Systematic Approach for PCB Assembly and Manufacturing

Sreenivasa Sai Jayanthi[1]· Ravindra Vutukuru[2]
[1]*Flashline EMS Pvt Ltd, Hyderabad*
[2]*Gitam University, Bangalore*

Abstract - Printed circuit boards are the heart of electronic devices, facilitating the connection and integration of various components through soldering. These boards are designed using computer-aided design (CAD) systems, which provide the necessary layout and configuration for the PCB. To translate this design into a physical product, the CAD data needs to be transferred to a photolithographic computer-aided manufacturing (CAM) system. It's common for CAD and CAM systems to be developed by different companies, necessitating a standardized data exchange format for seamless transfer. One such format is ODB, which facilitates the smooth transition of CAD data to CAM systems. ODB has gained popularity due to its comprehensive nature and ability to encapsulate various aspects of PCB design, including component placement, routing, and manufacturing information. Once the bare PCB is manufactured, electronic components are mounted onto the board using SMT (Surface Mount Technology) placement equipment. This equipment precisely positions components onto the PCB according to the design specifications. Subsequently, the components are soldered onto the board using techniques such as wave soldering or reflow soldering, ensuring secure electrical and mechanical connections. This research indicates that the CAD-to-CAM data exchange process, coupled with precise component placement and soldering techniques, plays a crucial role in the successful realization of PCB designs and the subsequent assembly of electronic devices.

Keywords-Printed circuit board ·FactoryLogix CAM350 Design for Assembly. Surface Mount Technology

INTRODUCTION TO PRINTED CIRCUIT BOARDS (PCBS)

A printed circuit board (PCB) serves as a medium for connecting or wiring components in a circuit. PCBs are ubiquitous in nearly all electronic products. While they require additional design effort to layout the circuit, manufacturing and assembly processes can be automated. PCBs come in different configurations: single-sided (with one copper layer), double-sided (with two copper layers on both sides of one substrate layer), or multi-layer (with outer and inner layers of copper alternating with substrate layers). Multi-layer PCBs allow for higher component density, as circuit traces on inner layers utilize space between components. The popularity of multi-layer PCBs, especially those with more than four copper planes, rose alongside the adoption of surface mount technology.

Component mounting on PCBs involves two primary methods: "through hole" and "surface mount." Through-hole components have wire leads passing through the board and are soldered to traces on the other side. Surface mount components, on the other hand, are attached by their leads to copper trace’s on the same side of the board. Some PCBs may use a combination of both mounting methods. However, PCBs with exclusively through-hole mounted components are now uncommon. Surface mounting is preferred for smaller components like transistors, diodes, IC chips, resistors, and capacitors, while through-hole mounting may be reserved for larger components such as electrolytic capacitors and connectors.

The earliest PCBs utilized through-hole technology, where electronic components were mounted by leads inserted through holes on one side of the board and soldered onto copper trace’s on the other side. Boards could be single-sided, with an un-plated component side, or more compact double-sided boards, with components soldered on both sides[1]. Installation of through-hole parts with two axial leads, like resistors, capacitors, and diodes, involved bending the leads 90 degrees in the same direction, inserting the part into the board, soldering the leads, and trimming off the ends. This process could be done manually or by a wave soldering machine.

The use of through-hole technology increases board costs due to the need for accurate hole drilling and limits available routing area for signal traces on multi-layer boards. With the advent of surface mounting, smaller-sized SMD components became preferred wherever possible, relegating through-hole mounting to components unsuitable for surface mounting due to power requirements, mechanical limitations, or potential PCB damage from mechanical stress.

LIST OF FILES MANUFACTURERS GET FROM CUSTOMER AS REQUIREMENTS

Fabrication drawing

A PCB fabrication drawing is crucial for communicating the manufacturing requirements of a circuit board to the manufacturer. Here's a comprehensive list of what should be included in a PCB fabrication drawing:

1. Board Outline: This section outlines the external perimeter of the PCB layout, defining its shape and dimensions. It includes cut-outs, holes, different radii, and specific distances. All dimensions should be referenced from a designated origin point, which must correspond with the origin point in the NC drill data file. Drilled holes should be clearly indicated with symbols linking them to the drill chart.
2. Drill Chart: The drill chart provides information on plated and non-plated holes. It displays symbols representing hole positions and sizes, along with details such as the number of holes of a particular size, whether they are plated or not, and hole tolerances.
3. Title Block: This section contains essential information such as customer name, part number, revisions, fabrication number, and measurement units. While not mandatory, including a title block helps ensure clarity and organization.
4. Circuit Board Layer Stack-up: This section details the layer order, dielectric spacing, copper weight, material type, and total thickness of the stack-up. It may also specify controlled impedance requirements, allowing manufacturers to adjust parameters such as trace thickness, width, and spacing to achieve the desired impedance.
5. Additional PCB Fabrication Notes: These notes provide further details to facilitate the manufacturing process. They may include:
    - Controlled impedance requirements, specifying trace thickness, width, distance from reference planes, and dielectric constant adjustments.
    - Board manufacturing specifications, such as primary side designation, raw material standards, plating details, solder mask specifications (color, finish), and surface finish (e.g., ENIG).
    - Gold fingers, if applicable, should be mentioned.
    - Critical mechanical specifications related to hole positions.
    - Presence of via-in-pad/plugged via or implementation of blind/buried via.
    - Tolerances for board dimensions, etching, and plating (typically ±10%).

By including these components in a PCB fabrication drawing, designers can effectively communicate the necessary manufacturing requirements to ensure the accurate production of circuit boards.

Assembly drawing

PCB assembly drawings indeed play a critical role in the manufacturing and assembly process of printed circuit boards [2]. Here's why they are important:

1. Provide Clarity: PCB assembly drawings visually represent the layout of the printed circuit board and its components. They provide clear instructions on the placement of components, their orientation, and the routing of traces. This clarity helps assemblers understand how to correctly assemble the components, reducing the likelihood of errors and increasing efficiency.
2. Ensure Accuracy: PCB assembly drawings ensure accuracy in the assembly process by providing detailed information on the position of each component, its orientation, and the routing of traces. This helps in eliminating errors and ensures that the final product is manufactured to the required specifications.
3. Streamline the Manufacturing Process: By providing all the necessary information required for the assembly process, PCB assembly drawings streamline the manufacturing process. They include details such as component positions,

orientations, trace routing, and any special instructions. This makes it easier for assemblers to assemble the board, ultimately reducing assembly time.

4. Assist with Quality Control: PCB assembly drawings serve as critical tools for quality control. They offer a detailed view of the PCB and its components, making it easier to identify any errors or defects in the assembly process. By facilitating thorough inspection and testing, they help ensure that the final product meets the required quality standards.

   In summary, PCB assembly drawings are essential documents that provide detailed instructions for assembling electronic components on printed circuit boards. They contribute to the efficiency, accuracy, and quality of the manufacturing process, ultimately ensuring the production of reliable electronic products.

ODB data

Indeed, ODB (Open Database) is a proprietary CAD-to-CAM data exchange format widely utilized in the design and manufacture of electronic devices. Its primary function is to facilitate the seamless exchange of printed circuit board design information between various design and manufacturing tools, particularly those from different EDA/ECAD vendors.

Here are some advantages of using ODB files:

1. Widespread Import Compatibility: ODB files are widely supported by CAD and CAM vendors, making them a de facto standard in the industry. This broad support means that there are no significant restrictions on licensing for vendors, facilitating easier adoption and integration into existing workflows.
2. Comprehensive Design Data in a Single File: ODB files contain all the necessary design data for manufacturing, assembly, and circuit board testing within a single file. This consolidation streamlines the data exchange process and reduces the risk of errors or discrepancies between different files.
3. Reduced Communication-Based Delays: By providing a standardized format for data exchange, ODB files help minimize delays and errors that can occur during communication between design and manufacturing teams. This efficiency contributes to faster turnaround times and smoother production workflows.
4. Support for Automation: ODB files support automation throughout the manufacturing and assembly process. This automation capability can help improve efficiency, accuracy, and consistency in manufacturing operations, leading to increased productivity and cost savings.
5. Preference Among Contract Manufacturers: ODB has become the preferred standard for many contract manufacturers (CMs) due to its widespread support and comprehensive data format. Choosing ODB can help ensure compatibility and smooth collaboration with manufacturing partners.
6. Reduction of Supply Chain Risks: ODB files help mitigate supply chain risks associated with data transmission errors. By providing a standardized format and comprehensive data, ODB minimizes the likelihood of misinterpretation or loss of critical design information during transmission.
7. Inclusion of Design Parameters: ODB files allow for the definition of various design parameters such as fiducials, testing points, and other critical elements. This ensures that all necessary information for manufacturing and testing is included in the file, enhancing overall efficiency and accuracy.

In the fast-paced world of circuit board design and manufacturing, where time is of the essence, leveraging ODB files can significantly optimize and simplify the process, leading to improved productivity and quality outcomes.

Gerber data

Gerber files are ASCII vector format files that serve as a fundamental part of the PCB manufacturing process. Here's a breakdown of their key characteristics and uses:

1. Representation of PCB Layers: Gerber files contain information about each physical layer of the PCB design, including copper traces, vias, pads, solder mask, and silkscreen images. Each of these circuit board objects is represented by a flash or draw code, defined by a series of vector coordinates.
2. Generated by PCB Design Software: Gerber files are typically generated by the PCB design software being used. The process of generating

Gerber files may vary depending on the specific CAD tool being utilized. Despite this variation, Gerber files are universally recognized in the industry.

3. File Format: Gerber files are 274x format and do not require a specific identifying filename. However, they are commonly given extensions such as .gb or .gbr for easy recognition. These files contain detailed information about the PCB layout, including layer configuration and component placement.
4. Description of PCB Attributes: Gerber files describe and communicate various attributes of the PCB image, such as the number of copper layers, solder masks, silkscreen layers, and other relevant details. This comprehensive information ensures accurate reproduction of the PCB design during the manufacturing process.
5. Input for PCB Printing Devices: Gerber files serve as input files for PCB printing devices, including photo-plotters and Automated Optical Inspection (AOI) machines. These devices utilize Gerber data to print or compare circuit board images, ensuring precise replication of the PCB design.

Overall, Gerber files play a crucial role in translating the details of a PCB design into physical properties during the manufacturing process. Their standardized format and comprehensive information make them essential for ensuring accuracy and consistency in PCB fabrication.

Bill of materials

A bill of materials (BOM) is an essential document in the assembly process of a circuit board, providing a detailed list of all the components and materials required for assembly. It serves as a reference for board assembly and communication between the customer and the assembly unit. Here are the key components typically included in a PCB BOM:

1. Item: Each component is listed as a separate item in the BOM, allowing for easy identification and tracking.
2. Quantity: The quantity column specifies the number of each component required for assembly.
3. Part Description/Value: This column describes the component or its value, providing necessary information for assembly.
4. Reference Designator: The reference designator indicates where each component should be placed on the circuit board, helping ensure accurate assembly.
5. DNI (Do Not Install)/DNP (Do Not Populate): These fields specify whether certain components should or should not be populated during assembly, based on design requirements.
6. Manufacturer Name: The name of the manufacturer of each component is listed to ensure sourcing from reputable sources.
7. Manufacturer Part Number (MPN): The unique part number assigned by the manufacturer to each component, facilitating accurate identification and sourcing.
8. Vendor Name: The name of the vendor from whom the component will be procured.
9. Vendor Part Number (VPN): The part number assigned by the vendor for the component, aiding in procurement and inventory management.
10. Footprint Type: This column specifies the footprint or physical layout of each component on the PCB, ensuring proper placement during assembly.
11. Alternate Part Number (Optional): Optionally, alternate part numbers may be included to provide flexibility in component selection and sourcing.

The BOM serves as a comprehensive guide for assembly, ensuring that the correct components are procured, placed, and soldered onto the circuit board. Additionally, it helps designers estimate the cost of manufacturing by providing information on component quantities and prices. Overall, the BOM plays a crucial role in ensuring efficient and accurate assembly of circuit boards.

Schematic

PCB schematic serves as a visual representation of the circuit design, illustrating the functionality and connectivity between various components. Here's an overview of its key aspects:

1. Two-dimensional Design: A PCB schematic is typically presented in a two-dimensional format, providing a clear depiction of the circuit layout and connections. It allows designers to visualize how components are interconnected without delving into the physical layout details.
2. Functionality Representation: The schematic outlines the intended functionality of the circuit, showcasing how different components work together to achieve specific tasks or operations.

This representation helps designers and engineers understand the purpose and operation of the circuit.

3. Component Symbols: Components in the schematic are represented using standardized symbols or customized symbols tailored to the specific application or design requirements. These symbols simplify the visualization of complex circuits and ensure consistency across designs.
4. Detailing Circuit Connections: The schematic details the connections between components using nets or traces, illustrating the flow of electrical signals throughout the circuit. Clear and concise line representations indicate the pathways through which signals travel, aiding in understanding circuit functionality.
5. First Step in PCB Design: The schematic serves as the initial stage in the PCB design process, providing a conceptual framework for the layout and arrangement of components on the circuit board. It lays the foundation for further development and refinement of the design.
6. Customization: While standardized symbols are commonly used in schematics, designers may opt for customized symbols to better represent specific components or circuit configurations. Custom symbols can enhance clarity and understanding, especially in complex or specialized designs.

Overall, a PCB schematic plays a crucial role in the design and development of electronic circuits, serving as a visual roadmap for designers to conceptualize, analyze, and refine circuit designs before moving on to the PCB layout phase.

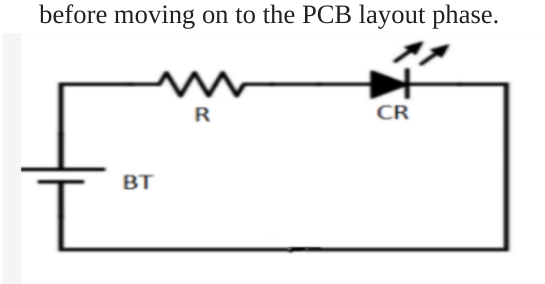
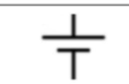


Fig. a: Basic schematic diagram

The circuit comprises three components: the battery, the resistor, and the LED. These components are interconnected via nets/traces. Each component is represented by a symbol, each with its unique attributes. For instance, a resistor may include attributes such as the reference designator, resistance value, size, symbol representation, voltage rating, wattage, and footprint. Similarly, the battery and LED will possess their own set of attributes.

Below is a table illustrating the names, symbols, and corresponding reference designators utilized in the circuit

Name of the component	Symbol	Reference designator
Battery		BT
Resistor		R
Light emitting diode		CR

hematic symbols

Where and How do manufacturers use the file gathered from the Customer

In today's intricate PCB designs, thorough verification is essential before transferring them to the fabricator to ensure successful and timely board manufacturing. Problems during fabrication can significantly impact product schedules, lead to costly design rework, and necessitate modifications that may compromise design integrity. By inspecting, preparing, and validating PCB designs before manufacturing, efficiency increases, the risk of design rework decreases, and electronic products can be successfully built faster and at lower cost.

To automate the PCB CAM engineering process, we use CAM350 which optimizes design files for fabrication, managing each operation efficiently. By streamlining data input, analysis, testing, milling, drilling, and bare-board production, it facilitates smooth transition into high-volume production. Any complications during fabrication can be mitigated with CAM350, safeguarding profitability. Additionally, CAM350 performs Design Rule Checks (DRC) to ensure Gerber layers comply with design rules and fabricator capabilities.

CAM engineers benefit from CAM350's ability to accurately prepare, optimize, and process design data[3]. It conducts in-depth analysis to identify manufacturing violations, generates optimized mill and drill files, prepares test data, and devises panelization strategies, all contributing to an automated, highly effective process.

Design for Assembly (DFA)

Design for assembly (DFA) is a structured process aimed at ensuring that components are accurately assembled onto a PCB without any malfunctions, with a secondary goal of minimizing assembly costs by simplifying the design. Here are some key considerations and guidelines for implementing DFA[4]

Component Availability and Compatibility: Ensure that components listed in the BOM are readily available and not obsolete or end-of-life products. Verify that the components' manufacturing part numbers (MPN) matches the footprint of the PCB.

Component Placement and Spacing: Check component placement, sizes, and distances to ensure compatibility with assembly manufacturing processes. To determine suitable soldering techniques and thermal dissipation requirements, adhere to component spacing guidelines.

Footprint Design and Pad Sizes: The component footprint should match the manufacturer's datasheet specifications. Consider pin pitch, hole sizes, pad sizes, and solder mask bridge sizes to ensure proper soldering and component attachment.

Solder Mask Clearances: Define sufficient solder mask clearances to prevent the formation of solder bridges between surface features such as copper traces.

Part-to-Part Spacing: Maintain adequate spacing between components to prevent solder bridging and facilitate rework processes. Discrete components like capacitors and resistors should have a minimum spacing of 10 mils between them, with an optimal distance of 30 mils.

Part-to-Edge Spacing: Provide sufficient spacing between components and the board edge to prevent damage during the depanelization process. Consider the placement of hold-down clamps to avoid overlapping with SMT components during solder application.

Part-to-Hole Spacing: Determine the minimum spacing required between component bodies/pads and mounting holes to prevent mechanical interference.

Component Clearance: Define clearances around components to accommodate assembly processes, rework, and mating connectors. Maintain specified clearances for different component types, including BGA devices, small packages, connectors, and crystals.

Component Orientation: To facilitate routing and ensure error-free soldering during assembly, align similar components in the same direction.

By following these DFA principles and guidelines, manufacturers can optimize component placement, streamline the assembly process, reduce downtime, and improve the overall quality and reliability of PCB assemblies.

Assembly Process

To prepare products or assemblies for efficient and error-free manufacturing, we use the FactoryLogix New Product Introduction (NPI) client application which offers recognition for CAD designs, Bill of Materials (BOM), and manufacturing processes, significantly reducing the time required from research and development to manufacturing [5].

When setting up a new part or assembly, we need to streamline complex BOM information into a more organized and usable format, set Gerber options which is utilized to format an aperture file (if loaded) to define the shapes of the pads. The CAD Controls tools allow users to set the PCB datum (origin), align Gerber layers with ECAD, rotate the board, display polarity, and assign Pin 1 as required. Fiducial markers on the PCB are critical for accurate alignment during manufacturing processes. The Footprints facilitates adjustments to the visual representation of each component on the board like overall outline or shape of the component footprint, adjustments to the specific area of the component used for colour coding, changes to the 0,0 (origin) point of a part. Assignment process informs about the specific parts to be populated by operations within the entire process flow.

Programming SPI (Solder Paste Inspection)

Solder Paste Inspection (SPI) plays a crucial role in verifying the solder paste deposits on bare PCB boards, making it a vital component of the surface mount assembly process. Detecting defects early on is key, as correcting them post-reflow can be significantly more expensive. The rule of thumb states that identifying a fault after reflow costs ten times the amount to rework than if detected before reflow; furthermore, identifying a fault after testing incurs an additional tenfold rework cost. To ensure the desired amount of solder paste is deposited accurately and consistently, a reliable inspection method like SPI is essential. SPI not only checks for solder paste area coverage and shorts but also accurately measures the shape and volume of solder paste deposits. By thoroughly inspecting every printed circuit board (PCB) before component pick-and-place, SPI helps monitor solder paste printing quality, reduces errors in printing, and ensures high-quality component placement.

Statistics suggest that around 80% of PCB assembly defects stem from improper solder paste printing, highlighting the importance of SPI in defect prevention. Using CCD cameras, SPI systems capture images of the printed solder paste, enabling measurement of alignment and volume. If defects are detected, boards can be removed from the production line, and solder paste can be reapplied until quality standards are met. Additionally, SPI helps engineers identify and rectify setup problems in solder paste printing. By facilitating defect-free PCB assemblies and reducing SMT costs, SPI significantly contributes to efficiency, especially in high-volume assembly projects. Implementing SPI not only ensures error-free solder paste printing but also helps shorten lead times in PCBA projects, making it a highly recommended process in surface mount assembly.

Automated Optical Inspection (AOI)

Automated Optical Inspection (AOI) is a sophisticated automated visual inspection system utilized in surface mount technology (SMT) assembly processes. It employs high-definition cameras to capture images of PCB boards after solder paste printing but before reflow soldering. Using computer vision technology, AOI conducts thorough 100% inspections of components and provides two types of data:

Attribute data: This includes information about missing, misplaced, or incorrectly oriented components.

Variable positional data: This data pertains to the measured position of components relative to their expected X/Y coordinates and placement.

Prior to the adoption of AOI technology, SMT manufacturers relied on manual visual inspections conducted by operators using magnifying glasses or microscopes. However, with the advancement of AOI technology, manual inspections have become less practical. AOI operates on optical principles, integrating photo acquisition and software analysis to detect and process defects encountered during SMT assembly manufacturing. AOI inspections are typically conducted both before and after reflow soldering, as this is when defect correction costs are lowest and efficiency is highest.

Shipping and Labelling Standards

All completed PCB boards (if size permits) must be labelled upon acceptance for shipment. The labeling of finished PCB assemblies should include, at a minimum, the following information:

- The part number of the finished PCB assembly.
- The current revision of the PCB assembly.
- A date code in the format "AWWYY," where "A" denotes the prefix followed by the week and year.
- A unique serial number.
- Vendors may include additional information such as lot codes, work orders, barcodes, etc., on these labels or use separate labels for such purposes.

Note: If the PCB assembly includes a battery, an externally conductive coated static shield bag must be used.

Incoming Inspection

The incoming inspection process shall inspect the following:

- Non-reference dimensions, such as board outlines, mounting hole locations, and diameters, as specified on the PCB assembly drawing per the Bill of Materials (BOM).
- Configurable components, such as DIP switches and jumpers, to ensure they are in the correct position as specified on the PCB assembly drawing per the BOM.
- Labeling applied according to the Final Labeling section of this document.
- The PCB board being shipped inside a static shield bag.
- Encouragement for including a 2 or 3 character supplier abbreviation in the labeling to simplify supplier identification.

Additionally, each individually bagged PCB should have the bag labeled with the part number and revision as established by the Purchase Order.

Key areas of PCB’s:

Consumer Electronics: PCBs are extensively used in consumer electronics such as smartphones, tablets, laptops, TVs, gaming consoles, and audio systems. They provide the necessary platform for integrating complex electronic components in compact and efficient designs.

Automotive Industry: PCBs play a crucial role in modern vehicles, controlling various electronic systems including engine management, infotainment, navigation, safety features, and advanced driver-assistance systems (ADAS).

Industrial Equipment: PCBs are integral to industrial machinery and equipment for automation, process control, monitoring, and data acquisition. They enable precise control and communication in manufacturing, robotics, and instrumentation applications.

Medical Devices: PCBs are used in medical equipment for diagnostics, imaging, patient monitoring, and treatment. They ensure the accuracy, reliability, and functionality of devices such as MRI machines, ultrasound systems, patient monitors, and medical implants.

Semiconductor Industry: PCBs play a crucial role in various aspects of semiconductor manufacturing, including testing, prototyping, burn-in, wafer handling, metrology, inspection, control, and communication. Their versatility and reliability contribute to the efficiency, accuracy, and quality of semiconductor production processes.

CONCLUSION

The smooth transfer of CAD data to CAM systems via standardized formats such as ODB is essential for streamlining the manufacturing of printed circuit boards (PCBs). This integrated process, complemented by accurate component placement facilitated by Surface Mount Technology (SMT) equipment and careful soldering techniques, guarantees the actualization of PCB designs and the assembly of electronic devices that meet high standards of functionality and reliability. The synergy between CAD, CAM, and manufacturing processes highlights the necessity of a harmonized workflow in the electronics sector, which is fundamental for achieving successful production outcomes in advanced electronic systems.

REFERENCE

[1] https://en.wikipedia.org/wiki/Printed_circuit_board
[2] https://www.acceleratedassemblies.com/blog/pcb-assembly-drawing--whats-and-whys-answered
[3] https://docplayer.net/21396799-Introducing-cam350-a-complete-pcb-fabrication-flow-for-both-pcb-designers-and-pcb-fabricators.html
[4] https://www.protoexpress.com/kb/dfa-rules/
[5] https://docs.aiscorp.com/fd/de/product-documentation/using-npi/create-an-assembly-using-a-3d-cad-file?scroll-translations:language-key=en