

## 3 Interconnection of Circuits

### 3.1 Introduction

All electronic circuits need some form of interconnection. When selecting a type of interconnection technology the following factors must be considered:

Resistivity of the connections	should be low
Connection density	should be high
Cost	should be low
Reliability	should be high
Stray impedances (high speed circuits)	should be small
Prototyping turn around	should be quick

Various competing technologies have differing advantages and these determine which will be used for a particular application. Some of the currently available interconnection technologies will now be described.

### 3.2 Printed Circuits

The most common form of interconnection is provided by the printed circuit board or PCB. The PCB is basically a set of interconnections or tracks placed onto insulating material or substrate. For high conductivity, tracks are made of copper. In this way connections can be made between the legs of components which have been placed on the PCB as shown in figure 1.

To make connection between the legs and the tracks, solder is used. Holes can be drilled through the PCB for the legs of the component, to physically hold the components in place before soldering and for support. This is called a '**Thru-hole**' PCB and is shown in figure 1(a). To allow for good connection by the solder a copper annulus or 'pad' is placed around each component leg.

Instead of drilling holes for the legs, the component can be placed on the surface of the PCB and solder is used to connect and fasten the component. This technique is **called 'Surface Mount Technology'** and is shown in figure 1(b). Pads are placed under each component pin to allow good mechanical and electrical connection. Surface mount is particularly suited to small and light components. Where heavy components must be used 'Thru-holes' can also be used in conjunction with surface mounted components or additional support for the component can be organised.

Usually the insulating material consists of fibre-glass which is 1/16th of an inch thick. It is called G10 fibre-glass. Other insulators can be used, for special applications and these are described in section 3.2.3.

#### 3.2.1 Plated Thru-Holes

Simple circuits can often be entirely connected using one layer of interconnects (tracks). This is called a '**Single-Sided PCB**'. Often all points on the PCB cannot be interconnected using a single layer of tracks and wire links need to be inserted to jump over other tracks. By using two layers or a '**Double-Sided PCB**' the insertion of links can be avoided. However connections must be made between the two layers. These connections are called a **vias**.

Vias are produced by drilling holes through the fibre-glass and then plating metal through the hole. This interconnects a track on one side with a track on the other side. These vias are called **Plated Thru-Holes**.

By using a matrix arrangement any two points on a double sided PCB can be interconnected. This is achieved if the tracks on one side always travel at right angles to the tracks on the other side. This is shown in figure 2.

### 3.2.2 *Multi-Layer PCBs*

Sometimes the circuit is so complex that interconnection cannot be made on two layers. Interconnecting tracks must avoid connecting to the pads of components not connected to that track. If the pad/pin density is high there will be insufficient room to run all the tracks across the PCB. PCBs with more than two layers are commonly called **Multi-Layer PCBs**.

Multi-layer boards are sandwiches of conductors and insulators as shown in figure 3. Plated thru-holes connect up all layers in the sandwich. It is desirable to use wide tracks for tracks carrying high currents, such as power supplies. The wider the track the better the conductivity and the lower the voltage drop along the track. Where multi-layer PCBs are used it is common to allocate an entire layer to a particular supply voltage. For the supplies VCC and GND, two layers would be used providing excellent connection to all components on the PCB.

### 3.2.3 *Printed circuits for High Speed Circuits*

At frequencies above 1 GHz ( $10^9$  Hz) fibre-glass becomes lossy and signals are absorbed by the insulator. Other materials such as Teflon and Alumina (Aluminium Oxide) are usually used in these applications.

**Thin-Film** circuits are circuits made from metal which has been evaporated onto the substrate (insulator). Alumina is commonly used as a substrate. Due to the stability and rigidity of alumina high accuracy circuits can be fabricated. This technology is used to manufacture microwave circuits. For this application the geometry of the interconnect produces stray impedances which alter the electrical properties of the circuit. Tight control of track and pad dimensions is essential.

**Thick-Film** circuits are lower accuracy circuits where a conductive metal paste is screen printed onto the substrate. The tracks are less conductive than thin-film circuits and therefore tend to be lossy.

## 3.3 **Manufacturing PCBs**

Except for thick-film circuits, tracks are always made by etching away metal surrounding the track. The starting point for the manufacturing process is an insulator coated or clad with solid copper. The following description gives the typical steps taken to produce a double-sided PCB.

Where plated thru-holes are to be produced, these are drilled first. Copper is deposited in the holes from solution. This temporarily joins the two layers.

A photo-sensitive layer or **resist** is applied to both sides of the PCB. A film which has an image of the tracks is placed over each layer in turn. Where the film is transparent light exposes (hardens) the resist in between the tracks as shown in figure 4. After developing, the tracks are uncovered while the areas between the tracks are covered with a hardened resist film.

The entire board is now plated with tin. This strengthens the plated thru-holes, protects the tracks and improves solderability.

The resist film is now dissolved to expose the unplated copper areas between the tracks. The copper is etched between the tracks and not under the tin coating.

Finally the tin layer is melted or **reflowed** to produce an easily solderable finish.

The board can now be screen printed with a solder mask. This leaves the pads exposed while covering the tracks. This step is essential for automated assembly where the board is soldered by dipping into a reservoir of molten solder. This step prevents solder shorts between closely spaced tracks.

To assist in assembly and repair, a component overlay can be screen printed onto the PCB. This allows each component to be easily identified and shows the component orientation. Testing or connection points can also be indicated.

Multi-layer PCBs are manufactured by gluing together pre-etched laminates. Holes are then plated through the sandwich. Single-sided and non-plated thru-hole PCBs can be drilled before or after etching the track outline.

### **3.3.1 Wire Wrapping**

This is a connection system suited to building prototype circuits. The components are mounted in **wire-wrap sockets**. These sockets have pins with a square profile. Fine wire is wrapped around these pins to make the connection and no solder is required. The edges of the pins cut into the connecting wire. The result is a high reliability, removable connection as shown in figure 4.

Although primarily a prototyping aid, wire-wrap can also be used for production PCBs. Computer controlled machines can automatically wrap entire boards. The interconnection information is directly available from the circuit diagram and no 'layout' of tracks needs to be done. Circuits for military applications are often made in this way to achieve high reliability.

Usually wire-wrap PCBs are quite bulky due to the long socket pins and the mass of wire under the PCB. However a manufacturing technique known as **Multiwire** uses hollow pins to wrap to. The mass of interconnecting wires is encased in resin and the pins form conventional component pads. The result is a conventional looking thru-hole PCB.

### **3.3.2 Speed Wires**

This is a very fast prototype connection system. It is much faster than wire wrap but does not have the reliability of wire wrapping. Speed wire is normally limited to small projects to prove a method not an entire circuit. It has not gained popularity due to its very short mechanical life, tending to 'fall apart' when moved.

### **3.3.3 Prototyping Board (also known as Breadboard)**

The protoboard is similar in concept to speed wire but more reliable. It also tends to fall apart when moved. However, with care and not allowing anything to touch the link wires, a design on a prototype board can be moved around. As with speed wire, this connection system is used mainly for small circuits when proving a method, *not* for part of a production-quality system as wire wrapping may be.

Protoboards can use pre-cut and stripped lengths of wire, which makes assembly of circuits very quick, rivalling the speed of speed wire assembly.

## **3.4 Cost**

The cost of a PCB increases with the number of layers. A considerable increase in cost and manufacturing turn-around is experienced if multi-layer PCBs are required. Wire-wrap tends to be more expensive due to the time taken to produce the connections. Multiwire PCBs are priced to be competitive with multi-layer PCBs.

*Original author: Erich Stumpf 1991*

*Revised: C R Stephens 1996, G. Tobin 2007 July 20*

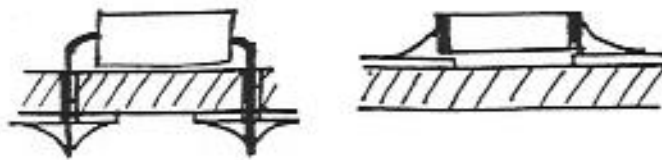


Figure 1 (a) Thru-hole

(b) Surface Mount

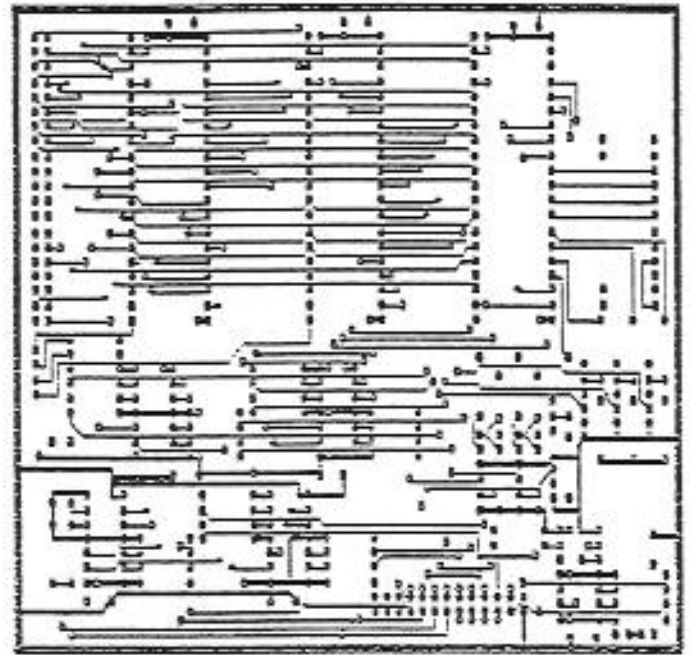
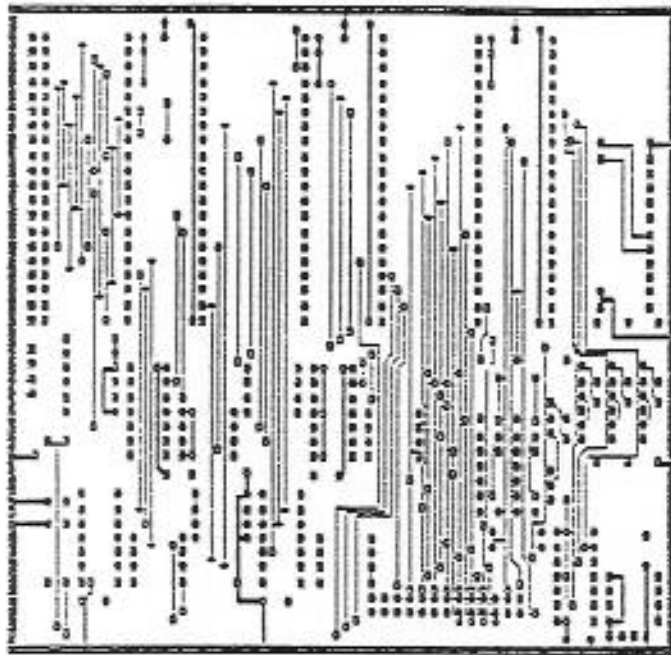


Figure 2 Component side

Solder side

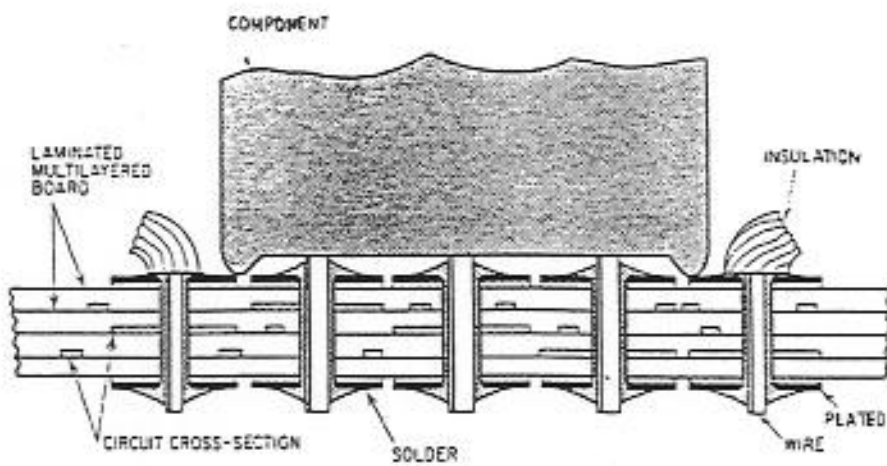


Figure 3 Multi-layer laminate

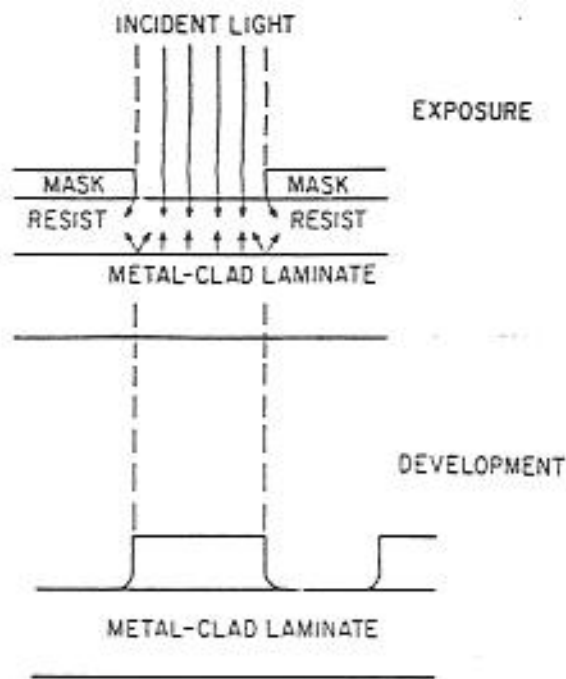


Figure 4 Image transfer

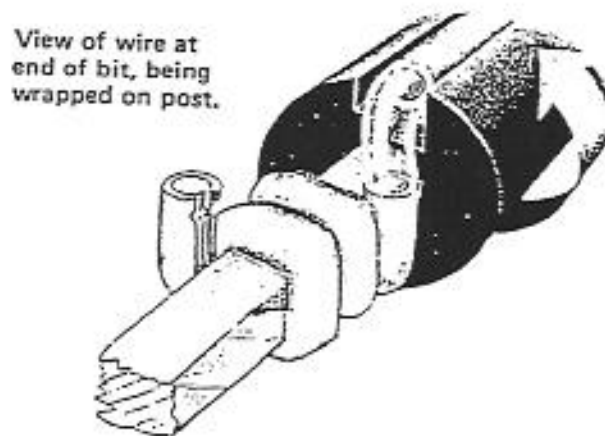


Figure 5 Wire-wrap connection