September 2017:
NQA: IATF 16949 : 2016
Automotive Quality
Management System
Certification

Contents

Acceptance tests	5	Cosmetic Defect	8
Additive process	5	C-PGA	8
ALIVH	5	CSP	8
Anisotropic adhesive	5	Conductive	8
Annular Ring	5	CTE	8
AOI	5	D code	8
AQL	5	Datum Reference	8
ASCII File	5	Daughter Board	8
ASIC	5	Dendritic growth	8
Bare Board	5	Destructive testing	8
Bed-of-Nails	5	Dewetting	8
BGA	5	DFM	8
Blind Via Hole	5	DFT	8
Board Type (Single Unit and Panel)	5	DIP	8
BOM	5	DNP	8
Boundary scan	5	DRC	8
Bow	6	Dry film	8
Breakdown voltage	6	ECN	9
Bridging	6	Edge Dip Solderability	9
BT / Epoxy	6	Electroconductive Paste Printed Board	9
BTC	6	EMC	9
Buildability	6	EMS	9
Buried Resistance Board	6	EOL	9
Burn-in Testing	6	ESD	9
CAE	6	FR-1	9
CAM Files	6	FR-2	9
Carbon Mask	6	FR-4	9
Castellation	6	FR-404	9
CEM	6	FR-406	9
CEM-1	6	FR-408	9
Centroid data file	6	FR-6	9
C-Flat Pack	7	Gerber File	9
Check plots	7	GIL Grade MC3D	9
Chip scale package (CSP)	7	Golden	10
Circuitry Layer	7	HA	10
Cladding	7	Haloing	10
C-LCC	7	HASL	10
Clean room	7	Hdi	10
COB	7	Heavy copper PCB	10
COF	7	HiPot Test	10
COG	7	HS	10
Conformal coating	7	Icicle	10
Constraining core substrate	7	ICT	10
Convection/IR	7	Image	10
Copper Weight	7	Induction soldering	10
Core Group	7	In-line placement	10
		IPC	10
		IPC Class 2	10

IPC Class 3	10	Probing systems	13
IPD	10	Protoboard	13
IS	11	PRP	13
ISO 13485	11	P-SO	13
ISO 9001	11	P-TSOP	13
ITAR	11	QFP	13
J-Standard	11	QMS	13
Just-in-time (JIT)	11	Reflow soldering	14
KGD	11	Ref	14
Known Good Board	11	Rework	14
Laser Photoplotter	11	Rigidflex	14
LCCC	11	RMA	14
lead frame	11	Robber	14
Legend	11	RoHS	14
Line certification	11	Schematic	14
LPI	11	Silkscreen	14
LTB	11	SIMMA	14
M2M	11	Single-center reflow soldering	14
Major Defect	11	SiP	14
Mask	12	SMT	14
MCM	12	Snapback	14
MDA	12	Solderability	14
Measling	12	Soldermask	15
Membrane Switch	12	SPC	15
Micro circuits	12	Step-and-Repeat	15
Micro	12	T/d	15
Minimum Annular Ring	12	T/g	15
Minimum Electrical Spacing	12	TAB	15
Minor Defect	12	Tented via	15
Misregistration	12	Test Coupon	15
MTBF	12	Tombstoning	15
Multilayer board	12	Traveler	15
NCNR	12	Turnkey Assembly	15
NPI	12	Type I, II, III assembly	15
NRE	12	UL	15
OEM	12	UL Certified Product	15
OSP	12	Underwriters Symbol	15
Packaging density	12	UUT	15
PCA	13	Uv cure	16
PCB	13	Vibration Testing	16
PCB design service bureau	13	VLSI	16
PFMEA	13	WIP	16
Pick-and-place	13	X-Ray Inspection	16
Pilot Order	13	Yield	16
PIV	13	Zero Defects Sampling	16
Placement rate	13	ZIF	16
P-QFP	13	ZIP	16
Printed Wiring Board	13	SOURCES	16

Acceptance tests

A set of tests performed to determine the acceptability of a PCB.

Additive process

A method for manufacturing PCB conductive patterns via selective deposition of conductive material (copper, solder, etc.) on a substrate.

ALIVH

Short for any layer inner via hole, this is a type of technology used to build multi-layer BUM PCBs. This method uses a solder to create an electrical connection between PCB layers. ALIVH often replaces traditional vias and is a useful production method for creating high-density BUM PCBs.

Anisotropic adhesive

A material filled with particles that will conduct current in the Z-axis only. (Also called Z-axis adhesive.)

Annular Ring

This term refers to the copper pad area that is left after a hole is drilled through it. This ring is measured from the edge of the pad to the edge of the hole and is an important consideration in PCB design, as it allows an electrical connection to be made from one side of the hole to the other.

AOI

Short for automated optical inspection, AOI refers to a type of inspection method used to find potential problems concerning soldering performance in multi-layer PCBs with components mounted on. The AOI equipment finds these issues by capturing images of the inner PCB surfaces, looking for any possible issues in terms of displacement, polarity etc.

AQL

Short for acceptance quality limit, AQL refers to the acceptable number of defective boards produced within a production run. These are identified, counted and removed during inspection. AQL is an important figure for monitoring the quality of an assembler's production practices.

ASCII File

Simple, text-based file format. Great for communicating design information for manufacturing a PCB.

ASIC

Application-specific integrated circuit. An IC designed for a specific function or product.

Bare Board

This term refers to a circuit board with no components mounted on it.

Bed-of-Nails

A method of testing printed circuit boards that employs a test figure mounting an array of contact pins configured so as to engage plated thru-holes on the board.

BGA

Ball grid array. A chip package in which the external terminals form a grid-style array, and are in contact with solder balls which carry the electrical connection to outside of the package. The PCB design will have round landing pads to which the solder balls will be soldered when the package and PCB are heated in a reflow oven.

Blind Via Hole

A plated-through hole connecting an outer layer to one or more internal conductor layers of a multilayer printed board but not extending fully through all of the layers of base material of the board.

Board Type (Single Unit and Panel)

This indicates the manufacturing method of a PCB in terms of volume. Usually, a board is classified into one of two types: single unit or panel. In single unit manufacturing, PCBs are fabricated one by one. In panel manufacturing, on the other hand, multiple units of PCBs are manufactured in a single panel.

BOM

Bill of Materials. A list of the raw materials, assemblies, components, and quantities of each needed to manufacture a project.

Boundary scan

A self-test method for active components in which a built-in test bus is used to access input/output pins.

Bow

the measurement of flatness of a circuit board, between corners and the center.

Breakdown voltage

Voltage at which the breakdown of a dielectric or insulator occurs. The breakdown voltage of a material is not a definite value because it is a form of failure and there is a statistical probability whether the material will fail at a given voltage. When a value is given it is usually the mean breakdown voltage of a large sample.

Breakover voltage

Minimum voltage required to cause a DIAC to break down and conduct.

Bridging

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

BT / Epoxy

The blending of bismaleimide / triazine and epoxy resin provides enhanced thermal, mechanical and electrical performance over standard epoxy systems.

BTC

Bottom termination component. Components where the leads are flat against the board and completely hidden from view. Located on the bottom so they can mate to the circuit board.

Buildability

Team meeting to review customer designs against manufacturing process capabilities. Used to identify possible failure modes prior to fabrication

Buried Resistance Board

The term refers to a printed circuit board with resistors buried inside. This design improves the integrity of resistant components to improve the overall function and reliability of the PCB.

Burn-in Testing

Electrically stressing a device at an elevated temperature, usually over a long period of time, to root out defects.

CAE

Computer Assisted Engineering. In electronics work, CAE refers to schematic software packages.

CAM Files

CAM means Computer Aided Manufacturing. These are the data files used directly in the manufacture of printed wiring. The types of CAM files are 1) Gerber file, which controls a photoplotter, 2) NC Drill file, which controls an NC Drill machine and 3) fab and assembly drawings in soft form (pen-plotter files). CAM files represent the valuable final product of PCB design. They are handed off to the board house which further refines and manipulates CAM data in their processes, for example in step- and-repeat panelization. Some PCB design software companies refer to all plotter or printer files as CAM file, although some of the plots may be check plots which are not used in manufacture.

Carbon Mask

This is a type of conductive carbon paste that is added to the surface of a pad. Made with a combination of resin and carbon toner, carbon masks are heat-cured and are typically applied to jumpers, keys, etc.

Castellation

Metallized semicircular radial features on the edges of LCCCs that interconnect conducting surfaces. Castellations typically are found on all four edges of a leadless chip carrier. Each lies within the termination area for direct attachment to the land patterns.

CEM

Contract Electronics Manufacturers. Companies that make electronics products under contract for other companies. They typically take on partial or whole manufacturing responsibility for original equipment manufacturers (OEMs).

CEM-1

A older NEMA grade of printed circuit laminate having a substrate of woven glass surfaces over a cellulose paper core and a resin binder of epoxy. It has good electrical and mechanical properties, it is inexpensive and can be punched.

Centroid data file

A computer aided design file that contains information utilized during printed circuit board

prototyping, fabrication, and assembly. It lists the reference designation, X, Y, side, top, bottom, and rotation coordinates that automated machinery uses to fabricate and place electronic components on printed circuit boards.

C-Flat Pack

Ceramic Flat Pack. A square, ceramic surface mount chip that provides leads on all four sides, affording a high lead count in a small area. It is a US military standardized Printed-circuit-board surface-mount-component package. Also commonly referred to as ceramic quad flatpack or ceramic quad flat-package.

Check plots

Photo plots that are suitable for checking only. Pads are represented as circles and thick traces as rectangular outlines instead of filled-in artwork. This technique is used to enhance transparency of multiple layers or may be holes only for drill checking.

Chip scale package (CSP)

A high-density IC packaging process in which the package is slightly larger than the chip (less than 1.5 times the chip area).

Circuitry Layer

The layer of a PCB containing conductors, including ground and voltage planes.

Cladding

A thin layer of metal foil bonded to a substrate to form the conductive PCB pattern.

C-LCC

Ceramic Leaded Chip Carrier. Ceramic package that's hermetically sealed, and has metallic contacts called castellations, which are flush with the package or recessed instead of leads consisting of metal prongs or wires. The castellations are usually on all four sides of the package.

Clean room

A room in which the concentration of air born particles is controlled to very specified limits to lessen the effect dust has on imaging.

COB

Chip On Board. The term is used to refer to a configuration in which a chip is mounted directly onto printed circuit boards, and techniques used for such mounting. As part of this process, a bare chip

that is mounted directly onto the printed circuit board. After wires are attached, a glob of epoxy or plastic is used to cover the chip and its connections.

COF

Chip on Flex. Refers to the mounting of flip chip components directly to flexible circuits. Lower product weight and size, cost and reliability risk can be achieved with Chip on Flex. Product shapes that would otherwise be difficult or impossible to produce economically without the use of a flexible substrate are made possible with Chip on Flex.

COG

Chip-On-Glass. A flip chip bonding technology for direct connection assembly of bare integrated circuits on glass substrate by using Anisotropic Conductive Film. It reduces the assembly area to the highest possible packing density, and allows cost-effective mounting of driver chips. Chip On Glass is a reliable and well established technology, often used in the automobile industry.

Conformal coating

An insulated protective coating that conforms to the components and is applied on the completed board assembly

Constraining core substrate

A composite PCB consisting of epoxy-glass layers bound to a low thermal expansion core material, such as copper-invar-copper, graphite-epoxy and aramid fiber-epoxy. The core constrains the expansion of the outer layers to match the expansion coefficient of ceramic chip carriers.

Convection/IR

A solder reflow oven for SMD interconnect that combines convection and infrared (IR) radiation heating.

Copper Weight

This term is used to indicate thickness of copper foil on each layer of a PCB. It's typically expressed in ounces of copper per square foot.

Core Group

Daily operational meeting held on each shift to communicate current status of plant in terms of producing and delivering high quality, cost-effective circuit boards to customers on time.

Cosmetic Defect

A defect such as a slight change in its usual color that doesn't affect a board's functionality.

C-PGA

Ceramic Pin Grid Array. A ceramic package capable of providing up to several hundred pins, all located on its underside. Its design minimizes the distance signals must travel from the chip to each designated pin. C-PGA is often used to package computer chips.

CSP

A chip scale package. A type of integrated circuit package. Since being introduced, they've become one of the biggest packaging trends in recent history. Manufacturing advantages range from package size reduction and saving printed circuit board (PCB) routing space to self-alignment characteristics during PCB assembly reflow and lack of bent leads which can cause coplanarity issues. CSPs are used to increase PCB assembly yields and lower manufacturing costs.

Conductive ink

The paste used on thick film materials to form the circuit pattern, usually containing metal, metal oxide and solvent.

CTE

Coefficient of Thermal Expansion. Solids undergo maximum expansion when their surface temperature is increased by heating, and contract when they are cooled. This temperature change response is referred to as the coefficient of thermal expansion. It is the rate change in the size of an object with the rate change in the temperature.

D code

A datum in a Gerber file which acts as a command to a photo plotter. A D code in a Gerber file takes the form of a number prefixed by the letter. "D20".

Datum Reference

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

Daughter Board

The "daughter" of a "mother" board, a daughter board contains plugs, pins, sockets and connectors and plays a big role in internal connections for electronic devices and computers.

Dendritic growth

A branching of solder filaments ("whiskers") at interconnect sites because of the presence of moisture or electrical bias, threatening an electrical short.

Destructive testing

Sectioning a portion of printed circuit panel and examining the sections with a microscope. This is performed on coupons, not the functional part of the PCB.

Dewetting

A condition that occurs when molten solder has coated a surface and then recedes, leaving irregularly shaped globules of solder separated by areas covered with a thin solder film; base metal is not exposed.

DFM

Design for manufacturability. Designing a product to be produced in the most efficient manner possible in terms of time, cost and resources, taking into consideration how the product will be processed, and using the existing skill base to achieve the highest yields possible.

DFT

Design For Testability. Design with testability features included, allowing for easier application of manufacturing tests to the PCB assembly. Validates test plans.

DIP

Dual in-line package. Integrated circuit package having two rows of connecting pins.

DNP

Do Not Populate. May be on the Bill of Materials for components/reference designators that will not get placed on the PCB during assembly.

DRC

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

Dry film

A photo imageable material which is laminated on to a bare copper panel, it is exposed with 365 nm UV light through a negative photo tool, the exposed dry film is hardened by the UV light, the unexposed film is

washed away in a developer solution of .8 percent sodium carbonate

ECN

Engineering Change Notice. A document communicating a change in the design of a product.

Edge Dip Solderability

Test A solderability test performed by taking a specially prepared specimen, fluxing it with a non-activated rosin flux, and then immersing it into a pot of molten solder at a predetermined rate of immersion for a predetermined dwell time, and then withdrawing it at a predetermined rate.

Electroconductive Paste Printed Board

This term is used to describe PCBs that are manufactured using a silkscreen printing method. The process involves applying an electroconductive printing paste to set traces and to implement stable through-hole connections.

EMC

Electromagnetic compatibility refers to the ability of electronic equipment and systems to operate in proximity of other electromechanical devices, without causing or suffering unacceptable output or performance degradation. Medical devices can be particularly vulnerable to electromagnetic interference if the levels of electromagnetic energy in its environment exceed the electromagnetic immunity of the device. That said, electromagnetic compatibility and interference affects all electronic devices.

EMS

Electronics Manufacturing Services. EMS design, manufacturing, testing, distribution, and provide repair for electronic components for OEMs. Often provided by a contracted company.

EOL

End of Life. When a product becomes obsolete or otherwise not worth producing, and a vendor stops marketing or selling it.

ESD

Electrostatic Discharge. A sudden discharge of electric current between two objects with different charges and numbers of electrons. Can damage your electronic products.

FR-1

A low-grade version of FR-2. Tg 130c.

FR-2

A grade of Flame-Retardant industrial laminate having a substrate of paper and a resin binder of phenolic. It is suitable for printed circuit board laminate and cheaper than the woven glass fabrics such as FR-4. Tg 105 c.

FR-4

A grade of Flame-Retardant industrial laminate having a substrate of woven-glass fabric and resin binder of epoxy. FR-4 is the most common dielectric material used in the construction of PCBs in the USA. Its dielectric constant is from 4.4 to 5.2 at below-microwave frequencies. As frequency climbs over 1 GHz, the dielectric constant of FR-4 gradually drops. Tg 150 to 175 c.

FR-404

A multifunctional epoxy system that is a standard multilayer material.

FR-406

A Multifunctional Epoxy Laminate and Prepreg that has a Tg of 170 C (DSC used in applications where a higher temperature material is needed).

FR-408

A high-performance FR-4 epoxy laminate and prepreg that has a Tg of 180 C (DSC). low dielectric constant is an ideal for faster signal speed and improved signal integrity. Used when layer counts begin to exceed 8 layers because it's higher thermal properties minimize Z-axis expansion and the potential for barrel cracking and pad lifting in complex designs.

FR-6

Fire-Retardant glass-and-polyester substrate material for electronic circuits. Inexpensive; popular for automobile electronics.

Gerber File

Data file used to control a photoplotter. Named after Gerber Scientific Co., who made the original vector photoplotter.

GIL Grade MC3D

A composite laminate comprised of woven glass surface sheets on both sides of a glass paper core.

MC3D exhibits excellent electricals with a low and stable dielectric constant and dissipation factor.

Golden

A component or assembly already tested as functional to spec and used, via comparisons, to test similar units. Also known as a known good board.

HA

Hand Assembly. A form of assembly completed by hand when automated electronic methods aren't the best option.

Haloing

Mechanically induced fracturing delimitation 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.

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 a series of hot air jets.

Hdi

High Density interconnect, Very fine lines and thin dielectrics. Made with sequential lamination. Microvias interconnect the layers.

Heavy copper PCB

Circuit boards with more than 4 oz of copper for power circuits.

HiPot Test

High Potential Test. Used to verify electrical insulation of cables, PCBs and wired harnesses, generally conducted at 40 V or more.

HS

Hand Solder. The soldering of a wire or component manually. This includes use of a soldering iron or gun.

Icicle

An unacceptable solder point that protrudes out of a solder joint, but does not make contact with another conductor.

ICT

In-circuit test. A component-by-component test within a circuit for electrical function to verify proper placement and orientation.

Image

That portion on artwork masters, working tools, silk screens, or photo masks that would be considered the photographic image. Also, would include images created with photoresists or silk-screening techniques. Generally, "one image" refers to a single circuit board image; thus, there may be several images per flat.

Induction soldering

An interconnect method in which solder, generally preforms, is reflowed.

In-line placement

A method of PCB assembly permitting conveyORIZED board handling into and out of the component placement equipment.

IPC

Institute for Printed Circuits. A trade association that provides global standards for acceptability in the electronics industry with the goal of standardizing assembly and production requirements.

IPC Class 2

Institute for Printed Circuits Class 2 standard. A type of workmanship standard for building electronic assemblies defined by the IPC. Includes products for which continued performance and extended life is required, but would not cause failures if it did not provide uninterrupted service.

IPC Class 3

Institute for Printed Circuits Class 3 standard. A workmanship standard for building electronic assemblies defined by the IPC. More stringent than Class 2. Requires high performance on-demand in potentially harsh or life-threatening situations.

IPD

Integrated Passive Devices are fabricated using standard wafer fab technologies such as thin film and photolithography processing. IPDs can be designed as flip chip mountable or wire bondable components and the substrates for IPDs usually are thin film substrates like silicon, alumina or glass. IPD

technology offers the ideal trade-off for system in package integration.

IS

Intrinsic Safety. Protection techniques for safe operation of electronic equipment in dangerous areas by limiting the energy available for ignition.

ISO 13485

International Organization for Standardization medical certification. Mandates for a comprehensive quality management system for the design and manufacturing of medical devices.

ISO 9001

International Organization for Standardization quality management certification. A certification for quality-management systems. Demonstrates a company's ability to consistently provide services that meet both customer needs and regulations.

ITAR

International Traffic in Arms Regulation. Controls imports and exports of defense-related articles on the United States Munitions list to ensure the technology doesn't reach the wrong hands. Any manufacturer, broker, or exporter of defense services and related data must be ITAR-compliant.

J-Standard

IPC certification covering materials, methods and verification criteria for producing high-quality soldered interconnections.

Just-in-time (JIT)

Minimizing inventory by supplying material and components directly to the manufacturing line just before incorporating them into the product.

KGD

Semiconductor die that has been tested and is known to function to specification.

Known Good Board

A "gold standard" used to compare the boards you're building to. This ensures the test is actually functional.

Laser Photoplotter

(also "laser plotter") A photoplotter which simulates a vector photoplotter by using software to create a raster image of the individual objects in a CAD database, then plotting the image as a series of lines

of dots at very fine resolution. A laser photoplotter is capable of more accurate and consistent plots than a vector photoplotter.

LCCC

Leadless ceramic chip carrier. A ceramic, hermetically sealed IC package commonly used for military applications. The package has metallized castellations on four sides for interconnecting to the substrate.

lead frame

A common type of chip package utilizing metal leads, and providing external terminals and mechanical support to align them. This metal structure carries electrical signals from the die to the outside

Legend

This is a shorthand guide for marking component names and positions. Legends help ease the assembly and maintenance processes.

Line certification

Assurance that a production line sequence is under management and will produce reliable PCBs in compliance with requirements.

LPI

Liquid Photo imageable Solder Mask. A mask sprayed on 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.

LTB

Last Time Buy. The final date a supplier will ship a product out before it stops manufacturing it.

M2M

Machine To Machine. Any form of technology that allows direct communication between devices using any communications channel, whether wired or wireless.

Major Defect

A defect that could result in a failure or significantly reduces the usability of the part for its intended purpose.

Mask

A material applied to enable selective etching, plating, or the application of solder to a printed circuit board

MCM

Multichip module. A circuit comprised of two or more silicon devices bonded directly to a substrate by wire bond, TAB or flip chip.

MDA

Manufacturer Defect Analysis. A straightforward test to reveal manufacturing defects.

Measling

Condition existing in the base laminate in the form of discrete white spots or "crosses" below the surface of the base laminate, reflecting a separation of fibers in the glass cloth at the weave intersection.

Membrane Switch

A membrane switch is applied to the front of a finished PCB. It indicates functions of the PCB and components, such as key functions, indicators and other parts. The membrane also provides protection for the PCB in the form of waterproofing and humidity protection.

Micro circuits

Very fine lines 2 mil and less and small micro vias 3 mil and less.

Micro sectioning

The preparation of a specimen for the microscopic examination of the material to be examined, usually by cutting out a cross section, followed by encapsulation, polishing, etching, staining, and so on.

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 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 reduce the usability of the unit for its intended purpose. It may be a departure from established standards having no significant bearing on the effective use or operation of the unit.

Misregistration

The lack of dimensional conformity between successively produced features or patterns.

MTBF

Mean time between failure. The statistical average time interval, usually in hours, to be expected between operating unit failures.

Multilayer board

A PCB that uses more than two layers for conductor routing. Internal layers are connected to the outer layers by way of plated via holes.

NCNR

Non-Cancellable Non-Returnable. A purchase order that cannot be returned to a supplier in the wake of a reduction in demand.

NPI

New Product Introduction. Launch of a new product to the market. Includes design, testing, manufacturing, and analysis.

NRE

Non-Recurring Engineering. One-time costs associated with engineering work at the beginning of production.

OEM

Original Equipment Manufacturer. OEMs resell another company's product or component under their own name and branding.

OSP

Organic solderability preservative is a method for coating of printed circuit boards. OSP uses a water-based organic compound that selectively bonds to copper and protects it until soldering. Compounds typically used in OSPs are from the azole family

Packaging density

The number of parts (passive/active components, connectors, etc.) placed on a PCB; expressed as low, medium or high.

PCA

Printed circuit assembly. General term for a PCB fully assembled with all electrical, electromechanical and mechanical components.

PCB

Printed circuit board. The general term for completely processed printed circuit configurations. It includes rigid or flexible, single, double or multilayer boards. A substrate of epoxy glass, clad metal or other material upon which a pattern of conductive traces is formed to interconnect components. (Also called a printed wiring board.)

PCB design service bureau

A business engaged in PCB design as a service for others, especially electrical engineers. The word bureau is French for desk, or office, and this service is indeed performed from an office while sitting at a desk. Also called PCB design shop.

PFMEA

Process Failure Mode Effects Analysis. Looks at each process step to identify risks and possible errors in manufacturing. Investigates man, method, material, machinery, and environment.

Pick-and-place

A programmable method of component placement using machines to automatically pick parts from a feeder, then move to a precise location on a PCB and place them in the correct site.

Pilot Order

First production order going through process.

PIV

Peak inverse voltage. The maximum rated value of an AC voltage acting in the direction opposite to that in which a device is designed to pass current.

Placement rate

The speed of a complete component placement cycle beginning with component/part pick-up, move to the placement site and the return to the feeding source.

P-QFP

A Plastic Quad Flat Pack is a type of integrated circuit packaging technology that allows gull wing pins to extend from all four sides of the body.

Printed Wiring Board

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

Probing systems

Equipment for high-reliability testing of PCBs, components and assemblies. Probing devices range from manual for lab use to low-volume test via computer-controlled systems.

Protoboard

Board with provision for attaching components without solder. Also called a breadboard. Primarily used for constructing experimental circuits.

PRP

Product Realization Process. Development process used throughout the lifecycle of a product. Guides planning, tracking, requirements, design, verification, document control, production, & more.

P-SO

Plastic Small Outlines include JEDEC standard compliance, footprint and height 50% of DIP, two side leaded for routing simplicity, 50 mil (1.27 mm) pitch for SMT simplicity, and gull wing formed leads for improved surface mount technology manufacturing.

P-TSOP

Plastic Thin Small Outline Packages are a type of surface mount integrated circuit package. They are very low-profile (about 1mm) and have tight lead spacing (as low as 0.5mm), frequently used for RAM and Flash memory ICs. They have a high pin count and are small volume.

QFP

Quad flat pack. A term used for SMT packages with leads on all four sides. Most commonly used to describe packages with gull wing leads. Also known as a flat pack, but flat packs may have gull wing leads on either two or four sides.

QMS

Quality Management System. A set of policies and procedures that defines efforts within an organization to meet customer requirements and continually improve its system

Reflow soldering

A process of joining metallic surfaces through the mass heating/cooling of preplaced solder paste between two surfaces (i.e., component leads and PCB pads).

Ref Des

Reference Designator. The name of component 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. By contrast, on an assembly drawing a reference designator is often placed within the boundaries of a footprint – a very useful technique for eliminating ambiguity on a crowded board where reference designators in the silk screen may be near more than one component

Rework

Repetition of a manufacturing process to bring an assembly into compatibility with a spec or contract requirement.

Rigidflex

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.

RMA

Return Material Authorization. An agreement between customer and supplier allowing the customer to return goods for a refund during the warranty period.

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 assuring that the parts will receive a uniform electroplated coating.

RoHS

Restriction of Hazardous Substances. Restricts the use of specific hazardous materials, including lead, during assembly of electronic products

Schematic

A technical drawing that illustrates the connections between PCB components. Schematics will often include abstract representations of components instead of pictures and is an important first step in PCB design.

Silkscreen

(Also called "silkscreen legend")

This is a layer of epoxy ink applied to a PCB that contains component names and positions. The labels included on silkscreens help to direct workers through the assembly process. Typically, silkscreens are white, which helps the labels stand out against the PCB's solder mask.

SIMMA

Single in-line memory module is a module containing one or several random-access memory (RAM) chips on a small circuit board with pins that connect to the computer motherboard.

Single-center reflow soldering

A process in which both surface mount, held by adhesives, and through-hole components are reflow soldered.

SiP

A System In A Package is a number of integrated circuits enclosed in a single module. The SiP performs all or most of the functions of an electronic system, and is typically used inside of mobile phones and digital music players.

SMT

Surface mount technology. A method of assembling PCBs or hybrid circuits in which components are mounted onto the surface rather than inserted into through-holes.

Snapback

The return of a stencil to normal (flat plane) after its deflection by a squeegee across its surface

Solderability

The ability of a conductor (lead, pad or trace) to be wetted (become solderable) to form a strong bond.

Soldermask

A PCB processing technique in which all surfaces are covered by a plastic coating except those for connections to be soldered.

SPC

Statistical Process Control is an industry-standard methodology for measuring and controlling quality during the manufacturing process. Quality data in the form of product or process measurements are obtained in real-time during manufacturing. This data is then plotted on a graph with pre-determined control limits. Control limits are determined by the capability of the process, whereas specification limits are determined by the client's needs.

Step-and-Repeat

A method by which successive copies of a single image are made to produce a multiple-up filling the panel.

T/d

Temperature of destruction, where the circuit loses 5 percent of its volume due to out gassing.

T/g

Glass transition temperature, in degree C the point at which the material starts to become soft and plastic like, also the point where the z axis starts to expand in linearly.

TAB

Tape automated bonding. The process of bonding the IC die to patterned inner leads on plastic tape and, in a subsequent operation, positioning and bonding the outer leads to the surface of the substrate.

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 top side of the board and left uncovered on the bottom side to permit probing from that side only with a test fixture.

Test Coupon

An area of patterns on the same fabrication panel as the PWB, but separate from the electrical circuits and outside the actual board outline. It is cut away

from the printed wiring board prior to assembly and soldering of components. It can be used for destructive testing.

Tombstoning

A soldering defect in which a chip-type part is "pulled" into a vertical or near-vertical position with only one of its terminals connected. It is caused by force imbalances during solder reflow. (Also called drawbridging.)

Traveler

A "recipe" for the manufacture of a board. It "travels" with each order from start to finish. The traveler identifies each order and gives instructions for each step in the process. It also provides information for traceability and history.

Turnkey Assembly

The manufacturer provides the entire manufacturing process of a product, from design to production to final assembly.

Type I, II, III assembly

Designating PCB assemblies: (I) SMDs mounted on one or both sides of the board; (II) mixed technology having leaded (through-hole) parts mounted on the primary side and SMDs on one or both sides; and (III) mixed technology featuring passive SMDs on the secondary side and leaded components mounted to the primary side.

UL

Underwriters Laboratories Inc. An independent testing organization that evaluates performance and safety of electrical equipment.

UL Certified Product

Underwriters Laboratory nationally recognized standards for safety. A global safety consulting & certification company that provides recognized standards of safety. OSHA-approved to conduct safety testing in electronics & other industries.

Underwriters Symbol

A logotype denoting that the product has been recognized by Underwriters Laboratory, Inc. (UL).

UUT

Unit under test. A term applied to any component or assembly being tested by automatic testing equipment.

UV cure

Polymerizing Hardening or cross inking a material by exposing to ultra violet light.

Vibration Testing

Testing involving use of a shaker at a predetermined frequency and time to see if components stop functioning or detach from the product.

VLSI

Very Large-Scale Integration.

WIP

Work In Progress.

X-Ray Inspection

Inspection of solder joints hidden from view, mainly bottom termination components.

Yield

The ratio of good assemblies at the end of processing to the number initially entered.

Zero Defects Sampling

A statistical based attribute sampling plan ($C = 0$) where a given sample of parts is inspected and any defects found are cause for rejection of the entire lot.

ZIF

Zero insertion force is a type of integrated circuit socket or electrical connector that requires very little force for insertion. With a ZIF socket, before the integrated circuit is inserted, a lever on the side of the socket is moved pushing all the sprung contacts apart so that the IC can be inserted with very little force. The lever is then moved back, allowing the contacts to close and grip the pins of the IC.

ZIP

Zig-zag in-line package is an integrated circuit encapsulated in a slab of plastic with 20 or 40 pins, measuring (for the ZIP-20 package) about 3 mm x 30 mm x 10 mm. The package's pins protrude in two rows from one of the long edges. The two rows are staggered by 1.27 mm (0.05"), giving them a zig-zag appearance, and allowing them to be spaced more closely than a rectangular grid would allow. The pins are inserted into holes in a printed circuit board, with the packages standing at right-angles to the board, allowing them to be placed closer together than DIPs of the same size. They're commonly used for dynamic RAM chips.

SOURCES

en.wikipedia.org
www.pcbdirectory.com
www.protoexpress.com
www.pcbway.com
www.eurocircuits.com
www.altium.com
www.allaboutcircuits.com
www.assemblymag.com