Chapter-2
Wiring Techniques for Electronic Circuits

1. Introduction

An electronic system (card) is composed of a set of interconnected components that communicate with each other and with the outside world through sensors. The performance of an electronic system is related to the quality of these interconnections, which depend on several factors such as:

- Connection method (soldering, pressure contact, etc.)
- Sizing (impedance variation, heating, etc.)
- Specific protections (mechanical, electrical, electromagnetic, thermal, etc.)
- Connection methods (wire, connector, printed circuit board, etc.)

2. Printed Circuit Board (PCB)

2.1 Definition

- A Printed Circuit Board (PCB) is a (rigid/flexible) support used to sustain and interconnect electronic components, in order to create an electronic card.
- The PCB is an assemblage of one or multiple layers of copper, covered with a thin layer of colored resin that protects the tracks from oxidation and short circuits. These copper layers are separated by an insulating material (epoxy, bakelite, FR-4).
- The copper layer is chemically engraved using a specific process to create tracks and pads (Non-Perforated / Perforated). Non-perforated pads are used for soldering SMD (Surface Mounted Device). Perforated pads are used to establish connections between components (soldering) and between the layers (using Vertical Interconnect Access "VIAs").

2.2 Types of PCB

There are four types of PCBs: single-sided, double-sided, multilayer, and flexible.

2.2.1 Single-Sided PCB

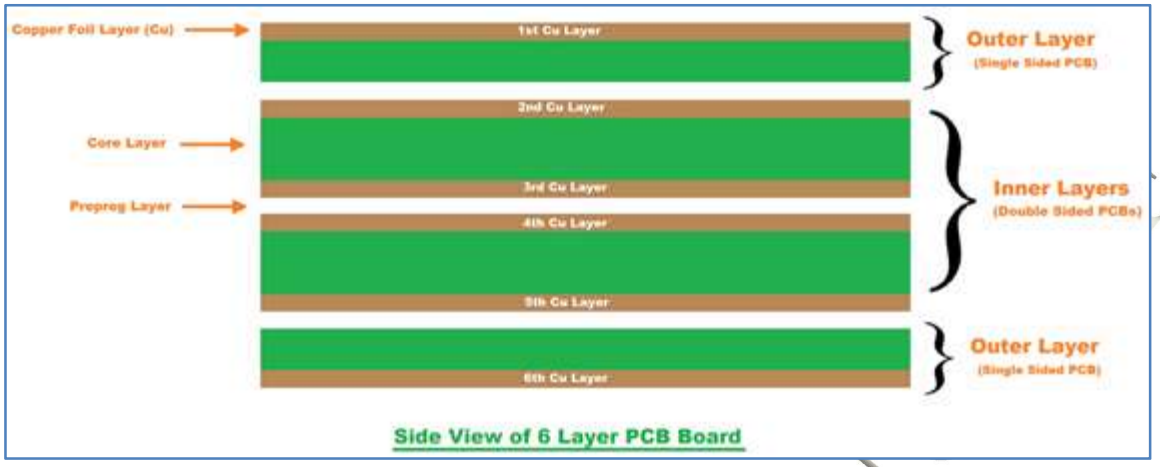
This is the simplest and most commonly used PCB, with a single conductive layer (copper) supported by an insulating layer (epoxy). The holes in this case allow for the insertion of electronic components.

2.2.2 Double-Sided PCB

This type of PCB has two conductive copper layers separated by an insulating layer. There are two types of holes: insertion holes, which hold the components in the circuit, and VIAs, which establish an electrical connection between the two conductive layers.

2.2.3 Multilayer PCB

The multilayer PCB is has at least three conductive layers separated by insulating layers. These PCBs are used to create complex electronic circuits with multiple components, and the multiple layers allow for complex routing of connections, reducing the need for extensive point-to-point wiring. The multilayer PCBs can have various types of holes, including through-hole VIAs, blind VIAs, and buried VIAs, which enable connections between the different conductive layers. They are commonly used in electronic devices and systems where space is limited, and a high level of integration is required.



2.2.4 Flexible PCB

This type of PCB is made from thin insulating material, often polyimide, and it serves an electrical connection function, much like an electrical cable connecting two connectors. Flexible circuits come in single-sided, double-sided, and multilayer configurations. The advantages of these PCBs include:

- Reduction in weight by approximately 30%.
- Space savings of about 60%.
- Reduction in traditional wiring errors (poor contact).

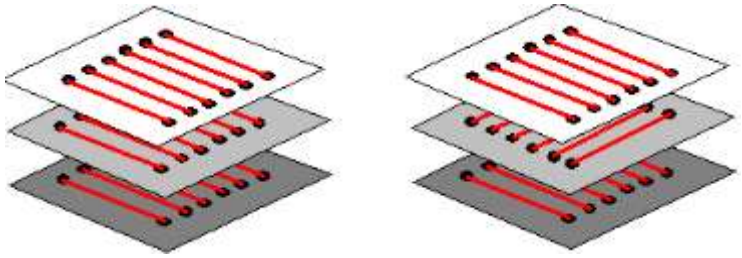
2.3 PCB Component Technology

2.3.1 Conductive Layers

- The standard copper thickness is 35µm per layer.
- For power electronics applications, thicker copper layers can be used (70µm/layer or 100µm/layer).

2.3.2 Insulating Layers

- These layers have a dual role: to harden the board and to insulate the electrical layers.
- The standard thickness of these layers is 16/10mm.
- For heavy components, it can go up to 20/10mm or 24/10mm.
- The permittivity of insulating materials is crucial in high-frequency applications (e.g., Internet routers). The arrangement of traces and disturbance calculations are performed by CAD (Computer-Aided Design) software.



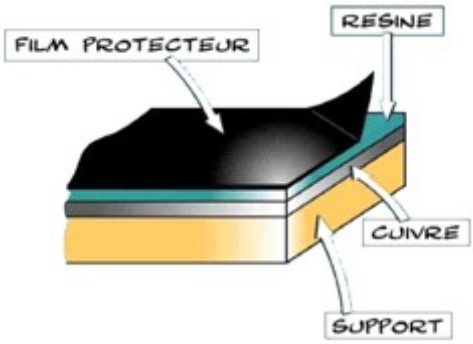
2.3.3 Les VIAs

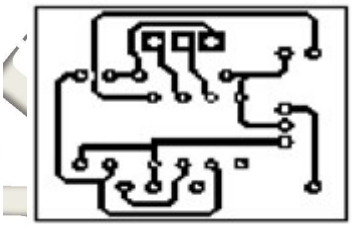
- VIAs (Vertical Interconnect Access) are metal rivets drilled through the layers of a PCB to allow the passage of electrical connections between different layers to facilitate the routing of signals and power between the various components on the layers.
- Theirs main role is to ensure communication between the conductive layers, as they can also receive a component lead and provide a mechanical function at the same time.

2.4 The different Steps in the Design of a PCB

2.4.1 The Preparation of the Board

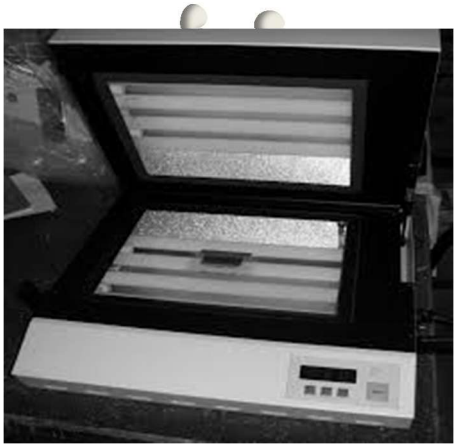
Cut the board to the dimensions of the layout using shears. Remove the black adhesive film (black plastic) that protects the photosensitive layer from UV rays. The resin on the surface has the property of changing when exposed to UV rays.

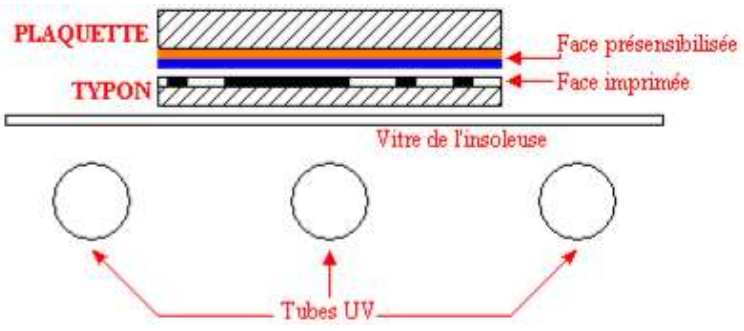




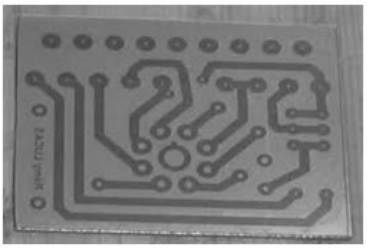
2.4.2 Insolation

Place the assembly (typon + copper side plate) in an insolator and expose it to UV rays for 120 sec, as illustrated in the figure below.





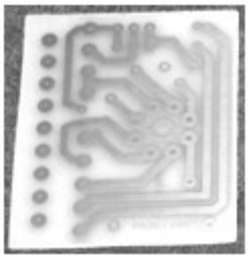
2.4.3 Development and rinsing



Dip the plate in a container containing a developer (NaOH) for a few seconds to dissolve the areas of the resin that were destroyed during exposure. Remove and thoroughly rinse the plate with water to stop the developer's activity.

Attention: Follow the safety directives carefully.

2.4.4 Etching and cleaning



Dip the plate in a container containing an acid (ferric chloride) to decompose the copper around the traces protected by the resin for 15 to 20 minutes. The chemical reaction can be accelerated by heating the container and adding an air diffuser.

Remove and thoroughly rinse the plate with hot water to stop the acid's activity.

Clean the plate using a cloth wet with alcohol to remove the remaining resin covering the traces.

Attention: Follow the safety directives carefully.

2.4.5 Drilling and implantation

Drill the plate in the middle of the pads with a drilling diameter between (0.8 and 1mm).

Test the continuity of the traces to check for any potential micro-cuts.

Once drilling is complete, components can be placed and soldered using a soldering iron and solder.