

GLOSSARY OF TECHNICAL TERMS

This glossary contains definitions of certain technical terms used in this document in connection with our business operations, manufacturing processes and product technologies. Certain definitions may not correspond to standard industry definitions.

“ADAS (Advanced Driver Assistance System)”	A technology that assist drivers with the safe operation of a vehicle using automated technology such as sensors and cameras to detect obstacles or driving errors and respond accordingly.
“AOI (Automated Optical Inspection)”	An automated inspection technology that uses optical imaging systems to detect defects such as open circuits, short circuits and pattern deviations on printed circuit boards during manufacturing.
“AVI (Automated Visual Inspection)”	An automated inspection method that uses imaging systems and algorithms to detect surface defects and irregularities on printed circuit boards during production.
“Backplane PCB”	A specialized multilayer printed circuit board designed to interconnect multiple electronic modules and support high-speed signal transmission in complex systems such as servers and communication equipment.
“BMS (Battery Management System)”	An electronic system used to monitor and manage the charging, discharging and health status of battery packs, particularly in electric vehicles and energy-storage systems.
“CAM (Computer-Aided Manufacturing)”	Software tools used in PCB manufacturing to convert design data into manufacturing instructions, including tooling, drilling coordinates and process parameters.
“CCL (Copper-Clad Laminate)”	A base material consisting of insulating substrate laminated with copper foil, widely used as the core material in the fabrication of printed circuit boards.
“Copper Foil”	A thin sheet of copper laminated onto insulating substrates in PCB manufacturing, forming the conductive layer that carries electrical signals.
“Desmear”	A manufacturing process used to remove resin residue and debris from drilled holes in a PCB, ensuring reliable electrical connection during subsequent plating processes.

GLOSSARY OF TECHNICAL TERMS

“Electroless Copper Plating”	A chemical deposition process that forms a conductive copper layer on non-conductive surfaces inside drilled holes before electroplating.
“Electroplating”	A process using the principle of electrolysis, a uniform coating is formed on the surface of an existing workpiece (such as copper) by applying direct current, which reduces and deposits metal ions from the plating solution onto the cathode.
“ENIG (Electroless Nickel Immersion Gold)”	A surface finishing process in which a layer of nickel and a thin layer of gold are deposited on PCB pads to enhance corrosion resistance, solderability and electrical performance.
“ERP (Enterprise Resource Planning)”	An integrated information-management system used to manage business operations such as procurement, production planning, inventory control, finance and customer orders.
“Etching”	A chemical process used to remove unwanted copper from a PCB panel after imaging, leaving behind the designed circuit patterns.
“FR-4”	A widely used flame-retardant fiberglass-reinforced epoxy laminate material serving as the standard substrate for many PCB applications.
“HDI (High-Density Interconnect) PCB”	A type of printed circuit board featuring higher wiring density, microvias and fine-line routing, enabling compact electronic devices with increased functionality and miniaturization.
“HASL (Hot Air Solder Leveling)”	A PCB surface finishing technique in which molten solder is applied and leveled by hot air to provide solderable surfaces.
“Imaging”	A photolithography process in PCB manufacturing where circuit patterns are transferred from design files onto copper-clad panels using photoresist and exposure systems.
“IoT (Internet of Things)”	A network of interconnected electronic devices capable of collecting and exchanging data through internet connectivity.
“LDI (Laser Direct Imaging)”	A digital imaging technology that directly transfers circuit patterns onto PCB panels using laser exposure, improving accuracy and supporting fine-line designs.

GLOSSARY OF TECHNICAL TERMS

“Lamination”	A process in multilayer PCB fabrication where multiple circuit layers and insulating materials are bonded together under heat and pressure.
“MES (Manufacturing Execution System)”	A digital production-management system used to monitor, track and control manufacturing processes in real time.
“Microvia”	A very small drilled hole, typically formed by laser drilling, used in HDI PCBs to electrically connect adjacent circuit layers.
“Mil”	A unit of length in PCB design used to specify very small dimensions. 1 Mil equals approximately 25.4 micrometers.
“NEV (New Energy Vehicle)”	A vehicle powered by electricity, hydrogen fuel cells, or other non-traditional energy sources.
“OBC (On-Board Charger)”	A power electronics unit installed inside new energy vehicles that converts incoming alternating current into direct current to charge the high-voltage battery.
“OSP (Organic Solderability Preservative)”	A PCB surface finishing process that applies a thin organic coating to copper pads to prevent oxidation and maintain solderability.
“PCB (Printed Circuit Board)”	A board used to mechanically support and electrically connect electronic components using conductive pathways etched from copper sheets laminated onto a non-conductive substrate.
“Paneling (Panel Combination)”	A production method in which multiple PCB designs are combined into a single manufacturing panel to improve production efficiency and reduce setup losses.
“Pearl River Delta”	The Guangdong – Hong Kong – Macao Greater Bay area.
“PTH (Plated Through-Hole)”	A drilled hole in a PCB whose inner walls are plated with copper to create electrical connections between different layers.
“Prepreg (Pre-Impregnated)”	A partially cured resin-impregnated fiberglass material used as an insulating layer between conductive layers during multilayer PCB lamination.

GLOSSARY OF TECHNICAL TERMS

“Signal Integrity”	The measure of the quality and reliability of electrical signals transmitted through a PCB without distortion, interference or loss.
“Solder Mask”	A protective polymer coating applied to PCB surfaces to prevent solder bridges and protect copper traces from oxidation and environmental damage.
“Surface Finish”	The final coating applied to exposed copper pads on a PCB to improve solderability, corrosion resistance and electrical performance.
“Thermal Management”	A design and engineering techniques used to dissipate heat generated by electronic components to ensure stable operation and reliability.
“Thick-Copper PCB”	A type of printed circuit board manufactured with thicker copper layers to support high current-carrying capacity and improved heat dissipation.
“Ultra-Thin PCB”	A printed circuit board designed with extremely thin substrates to support lightweight and compact electronic devices.
“Via”	A plated hole in a PCB used to electrically connect conductive layers within the board.
“VCP (Vertical Continuous Plating)”	A plating technology used in PCB manufacturing that enables uniform copper deposition through continuous vertical plating processes.
“YRD (Yangtze River Delta)”	The Shanghai – Jiangsu – Zhejiang – Anhui area.