CHAPTER

PRINTED CIRCUIT BOARD (PCB): DESIGN AND TEST

4

CONTENTS

4.1 Introduction 82
4.2 Evolution of PCB and Components 83
4.2.1 Timeline of PCBs 83
4.2.2 Modern PCB Components 86
4.3 PCB Life Cycle 87
4.3.1 PCB Designer 87
4.3.2 Design House 88
4.3.3 Fabrication House 88
4.3.4 Current Business Model 89
4.3.5 Component Research and Selection 90
4.3.6 Schematic Capture 90
4.3.7 Simulation 91
4.3.8 Board Layout 91
4.3.9 Prototype Test 92
4.3.10 Best Practices in Overall Design Flow 92
4.4 PCB Assembly Process 92
4.4.1 Through-Hole Technology 93
4.4.2 Surface Mount Technology 93
4.4.3 PCB Complexity Trend and Security Implications 94
4.5 PCB Design Verification 95
4.5.1 Overview of PCB Inspection and Testing 95
4.5.2 PCB Defects 96
4.5.3 PCB Inspection 97
4.5.3.1 Automated Optical Inspection (AOI) 97
4.5.3.2 X-Ray Inspection 98
4.5.4 PCB Testing 98
4.5.4.1 In-Circuit Test (ICT) 98
4.5.4.2 Functional Test (FCT) 100
4.5.4.3 JTAG Boundary Scan Tests 100
4.6 Hands-on Experiment: Reverse Engineering Attacks 102
4.6.1 Objective 102
4.6.2 Method 102
4.6.3 Learning Outcome 102
4.6.4 Advanced Options 102
4.7 Exercises 102
4.7.1 True/False Questions 102
4.7.2 Short-Answer Type Questions 103

Hardware Security. https://doi.org/10.1016/B978-0-12-812477-2.00009-5

4.7.3 Long-Answer Type Questions 103
References... 104

4.1 INTRODUCTION

Printed Circuit Boards (PCBs) can be defined as rugged nonconductive boards built on substrate-based structure as shown in Fig. 4.1. The PCBs are mainly used to provide electrical connection and mechanical support to the electrical components of a circuit. They are prevalent in electronic devices and can be easily identified as the green-colored board in most cases. Based on the design specifications and requirements, many active (for example, operational amplifiers and batteries) and passive components (such as inductors, resistors, and capacitors) are mounted on the PCBs to match the form factor of the final design. Form factor can be defined as a feature of any hardware design that specifies the size, shape, and other relevant physical properties of the PCB in its entirety. While determining a form factor of a PCB design, aspects such as chassis, mounting schemes, and board configurations are taken into consideration. The connection among the components on a PCB are established with copper interconnects (routes), which act as the pathway for the electrical signals.

An Austrian engineer named Paul Eisler was the first to develop PCBs during the time of World War II. His patented methodologies for PCB etching process, various mechanisms of interconnect routing, and employment of electrical conduit in the boards are put to practice for decades [6]. Since its first development, PCB designs have significantly evolved over time. Modern-day PCBs largely vary in complexity, starting from single layer PCBs to complex designs with as many as 20 to 30 layers with hidden vias and embedded components [18]. PCB vias can be defined as vertical interconnect accesses for establishing the electrical connection through one or more adjacent layers of the circuit board.

PCBs play a vital role in area, power, performance, reliability, and security of a computing system. The PCB design and test process should consider these parameters. This chapter provides an overview of the PCBs with a highlight on current practices of design and test. It discusses the electrical com-

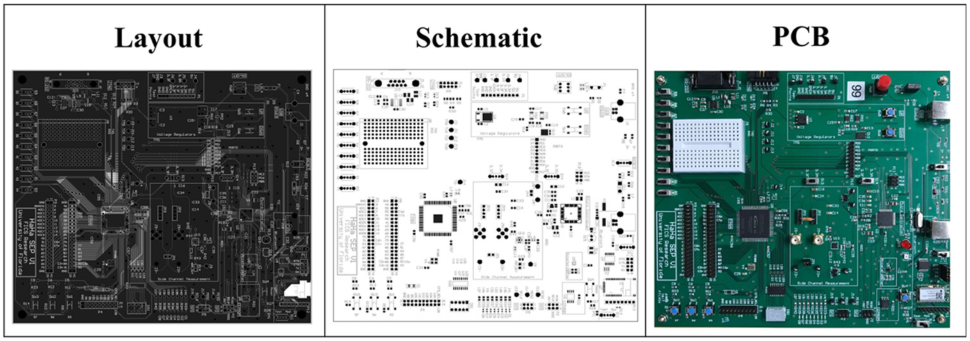


FIGURE 4.1

Modern PCBs are very complex with multiple layers and several components arranged in a compact manner to minimize the overall size. A modern PCB in different forms, such as layout, schematic, and its final output is shown in this figure.

ponents used in a PCB and different types of boards available. It also presents a brief history of PCB evolution highlighting the changes in PCB design with technological progress. The complete life cycle of modern PCB design is also depicted in the chapter with an illustrative description of the steps and parties involved in these steps.

4.2 EVOLUTION OF PCB AND COMPONENTS

The first patent for PCB was documented under the title "Printed Wire", and it can be traced back to early 1900s. In 1925, Charles Ducas filed the patent for printed wire technique to develop an electrical path on the surface of an insulated material. The concept was revolutionary as it demonstrated an efficient way of designing electric circuits without the complexity of vigorous wiring. Consequently, the design tremendously improved the overhead and performance results of conventional circuits. However, it was in 1943, when the first PCB came into production. Dr. Paul Eisler from Austria pioneered the development of the first operational PCB after World War II [6,24]. A short timeline of PCB evolution is given in the following subsections.

4.2.1 TIMELINE OF PCBs

Before the full-fledged production of PCBs in the electronics industry, the common norm was to implement point-to-point connections. The major drawbacks of this practice involved developing large sockets that required regular maintenance and replacement. Moreover, these components made the designs bulky and often led to design flaws. However, it was possible to address these issues by integrating the components on a PCB that could drastically improve the area, power, and performance. The major milestones for PCB design are described below (Fig. 4.2).

1920s: In the 1920s, commonly referred to as the roaring twenties, the material for PCBs varied significantly, ranging from Bakelite and Masonite to pieces of plain and thin wood. The practice was to drill holes in the board material and insert flat brass wires in the holes for completing the path of the circuit. Despite the lack of efficiency or craftsmanship in building PCBs at the earlier stages, the designs were able to meet the electrical requirements. The significant number of these boards were used in radios and gramophones. The invention by Charles Ducas, that is, exploiting conductive ink on an insulating material for electrical connectivity, was the highlight of this decade. This period introduced the idea of flat conductors, multilayer PCBs, and through-hole application in two layers, which was patented by a German Inventor Albert Hanson in 1903 [6,8].

1930s to 1940s: The application of proximity fuse in precision weapons of World War II accelerated the development and application of PCB in the late 30s and early to mid-40s. The purpose of employing a proximity fuse in the weapons was to acquire greater distance with higher accuracy. In 1943, Dr. Paul Eisler patented this method of developing PCBs. He proposed the use of copper foil on a nonconductive base, reinforced by glass. The initial application scope of the patent was radios and communication equipment. However, the application range grew over time in the later years. The production of double-sided PCBs with plated through holes in 1947 also extended the application range. This design addressed many limitations of previous designs by providing an efficient way of developing electronic circuits [1,3,6].

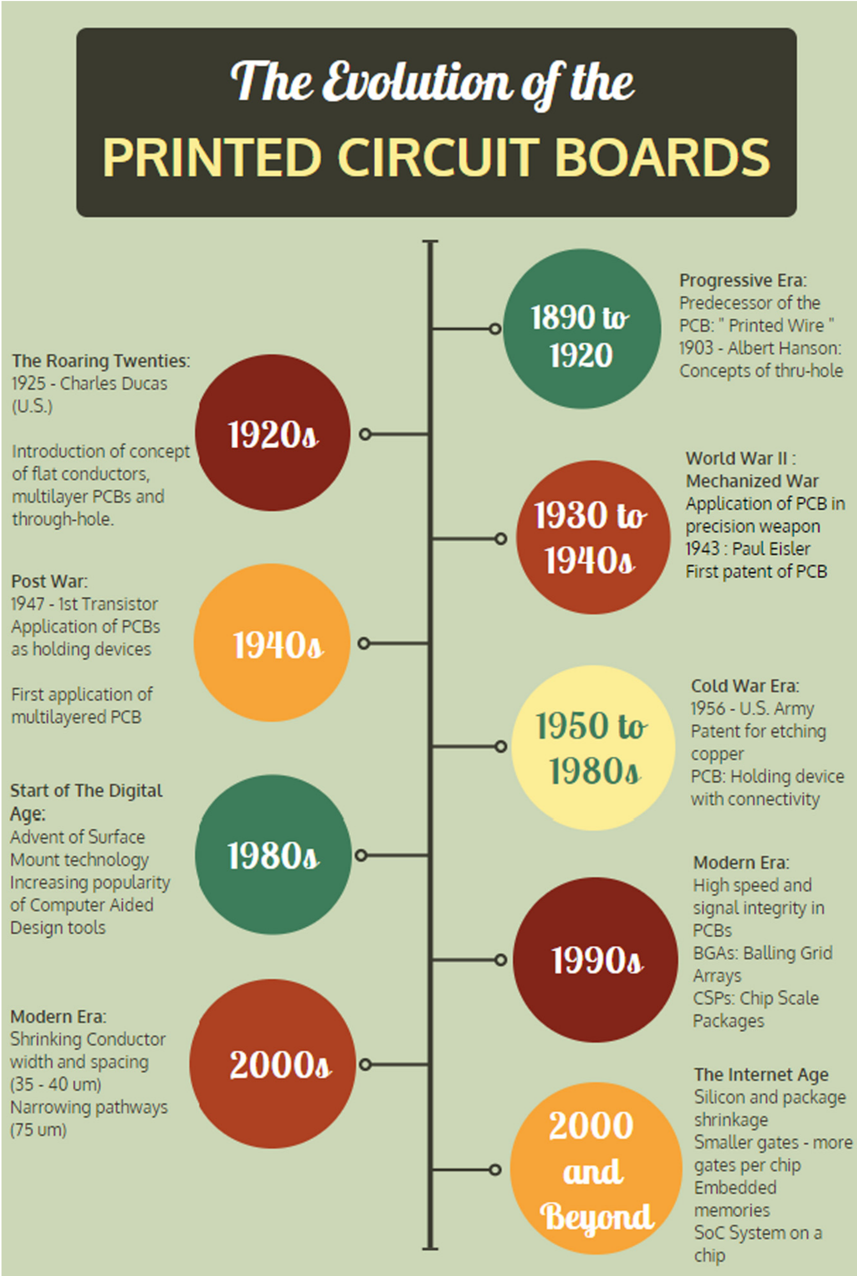


FIGURE 4.2

The evolution of printed circuit boards over time.

1950s to 1960s: Between the 1950s to 1960s, different variations of board materials were employed in PCB designs, including various kinds of resins and other compatible materials. At that time, the wiring was installed on one side of the boards, whereas the electrical components were inserted on the other side. PCBs at this time were adopted in many applications as it helped to eliminate the rigorous process of installing bulky wiring and greatly improved the performance of the circuit. A major stepping stone in the development of PCB was a patent titled “Process of Assembling Electrical Circuits”. It was filed to the U.S. Patent Office in 1956 by a small group of scientists from the U.S. Army. They introduced a technique for drawing the wiring pattern and photographing the pattern on a zinc plate. The zinc plate is used as a printing plate for circuit boards in offset printing press. The wires were printed on the copper foil using an acid-resistant ink and, later, they were etched by acid solutions. In 1957, The Institute of Printed Circuits (IPC) decided to form a coalition. The first general meeting of the organization took place in Chicago, Illinois in the same year [6,9].

1960s to 1970s: The concept of multi-layer PCBs came into production in 1960. The maximum number of layers in the manufactured board varied over a range from 2 to 5. The first batch of boards was manufactured with 4:1, red-and-blue line vellum method. This process helped with the hand taping of components and tracks. Afterwards, a precision method was employed to produce the 1:1 negative manufacturing film. The process enhanced the pipelining of the design cycle and helped with speedy production. It took about 2 hours of effort for an experienced designer to produce the layout of a PCB with an equivalent of 14-pin ICs mounted on the board. By the 70s, the circuitry and overall size of PCBs started getting significantly smaller than the previous versions. The hot air soldering methods came into regular practice in this period. Moreover, a Japanese practice of developing liquid photo imageable masks (LPIs), using a wide variety of aqueous material, was adopted by many PCB manufacturers. Eventually, this practice became the industry standard in the later years. RS-274-D, a machine-based formatting method for vector photoplotters was introduced by Gerber Scientific Inc. in this decade [6].

1980s: In the 1980s, the PCB producers started choosing the surface mounting technique for integrating components on a board over the through-hole method. The preference led to the subsequent reduction in the size of a PCB without posing any obstacle for PCB functionality. An improvement on the data formatting of RS-274-D was released in 1986. The new release, RS-274X, provided support for embedded aperture information and eliminated the necessity of external aperture definitions files [10].

1990s: The size and price of PCB boards started decreasing in the 90s, while the complexity of the boards was increasing due to multi-layer designs. The multi-layer construction, however, helped the progress by facilitating the incorporation of rigid and flexible PCBs in the design. The era of high density interconnect (HDI) began in 1995 through the use of micro-via technology in PCB production [6,7].

2000 to Present: The highlight of this decade involves a peak market value of 10 billion USD for the first time in United States. PCB fabrication industry and the ELIC (Every Layer Interconnect) process flow started from 2010. ELIC provides smaller pitches and eliminates the mechanical holes inside the board, which consumes space, thus increasing interconnect density. In ELIC process, a PCB supports HDI using several layers of copper-filled stacked in-pad microvias. During this period, enhanced usage

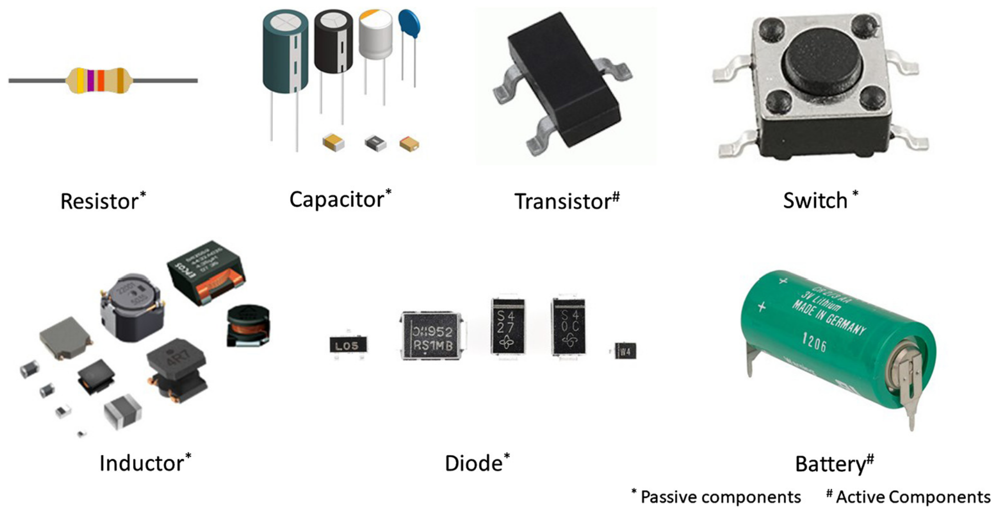


FIGURE 4.3

Various active and passive components used in PCBs.

of 3-D modeled boards and System on Chips aid the process of developing compact PCBs with greater performance. Consequently, these technology enablers keep the PCB design industry dynamic and rapidly adaptable in the future [5,6].

4.2.2 MODERN PCB COMPONENTS

Every PCB is comprised of various electronic components (Fig. 4.3). These components are typically industrial products manufactured individually in bulk with a wide range of values. They are provided with electronic terminals to build electronic circuits. Electronic components are packaged based on their type, functionality, and application. Common classification of electronic components is done depending on the energy source. The components, which act as energy sources, are called active components, whereas the components that need external sources of energy for working are called passive components [15].

In DC circuits, active components rely on energy sources like batteries. In some cases, they can introduce power into the circuits. In AC circuits, active components comprise of transistors and tunnel diodes. Passive components cannot introduce energy in the circuit. They are completely dependent on power from the AC circuit to which they are connected. These components have two electronic terminals. Some of the passive devices include resistors, inductors, and capacitors.
Some of the active components are described next:

- Battery: Batteries provide required voltage to the circuit; these are primarily used in DC circuits.
- Transistor: A transistor is used to amplify the charge; they are also used to switch electronic signals.

Some of the passive components are described next:

- Resistors: Resistors control the flow of current through them; they are also color-coded to facilitate recognizing their resistance.
- Capacitors: Capacitors store potential energy in the form of an electric field.
- Inductors: Inductors store electrical energy in the form of a magnetic field when a current passes through it.
- Diode: Diode conducts current only in one direction; they have very low resistance in the forward current path and large resistance to restrict it in opposite direction.
- Switches: Switches are used to block or change the direction of the current.

4.3 PCB LIFE CYCLE

Modern PCBs are built through a complex process consisting of number of stages. Multiple entities get involved in the PCB design and manufacturing process. A brief overview of the PCB life cycle is described in this section (Fig. 4.4).

4.3.1 PCB DESIGNER

The first entity in the PCB life cycle is the design engineer. The PCB designer goes through the basic steps of design flow, that is, part research and selection, schematic capture and simulation, board

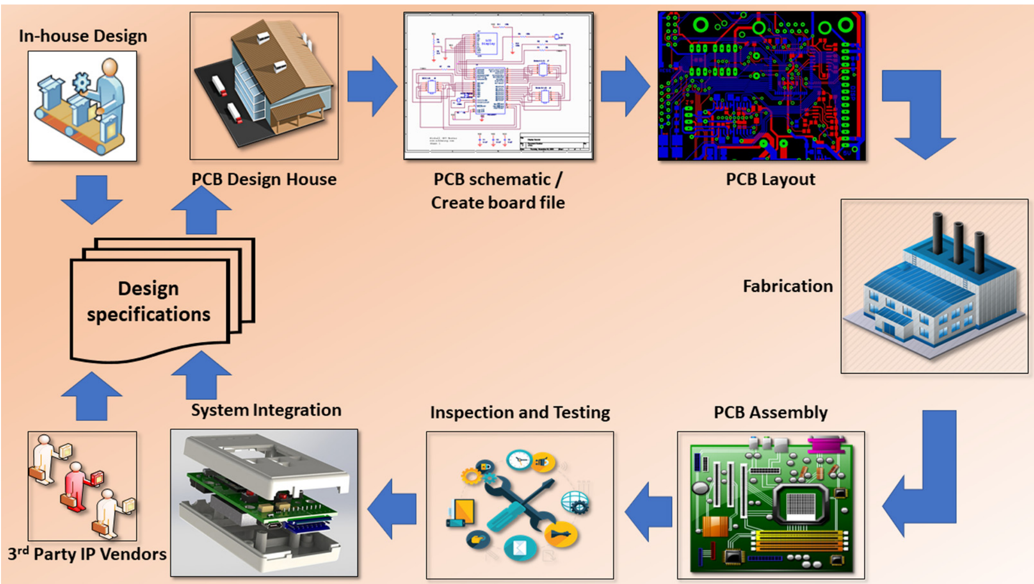


FIGURE 4.4

Detailed illustration of PCB life cycle.

layout, and verification and validation to finalize the complete process. Once the designer is satisfied with the final version of the design, the design is forwarded to the design house with exact design specifications.

4.3.2 DESIGN HOUSE

The design house of PCB obtains required design specifications and necessary design files from the design engineer. The design files can come from third-party vendors or in-house design engineers. Once the design files are acquired, the engineer at the design house starts creating the board files by analyzing the netlist. A netlist is a list of components that describe the connection among components in any specific design.

Once the netlist is finalized, the engineers in the design house create libraries for the components of the design. In this stage, the design engineers review and analyze the data sheet for each of the components. The key information retrieved from the datasheet include mechanical dimensions, package types, symbols, footprints, and pad stacks. As a good design practice, the components with mechanical constraints are placed first on the board in the component placement phase. Then, the placement of performance-critical components, such as microprocessors, video graphic arrays (VGAs), memory, and FPGA is performed, followed by the placement of other components, such as decoupling capacitors, passive elements, and test points [6].

After the placement of components, the nets or interconnect wires are routed on the board. The critical nets are given higher priority in the routing process. Afterward, the non-critical nets are either hand-routed or automatically routed using a design tool. Finally, the design is verified by performing a design rule check and eliminating the errors. The final step of PCB design in the design house is post-processing of the design. This step includes creating the Gerber and drill files with associated assembly diagrams.

4.3.3 FABRICATION HOUSE

The foundry or fabrication house is another key entity in the PCB life cycle. The final boards are fabricated in the fabrication house. Once the fabrication is done, the PCB assembly process begins. A detailed description of the common assembly techniques in current practice is provided later in this chapter. During the assembly process, the PCBs usually go through multiple steps, such as stencil printing and placement of components. Afterwards, the pick and place machines are used in the assembly technique line to select the boards and send them to the reflow oven. The manufacturing companies use several types of pick and place machines according to the design needs and requirements. The programmable pick and place machines offer high flexibility in terms of high-mix and low-volume products without causing much downtime in the process. As the next step of the design process, the manufactured boards proceed for inspection. The common inspection techniques are automated optical inspection (AOI) and X-ray inspection. The inspector verifies the physical integrity of the PCB design through automated machines. Once this is done, the power-up tests of the boards are performed. Commonly applied testing approaches include in-circuit, functional testing, and JTAG-based boundary scan testing. Each of the PCBs is tested rigorously to verify and validate the functionality of the circuit operation. Different simulation environments are generated to evaluate the performance of a PCB in the real-life scenario. In case of faulty boards, repairs are performed followed by testing that checks if

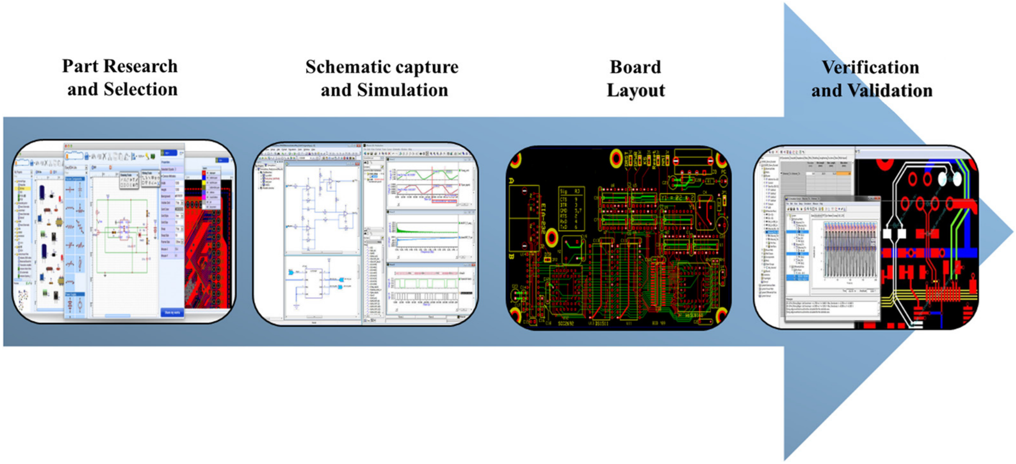


FIGURE 4.5

Basic steps of PCB design process.

they meet the performance targets. Furthermore, feedback is provided to the design engineers based on the analyzed faults in order to take necessary measures to prevent further disruptions [1].

4.3.4 CURRENT BUSINESS MODEL

Due to the global economic trend, current business models of PCB design and fabrication increasingly rely on extensive outsourcing. The manufacturers are more prone to integrating untrusted third-party vendors in the PCB design and fabrication steps for cost minimization. The cost benefit of outsourcing the PCB design/fabrication to a foreign country with economical labor is primary driving factor of PCB business model. Other factors behind adopting a global business model include time-to-market requirements, resource constraints, rapid growth, and collaboration. However, the security implications of such a global business model can be severe if necessary measures are not taken to ensure the trust and reliability of the product obtained from third-party entities. It is important for any PCB manufacturer to make judicious tradeoff between cost and efficiency benefits through outsourcing and associated security concerns [1,12,30]. Major steps of PCB design process are delineated next in this section. These steps are generally applicable to all PCBs designed across the world.

- Component Research and Selection
- Schematic Capture and Simulation
- Board Layout
- Verification and Validation

Figure 4.5 illustrates the steps of PCB design flow. In the current practice of PCB design flow, the schematic capture and simulation phase is well integrated with the board layout phase. The integration is usually made convenient by the usage of a single corporate toolchain. However, the starting point for any PCB design is performing part research and selection, and finalizing the design with prototype validation. Unfortunately, these two steps of the design flow are isolated and separated from the core stages. The separation often leads to major challenge, as every PCB design goes through a large number of iterations before reaching the final stage. Lack of integration in the iterative design process leads to improper use of task times. Hence, the goal of an efficient PCB design process is to remove the isolation between the initial and final stages, and integrate them with the entire flow.

4.3.5 COMPONENT RESEARCH AND SELECTION

The very first step of PCB design is the research and selection of the physical components. Components such as resistors, transistors, operational amplifiers, processors, memory, etc. act as the building blocks of the design. A significant task in the part research and selection phase is to investigate and evaluate each of the components and determine their role in the design topology. The design specifications and device performance information for each of these components can be found at numerous online repositories of datasheets. Datasheets usually contain important information, such as design considerations, operating points, and switching characteristics [11, 12].

Most manufacturers have the component datasheets available on their websites as well as web repositories, such as DatasheetCatalog.com. Obtaining different components and physically developing a breadboard prototype is often the most time-consuming step of the design process. This process involves purchasing different components and breadboarding those at the preliminary phase of design. Physical breadboarding is preferred by engineers who want to obtain a better understanding of the functionality of the components instead of simply going over the product datasheets.

However, the usage of a schematic capture and simulation tool can help designers avoid the initial expenses of purchasing the components and developing a physical breadboard. There are many off-the-shelf commercial tools that can generate the simulation environment and overall behavior of the circuit. For instance, a designer can select any of the open-source PCB design software (for example, TinyCAD, KiCAD, ExpressPCB, EasyEDA, and DesignSpark PCB) and select the required components from the database to design the circuit. Furthermore, the operation of the components can be visualized with the associated simulation models. Thus, the capture and simulation environment can help in creating the design and simulating the operation of the circuitry quickly. The omission of the physical implementation of a breadboard prototype reduces the design expenses, and time required for the selection procedure [28].

4.3.6 SCHEMATIC CAPTURE

The design process starts with the selection and placement of right components in the circuit. All the schematic generation tools provide a database of components from which the engineer chooses the desired component. Due to the enormity of the number of components, the tools offer categorized groups of each class. The classes are designed according to component type and functionality. The initial schematic design process can be summarized in the following steps: navigating the component database, selecting the desired part, and placing the part in the circuit with proper wiring. Some tools provide a wireless working environment to reduce the efforts of continuous switching between the placement of parts and wiring modes [29]. The circuit schematic is usually represented in the form of symbols in ASCII and DIN standard and connected to each other by wires.

4.3.7 SIMULATION

Once the schematic is prepared, a PCB designer typically utilizes a simulation environment, for example, SPICE or XSPICE, to simulate the behavior of the design and evaluate the effects of different circuit components and signals. Simulation is a critical part of the design as it captures the behavior and performance of the circuit without building it physically. By definition, simulation is a mathematical representation that depicts the functionality of a real circuit component under varying conditions, including specific operating voltages and temperatures. Through simulation, it is possible to test the design topology and find out if any modification is required before building the prototype. Thus, simulation saves time, effort, and money by providing the opportunity to quickly fix the flaws of any PCB design before reaching the prototype stage.

The depiction of the real-life behavior of components in a simulation environment depends on the accuracy of device models. Thus, it is important to develop device models as accurate as possible as it has great impact on the designer's analysis of the PCB design. Many models of devices, such as BJTs, FETs, and operational amplifiers are designed with high accuracy to demonstrate the real-world behavior via simulation. The precision percentage of the results is finely tuned with advanced features, for instance, parasitic effects and complex behaviors of the devices. For effective simulation of circuits, PCB design engineers are expected to have good knowledge of SPICE or any of its variants. It is basically a text-based language used to simulate circuits effectively.

4.3.8 BOARD LAYOUT

The third step of any PCB design flow is building a robust prototype. The robustness of the design, however, will depend on the efficacy of schematic capturing and validity testing via proper simulation. The prototype development is required to evaluate the design for real-life performance. The development process requires a CAD environment that supports the format of an exact physical dimension of components used in the design. The CAD tool for generating layout outputs the final design in the form of Gerber files. The Gerber files are later utilized by the manufacturers in the industry to produce the actual physical boards. The ICs and other components are placed on the PCB during the layout design phase. Furthermore, the components are connected to each other via current conducting conduits, i.e., copper traces (routes). As a final step of this phase, the form factor of the respective PCB is calculated. Determining the right form factor of the board is crucial as it ensures that the design will fit inside the physical environment in which the PCB will be used.

Many advanced CAD tools are available for automated placement and routing. However, it is important to use these tools judiciously. For instance, designs operating at critical conditions or under tight energy or performance constraint, require a high level of scrutiny. Generally use of manual methods is the best approach for performance-critical parts of a PCB design. For lesser critical parts of the design, however, the application of automated tools, such as auto-router, is an accepted norm.

4.3.9 PROTOTYPE TEST

The final steps of PCB design flow are prototype development, followed by manufacturing test. The purpose of prototype testing is to verify if the design meets the intended or desired specifications. On the other hand, the manufacturing tests evaluate the appropriate standards of the final product being deployed in field. Any design flaw discovered at a later stage leads to additional expenses in terms of time and money. Therefore, iterative approaches in the schematic capture and simulation phase greatly help to produce a flawless product up to the final stages.

While performing the prototype tests, the real-life operation of the PCB is analyzed and the results are compared with original design specification. Through these tests, a test engineer verifies the final product with the design specs obtained from the design engineer and evaluates the performance of the end product. Finally, based on the results, the engineers conclude on whether to forward the product to market or re-evaluate for further performance improvement.

4.3.10 BEST PRACTICES IN OVERALL DESIGN FLOW

As an engineer moves through the design flow, from part selection to layout, integration becomes paramount in streamlining the creation of PCBs, reduce iterations, correct common errors, and have a faster transition to market. This integration, as discussed in this chapter, breaks down the walls that generally disrupts the traditional design flow, and is a best practice in board-level design.

It merges the part selection, schematic capture, simulation, board layout, and design validation in a single thread. Existing schematic capture tools can be exploited to test and validate the PCB designs for any topology. Moreover, features like a large repository of components and wiring can expedite the design process. An interactive simulation environment and tools for advanced analysis and virtual prototyping are the key components that allow a designer to reduce the design errors and develop an improved simulation model for board level designs. Finally, a designer can exploit the manual and automatic layout tools and effectively produce Gerber data for creating PCB prototypes [13,14,29].

4.4 PCB ASSEMBLY PROCESS

The primary steps of PCB assembly are the placement of electrical components on the board and soldering the components to the substrate. Whereas these steps are typically followed for hand soldering components into the through holes, the assembly process of modern PCBs is fairly complex. It is a multi-step process that offers the flexibility to incorporate various package types and a diverse range of substrates and materials. The current practice also facilitates adaptability of PCB designs in terms of reliability and defect level threshold with varying amount of production quantities. In general, the steps listed below describe the PCB assembly process with more accuracy:

- Preparing the component and substrate material on the board
- Applying flux and solder
- Melting the solder to complete the connections
- Cleaning the soldering as a part of post-processing
- Inspecting and testing the final product

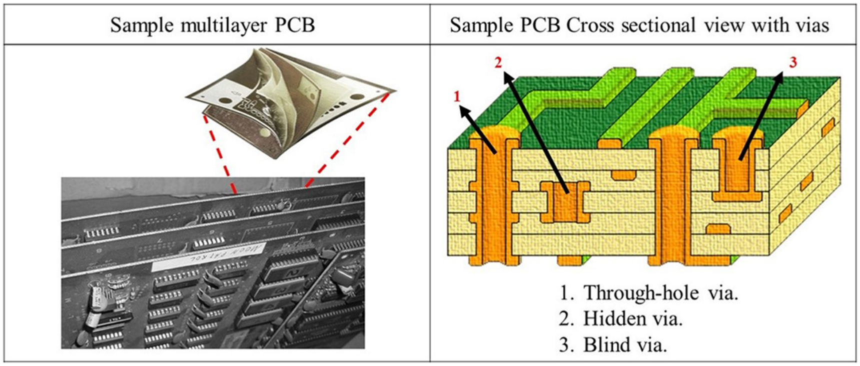


FIGURE 4.6

A sample multilayer PCB and its cross-sectional view with different kinds of vias.

In many instances, some of these steps are merged together or omitted, based on the product requirements. It is the responsibility of the manufacturer to adopt an assembly process that encompasses all the crucial steps. The assembly process of the PCB can be broadly categorized into through-hole technology and Surface Mount Technology (SMT). Each of these technologies offers various levels of automation in the assembly process, based on the equipment resources. The degree of automation usually depends on the design of the boards, expenditure of the equipment, bill of materials, and the manufacturing costs.

4.4.1 THROUGH-HOLE TECHNOLOGY

Through-hole technology can be defined as the assembly process in which the lead of electrical components are inserted into the holes on the boards and mounted through soldering process (Fig. 4.6). This technology was automated by the application of wave soldering. A major disadvantage of this technique is the low density of assembly. It poses a great obstacle to the miniaturization of the design and flexibility of functionality, due to the requirement of additional board space for large components and big holes. An advantage of this technology is lower assembly cost compared to alternative approaches. Even in case of fully automated assembly process that incorporates wave soldering, selective soldering, or past-in-hole/reflow, the overall equipment and setup cost of through-hole techniques is usually lower than surface-mount techniques.

4.4.2 SURFACE MOUNT TECHNOLOGY

The components can be placed on one side or both sides of a board. The initial motivation for developing SMT was facilitating the assembling of hybrid microcircuits (HMCs) into ceramic substrates. Later, SMTs were developed for laminate substrates as well. The major advantage of SMT lies in increased board density with a higher number of small components. In SMT, small vias are employed to connect sides and internal layers as the replacement of conventional large holes. SMT also helps with miniaturization of PCBs and improvement of performance by aiding finer traces and shortened

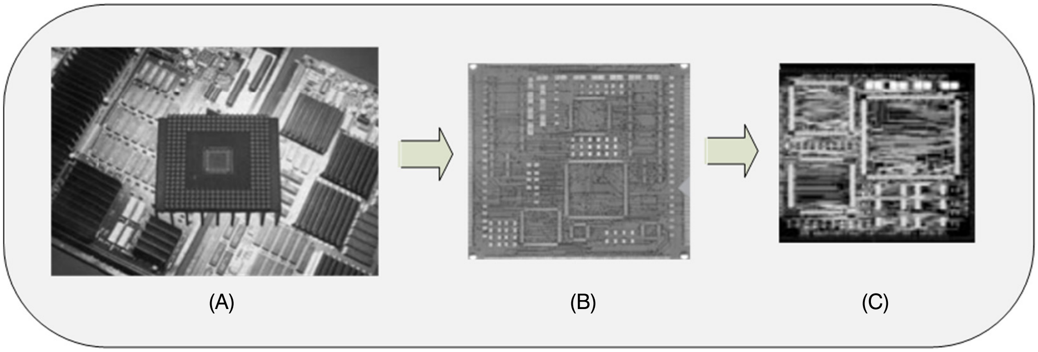


FIGURE 4.7

An illustrative example of a CPU with a different component, assembly, and PCB technologies over time. (A) First RISC processor developed in 1986: 14-layer through-hole board with a surface area of 128 square inches. (B) The same processor in 1991: 10-layer board on a surface area of 16 square inches (built with SMT). (C) The processor in 1995: HDI board with a surface area of 4 square inches (includes buried and blind vias and sequential build-up microvias).

components in the circuit. The common practice in SMT is to use smaller devices, such as resistor, capacitors, and inductors. SMT also facilitates the usage of embedded passive devices like resistors and capacitors placed inside the board laminate. Consequently, the embedded passive devices provide a larger surface area for the active components of a design.

4.4.3 PCB COMPLEXITY TREND AND SECURITY IMPLICATIONS

Shrinking size of the components, increasing complexity of a design that requires large number of components to be integrated into a PCB, and the advent of efficient packaging techniques, for instance, the ball grid array (BGA), system on chip (SoC), chip-scale packaging (CSP), and chip-on-board (COB) guided the progression of conventional PCB technologies to HDI era. Figure 4.7 shows an example of the changes in interconnect technology over time. The figure shows the evolution of a computer central processing unit (CPU) due to the changing trends in component technology, PCB technology, and assembly technology. It also shows the direction and rate of the changes of technologies that drive the PCB design trends over time [19–21].

The current drivers in packaging and interconnect technology selection are mainly the speed of operation, power consumption, thermal management, electronic interference, and system operating environment. The shift towards the miniaturization and portability of electronic devices is another factor that determines the PCB design and technology trend. However, with the rapid increase in PCB design complexity, there are rising concerns about the security aspects of modern PCBs in the design community. It is possible for any rogue entity to take advantage of today's complex and highly integrated PCB designs with 20 to 30 layers, hidden vias, and embedded passive components to tamper or insert additional malicious circuitry in the form of hardware Trojan. Current industry practices do not employ adequate security measures to deter such threats. There is a critical need to design PCBs that are resilient against these security threats while meeting the performance and other constraints. A detailed description of the security vulnerabilities and potential attack scenarios on modern-day PCB designs is provided in Chapter 11 of this book.

4.5 PCB DESIGN VERIFICATION

Despite the diversity of application areas, the requirements for unimpaired operation and high performance are common in every system built upon the PCBs. There are many instances of performance critical systems in which human lives are at stake. Consequently, it is of utmost importance that the PCBs should perform flawlessly. The inspection and testing process constitutes a significant part of any PCB life cycle. The challenges of PCB testing mostly arise from the complexity of dealing with PCBs containing hundreds of components and thousands of soldering connections. To overcome such obstacles, the PCB manufacturing companies incorporate a variety of inspections and testing methodologies to produce high-quality end products.

A taxonomy of PCB verification approaches is shown in Fig. 4.8. During the inspection and testing phases, the faulty boards are identified and considered for repair. The cyclic process of getting feedback on the manufactured board helps engineers to continuously improve the design through multiple iterations. Each PCB is inspected and tested by the manufacturer according to the design and performance specifications to ensure the maximum yield and reliability from the final product. Hence, inspection and testing are key stages in the PCB life cycle. In this section, a brief description of different PCB inspection and testing methods will be provided to help the reader get a better insight on inspection and testing processes suitable for any PCB.

4.5.1 OVERVIEW OF PCB INSPECTION AND TESTING

For a reasonably simple PCB consisting of few components and solder connections, manual visual inspection (MVI) might suffice for detecting the placement errors on the board or solder problems. However, the process of MVI is inherently limited to the flaws of human inspectors performing repetitive tasks. It is very likely that a human inspector may overlook defects in a design. Any unidentified defect in the manual inspection stage may cause serious flaws during system operation. To mitigate the errors introduced by human involvement in the inspection process, the inspection process has been

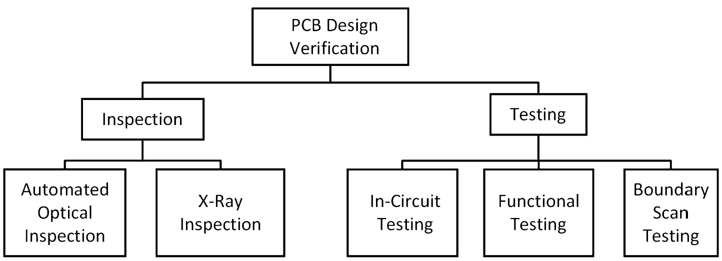


FIGURE 4.8

A taxonomy of PCB verification methodologies.

automated. Automated optical inspection (AOI) processes are integrated with pre-reflow, post-reflow, or both, to detect and pinpoint potential faults in a board. Pick-and-place machines with AOI capability are usually employed to check for misalignment and faulty components [26].

A major obstacle in the AOI process was posed by the advent of SMT. It enabled the integration of smaller components, novel microchip packages, and aided the development of complex multi-layered boards. Due to the increased density of PCBs, it is difficult to analyze the solder joints and chip packages with ball grid arrays (BGAs) that have connections inside the packages. The issues caused by the difficulty to view the layered contents of the complex PCBs is addressed by X-ray based inspection methodologies. The Automated X-ray inspection (AXI) helps the PCB inspectors analyze the multi-layered and double-sided dense PCBs with components and interconnect fabric.

The inspection process is followed by testing of PCBs. Whereas inspection techniques, such as AOI, checks for overall PCB construction quality and major flaws (e.g., missing component), detailed PCB testing is essential for high-level quality assurance. The commonly employed testing mechanisms are in-circuit tests (ICTs), functional tests (FCTs), and JTAG-based boundary scan tests. ICTs help to verify if a board and each of its components are performing according to the specifications. FCTs, on the other hand, are applied to conclude with a pass/fail decision on a PCB. Boundary scan tests are inspired by the limitations of conventional tests (that is, functional and in-circuit tests), such as failure to employ bed-of-nails fixtures on modern PCBs with double sided components and reduced spacing of interconnects. Boundary scan tests facilitate built-in test delivery system and provide standardized test ports and buses.

4.5.2 PCB DEFECTS

PCB boards can have a variety of defects due to improper design practices. The commonly found defects in the PCBs include misaligned components, incomplete solder connections, and short circuit caused by excess solders. A summary of the common defects of PCBs is presented in Table 4.1. The table also provides a classification of the defects based on the type, rate of occurrence, and relevance to soldering issues [16,17,25].

Table 4.1 An illustrative summary of defects commonly found in PCBs

Defect	Rate (%)	Type	Solder related
Open	25	Structural	Yes
Insufficient solder	18	Structural	Yes
Short	13	Structural	Yes
Missing electrical component	12	Structural	No
Misaligned component	8	Structural	Yes
Defective electrical component	8	Electrical	No
Wrong component	5	Electrical	No
Excess solder	3	Structural	Yes
Missing nonelectrical component	2	Structural	Yes
Wrong orientation	2	Electrical	No
Defective nonelectrical component	2	Structural	No

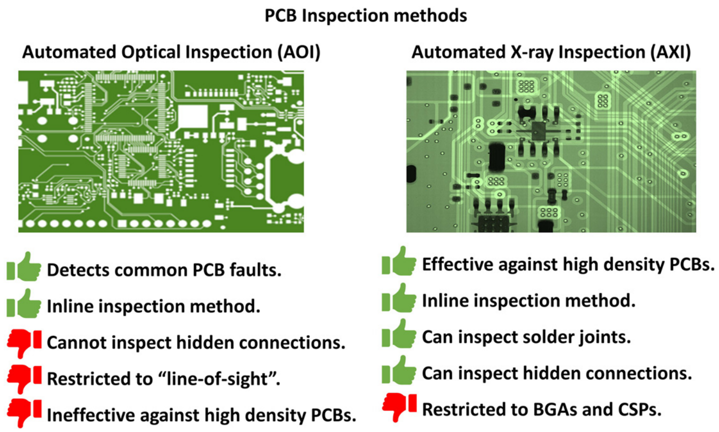


FIGURE 4.9

Comparison of PCB inspection methods.

4.5.3 PCB INSPECTION

The early detection of any fault in the PCB significantly reduces the repairing cost. Hence, a manufacturer incorporates inspection of the PCBs at various stages of the life cycle. In the subsequent sections, two PCB inspection methods, that is, automated optical inspection and automated X-ray inspection, which are commonly practiced in the industry (Fig. 4.9), are described in detail.

4.5.3.1 Automated Optical Inspection (AOI)

AOI is a process of visually inspecting the PCBs. The process involves the use of multiple high-definition cameras to capture images and videos of the board under test from multiple angles. The high-quality images are stitched together to form a large image file. Then, the resultant image is compared with the golden board (ideal board with exact design specifications) to figure out the discrepancies. AOI systems are employed to primarily find out physical defects. The type of defects can be scratches, strains, or nodules on the boards. Moreover, AOI can help detect issues, such as open and short circuits and thinning of soldering due to manufacturing flaws. Other defects commonly identified by the AOI systems include missing components and wrongful or skewed alignments. In every case, the performance of AOIs surpasses the ability of human inspectors through greater accuracy in a shorter time period [27].

The current practice in the industry involves the application of 3-D AOI equipment that can determine the component height as well. Previously, it was not feasible to measure the height of several components through the conventional 2-D AOI machines. Further, the 3-D AOI machines deliver superior performance in terms of capturing height-sensitive devices, for example, leaded components. The 2-D AOI machines use colored lighting from multiple angles and also incorporates side-angle cameras for inspecting these devices. However, the process does not produce accurate results. 3-D AOI machines, on the other hand, facilitate the detection of coplanarity for height-sensitive devices. The contemporary AOI technologies are well-reputed in the industry and compatible with various commercial standards. However, these techniques are capable of detecting many common errors in the PCBs;

can be well-integrated with the existing steps of PCB manufacturing; and deployed in-line. The major disadvantage of the AOI techniques includes failure to inspect hidden interconnects inside the BGAs and component packages. AOI machines inherently suffer from a limited visibility when it comes to complex PCBs. Furthermore, AOI machines might produce erroneous results when inspecting densely loaded PCBs due to hidden or shadowed components.

4.5.3.2 X-Ray Inspection

With the advent of SMT, the density of the components on a PCB is steadily increasing. Modern PCBs typically contain 20,000 or more solder joints. Moreover, SMT has helped the development of novel chip packages, such as, BGAs and CSPs, where the solder connections are not visible and, hence, cannot be inspected by conventional AOI equipment. To address the issues of limited visibility in modern PCBs, the X-ray inspection equipment is developed to scan solder joints inside the components and check for potential defects in a design. The X-ray inspection can be either manual or automatic. The rate of X-ray absorption by different materials vary according to their atomic weight. Any material with heavier weight is inclined to a higher amount of X-ray absorption. Consequently, materials with lighter elements provide comparatively better transparency in X-ray inspection. The solder joints of the PCBs are usually made of heavier elements, such as bismuth, tin, indium, silver, and lead. The PCBs, on the other hand, are made of lighter elements, for example, carbon, copper, aluminum, and silicon. X-ray inspections are efficient for analyzing solders as these joints show up very well when exposed to X-ray. However, most packages, including board substrate, component leads, and ICs are not properly visible through X-ray inspection.

The working principle of AOI and X-ray inspection is different as the X-rays are not reflected from the design under test (DUT). The X-rays pass through the design and help the extraction of an image on the other side of the board. Thus, X-ray inspection can aid the process of inspecting BGAs, where the interconnects are inside the components. As the shadow of components is not an issue in X-ray inspection, it is possible to inspect complex and highly dense boards with this mechanism. For instance, X-ray based inspection methods provide an internal view of solder joints of PCBs and help to determine any existing bubbles in the joints. These methods also help with the visibility of solder joint heels. The major advantage of X-ray-based inspection is the transparency in case of chip packages with connections underneath the components. These techniques also help with thorough inspection of a densely loaded board including the solder joints. The disadvantages of the X-ray-based inspection method are as follows. It is not well understood since the technology is newer and the investment in the technology is only effective for specific packages, e.g. BGAs and CSPs.

4.5.4 PCB TESTING

Once the inspection is completed, the manufactured boards become ready for testing. In this section, the in-circuit testing, functional testing, and JTAG-based boundary scan testing methods, which are most commonly employed in the industry (Fig. 4.10), are explained in detail [22,24].

4.5.4.1 In-Circuit Test (ICT)

The purpose of the ICT is to verify the design specifications through the exact list and placement of components. The common approach of performing ICTs include testing a loaded PCB with electrical probes and determining any discrepancy in the functionality caused by open or short circuit scenario.

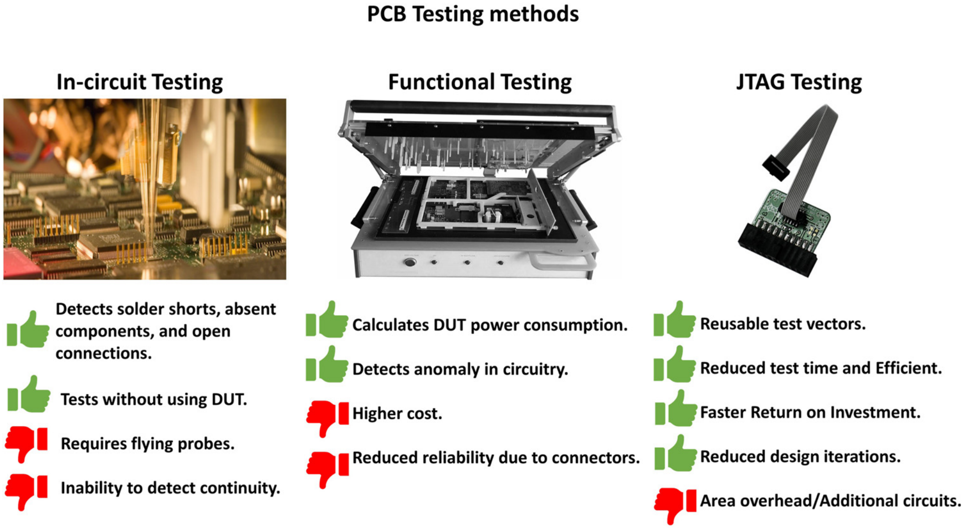


FIGURE 4.10

Comparison of PCB testing methods.

These probes also help to find out if the resistance, capacitance, or inductance of a design is correct according to the design specs. To perform the ICT, a set of flying probes or a fixture of bed-of-nails is required. A bed-of-nails tester is basically a test fixture made of arrays of small, spring-loaded pogo pins. The pins are connected to every node of the circuitry of the DUT. A successful connection between the nodes and pins establishes contact between the tester and the numerous testing points of the circuitry. However, the bed-of-nails fixtures are usually expensive to set up and not very compliant with changes based on design types. Further, these testers do not work very well with densely loaded PCBs.

A solution to the obstacles faced by the bed-of-nails tester is the usage of roving or flying probes. The deployment of flying probe involves using a fixture to hold the board steadily, and roving the tester probes constant over the required points of contact. As the movement of the probes can be programmed as per requirement, the flying probe facilitates testing of large number of boards with various design specifications. ICT tests, however, do not verify the operational validity of the boards as it assumes that the circuit is fully operational and delivers error-free performance.

The ICTs are efficient at finding physical defects, such as short or open circuit situation due to soldering, detecting missing or wrong components, and open connection between components. Testing the circuit without the power supply is another key aspect of ICT as the process of power input is associated with the risk of potential circuit damage. The ICT requires expensive test fixtures for the bed-of-nails or elaborate programming for roving probe set up. Moreover, these tests do not check the continuity of the design through the connectors of the circuit. Thus, there is a possibility that ICT might overlook existing connector faults in the circuit.

4.5.4.2 Functional Test (FCT)

FCTs are performed on PCBs to determine the functionality of a design. FCT is usually the final step of the manufacturing process. A manufacturer seeks to detect faults in the final hardware through functional testing. Any unmitigated fault in the final product might adversely affect the operation of the target system. Hence, a suite of functional tests is used to determine the quality of a board. The requirements of typical functional tests vary, based on the systems and designs under test. The development and testing procedures for functional testing also vary accordingly. The edge connectors and test probe pins of a PCB are utilized to interface the design with the functional testing tools. The testing also includes generating the proper electrical environment to simulate the operating conditions for the design. Hot Mock-up is a form of functional test prevalent in the industry. Other forms of testing include cycling the design with a comprehensive range of operational tests.

Primarily, functional tests help to identify defects in the PCB. These tests also help in determining the power consumption of the design under test. Functional testing can be applied to both analog and digital circuitry. However, the programming associated with functional testing is expensive as it requires a comprehensive understanding of the DUT and the working environment. The process typically requires costly high-speed instrumentation for the characterization of signals under consideration. Another drawback is that functional testing is performed through the connectors, which might result in further reliability issues due to regular wear and tear.

4.5.4.3 JTAG Boundary Scan Tests

In today's PCB industry, Joint Test Action Group (JTAG) boundary scan test is considered as an industry standard post-assembly verification and testing. The boundary scan technology of JTAG facilitates accessibility to numerous signals and device pins of complex modern ICs. Boundary scan cells are exploited to access the required signals via Test Access Port (TAP). The TAP can consist of two, four, or five signals based on the version of the JTAG being used. The four or five pin interface helps to establish a daisy-chained connection to test multiple chips located on the board. It is possible to test and control the states of the signals through the TAP interface. Hence, TAP helps with monitoring and detection of PCB faults via run-time operations. The operational modes of boundary scan cells can be classified into two categories, that is, functional and test mode. In functional mode, the scan cells do not have any effect on device operation. In test mode, the functional cores of the devices are isolated from the pins and boundary scan cells are employed to control and monitor values from devices under test. Figure 4.11A shows the system-level block diagram of JTAG-based boundary scan test signals and connection. Figure 4.11B shows the JTAG interface on real-life PCB.

JTAG-based boundary scan testing is easier to perform in comparison to traditional functional testing as the control pins are disconnected from enabled devices, and there is no requirement of additional pin configuration or booting to use the pins for testing purposes. The TAP interface significantly minimizes the physical access required for testing the PCB board by providing the opportunity to control and monitor the enabled signals of the device. The boundary scan capability of JTAG can be utilized mainly in two ways: a) To obtain good coverage in testing, connection testing is applied on the PCB boards that relies on the device capabilities of JTAG and the connections and nets on the board; b) One can use the JTAG enabled devices on a PCB to enhance test coverage and to communicate with peripheral devices that lack JTAG support. Figure 4.12 shows the cross-sectional view of an IC with JTAG boundary scan integrated into its periphery [23].

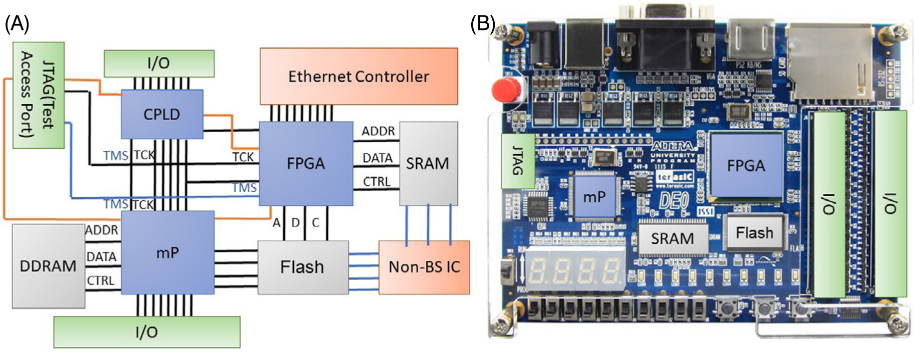


FIGURE 4.11

(A) System level block diagram of JTAG-based boundary scan test access. (B) Illustrative depiction of JTAG interface on PCB.

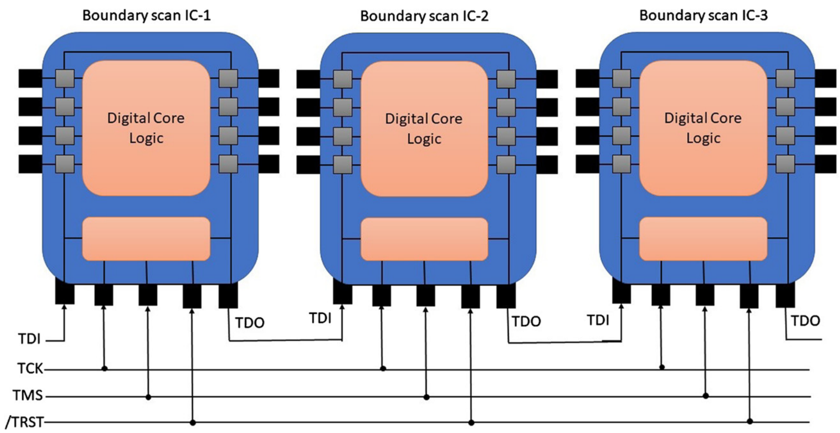


FIGURE 4.12

Integrated circuits with JTAG boundary scan.

JTAG reduces the cost of test generation by reducing the expenses and complexities associated with bed-of-nails fixture of ICT. Moreover, JTAG addresses the issues raised by the limitations of physical access for testing the interconnects placed between devices on a multi-layer PCB. The reusable test vectors of JTAG minimizes the expenses significantly. Conventional ICT techniques poses serious challenges in the diagnostic process of structural failures. On the contrary, JTAG provides an efficient way to testing these failures with reduced test pins and lesser testing time. Consequently, JTAG technique is cost-effective compared to ICT approaches. However, JTAG suffers from the drawback of area overhead for additional circuitry. Further, in many cases, it might be a challenging task to evaluate the effect of JTAG boundary cell capabilities on the circuit size as it depends on implementation details. In terms of design constraints, JTAG technique requires higher design effort to integrate the boundary scan into the periphery of the functional core.

4.6 HANDS-ON EXPERIMENT: REVERSE ENGINEERING ATTACKS

4.6.1 OBJECTIVE

This experiment is designed to give students practical experience on reverse engineering attacks on electronic hardware. In particular, it allows students to reverse-engineer a simple two-layer PCB.

4.6.2 METHOD

By setting the HaHa platform as the target of the attack, the students will apply a design-capturing technique to retrieve all the necessary design details for a PCB needed to re-create the design from the actual hardware. The experiment focuses on design information retrieval using visual inspection for the two-layered example PCB. The information includes both the type of components and the routing structure of the system under attack. Next, the students need to compile the obtained information to recreate a detailed schematic design of the HaHa platform.

4.6.3 LEARNING OUTCOME

Through carrying out the steps of the experiment, the students will experience the ease and the challenges with respect to PCB reverse engineering and understand the vulnerability associated with PCB piracy. They will also learn how to track components, capture their connectivity, and identify their functionality as it relates analyzing and debugging a PCB. Finally, this experiment is expected to help students understand the countermeasures to PCB reverse engineering and motivate them in developing new solutions.

4.6.4 ADVANCED OPTIONS

Additional exploration on this topic can be done through the application of this attack on more complex PCBs having complex circuitry with basic obfuscation.

More details about the experiment are available in the supplementary document. Please visit: http://hwsecuritybook.org.

4.7 EXERCISES

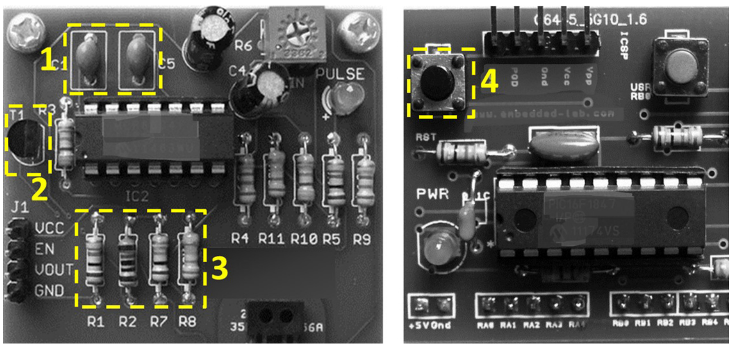
4.7.1 TRUE/FALSE QUESTIONS

1. The primary purpose of Printed Circuit Boards is to provide mechanical support and electrical connectivity.
2. Battery is a passive component found on PCBs.

3. Conductive substrate material is the base of any PCB board.
4. Part research and selection is a vital step of PCB design.
5. The manufacturers in the PCB industry have their own fabrication labs.
6. Automation in PCB inspection can produce worse results compared to manual inspection.
7. Functional testing verifies the circuit operation of the design.
8. Power-up testing can damage the PCB.
9. X-ray-based inspection is detrimental to PCB base material.
10. Surface Mount Technology facilitates the production of densely loaded complex PCBs.

4.7.2 SHORT-ANSWER TYPE QUESTIONS

1. What are some major events in the history and evolution of PCB design and development?
2. Briefly describe two automated PCB inspection techniques commonly used in the industry.
3. State the differences between in-circuit testing and functional testing.
4. Label the PCB components in the following figure.



5. What is JTAG-based boundary scan testing? How does it address the limitations of in-circuit testing?
6. What are the two common PCB assembly techniques? Briefly describe them.

4.7.3 LONG-ANSWER TYPE QUESTIONS

1. Briefly describe the common PCB components in terms of active and passive electrical elements.
2. Describe the life cycle of a modern PCB with a brief description about the steps and parties involved.
3. Describe the PCB design process with brief descriptions about the major steps.
4. Briefly discuss the contemporary PCB testing techniques with their advantages and disadvantages.
5. Briefly discuss the PCB assembly techniques currently practiced with their pros and cons.
6. Discuss the taxonomy of PCB design verification.
7. Briefly discuss the JTAG testing.

REFERENCES

[1] J. Li, P. Shrivastava, Z. Gao, H.-C. Zhang, Printed circuit board recycling: a state-of-the-art survey, IEEE Transactions on Electronics Packaging Manufacturing 27 (1) (2004) 33–42.
[2] J. Howard, Printed circuit board, Metal Finishing 11 (95) (1997) 117.
[3] Y. Crama, J. van de Klundert, F.C. Spieksma, Production planning problems in printed circuit board assembly, Discrete Applied Mathematics 123 (1–3) (2002) 339–361.
[4] J. LaDou, Printed circuit board industry, International Journal of Hygiene and Environmental Health 209 (3) (2006) 211–219.
[5] H.-H. Loh, M.-S. Lu, Printed circuit board inspection using image analysis, IEEE Transactions on Industry Applications 35 (2) (1999) 426–432.
[6] M.W. Jawitz, Printed Circuit Board Materials Handbook, McGraw Hill Professional, 1997.
[7] A. Kusiak, C. Kurasek, Data mining of printed-circuit board defects, IEEE Transactions on Robotics and Automation 17 (2) (2001) 191–196.
[8] I.E. Sutherland, D. Oestreicher, How big should a printed circuit board be? IEEE Transactions on Computers 100 (5) (1973) 537–542.
[9] T.F. Carmon, O.Z. Maimon, E.M. Dar-El, Group set-up for printed circuit board assembly, The International Journal of Production Research 27 (10) (1989) 1795–1810.
[10] J. Vanfleteren, M. Gonzalez, F. Bossuyt, Y.-Y. Hsu, T. Vervust, I. De Wolf, M. Jablonski, Printed circuit board technology inspired stretchable circuits, MRS Bulletin 37 (3) (2012) 254–260.
[11] P.-C. Chang, Y.-W. Wang, C.-Y. Tsai, Evolving neural network for printed circuit board sales forecasting, Expert Systems with Applications 29 (1) (2005) 83–92.
[12] O. Maimon, A. Shtub, Grouping methods for printed circuit board assembly, The International Journal of Production Research 29 (7) (1991) 1379–1390.
[13] M. Gong, C.-J. Kim, Two-dimensional digital microfluidic system by multilayer printed circuit board, in: Micro Electro Mechanical Systems, 2005. MEMS 2005. 18th IEEE International Conference on, IEEE, 2005, pp. 726–729.
[14] P. Hadi, M. Xu, C.S. Lin, C.-W. Hui, G. McKay, Waste printed circuit board recycling techniques and product utilization, Journal of Hazardous Materials 283 (2015) 234–243.
[15] P.T. Vianco, An overview of surface finishes and their role in printed circuit board solderability and solder joint performance, Circuit World 25 (1) (1999) 6–24.
[16] E. Duman, I. Or, The quadratic assignment problem in the context of the printed circuit board assembly process, Computers & Operations Research 34 (1) (2007) 163–179.
[17] P. Johnston, Printed circuit board design guidelines for ball grid array packages, Journal of Surface Mount Technology 9 (1996) 12–18.
[18] S. Ghosh, A. Basak, S. Bhunia, How secure are printed circuit boards against Trojan attacks? IEEE Design & Test 32 (2015) 7–16.
[19] W. Jillek, W. Yung, Embedded components in printed circuit boards: a processing technology review, The International Journal of Advanced Manufacturing Technology 25 (2005) 350–360.
[20] S. Paley, T. Hoque, S. Bhunia, Active protection against PCB physical tampering, in: Quality Electronic Design (ISQED), 2016 17th International Symposium on, IEEE, pp. 356–361.
[21] J. Carlsson, Crosstalk on printed circuit boards, SP Rapport, 1994, p. 14.
[22] B. Sood, M. Pecht, Controlling moisture in printed circuit boards, IPC Apex EXPO Proceedings (2010).
[23] O. Solsjö, Secure key management in a trusted domain on mobile devices, 2015.
[24] S.H. Hwang, M.H. Cho, S.-K. Kang, H.-H. Park, H.S. Cho, S.-H. Kim, K.-U. Shin, S.-W. Ha, Passively assembled optical interconnection system based on an optical printed-circuit board, IEEE Photonics Technology Letters 18 (5) (2006) 652–654.
[25] B. Archambeault, C. Brench, S. Connor, Review of printed-circuit-board level EMI/EMC issues and tools, IEEE Transactions on Electromagnetic Compatibility 52 (2) (2010) 455–461.
[26] T. Hubing, T. Van Doren, F. Sha, J. Drewniak, M. Wilhelm, An experimental investigation of 4-layer printed circuit board decoupling, in: Electromagnetic Compatibility, 1995. Symposium Record., 1995 IEEE International Symposium on, IEEE, 1995, pp. 308–312.
[27] H. Rau, C.-H. Wu, Automatic optical inspection for detecting defects on printed circuit board inner layers, The International Journal of Advanced Manufacturing Technology 25 (9–10) (2005) 940–946.

[28] R.G. Askin, Printed circuit board family grouping and component, Naval Research Logistics 41 (1994) 587–608.
[29] V.J. Leon, B.A. Peters, A comparison of setup strategies for printed circuit board assembly, Computers & Industrial Engineering 34 (1) (1998) 219–234.
[30] G. Reinelt, A case study: TSPs in printed circuit board production, in: The Traveling Salesman: Computational Solutions for TSP Applications, 1994, pp. 187–199.