

PRINTED CIRCUIT BOARDS

*Design, Fabrication,
Assembly and Testing*

Dr R S Khandpur

Director General,

Pushpa Gujral Science City, Kapurthala, Punjab

Formerly

Director General,

Centre for Electronics Design and Technology of India (CEDTI)

Dept. of Information Technology, New Delhi

and

Director

CEDTI, Mohali (Chandigarh) Punjab



McGraw-Hill

New York Chicago San Francisco Lisbon London

Madrid Mexico City Milan New Delhi San Juan

Seoul Singapore Sydney Toronto

Cataloging-in-Publication Data is on file with the Library of Congress

Copyright © 2006 by The McGraw-Hill Companies, Inc. All rights reserved. Printed in the United States of America. Except as permitted under the United States Copyright Act of 1976, no part of this publication may be reproduced or distributed in any form or by any means, or stored in a database or retrieval system, without the prior written permission of the publisher.

1 2 3 4 5 6 7 8 9 0 DOC/DOC 0 1 0 9 8 7 6 5

ISBN 0-07-146420-4

Printed and bound by RR Donnelley.

This book was previously published by Tata McGraw-Hill Publishing Company Limited.

McGraw-Hill books are available at special quantity discounts to use as premiums and sales promotions, or for use in corporate training programs. For more information, please write to the Director of Special Sales, McGraw-Hill Professional, Two Penn Plaza, New York, NY 10121-2298. Or contact your local bookstore.



This book is printed on recycled, acid-free paper containing a minimum of 50% recycled, de-inked fiber.

Information contained in this work has been obtained by The McGraw-Hill Companies, Inc. ("McGraw-Hill") from sources believed to be reliable. However, neither McGraw-Hill nor its authors guarantee the accuracy or completeness of any information published herein, and neither McGraw-Hill nor its authors shall be responsible for any errors, omissions, or damages arising out of use of this information. This work is published with the understanding that McGraw-Hill and its authors are supplying information but are not attempting to render engineering or other professional services. If such services are required, the assistance of an appropriate professional should be sought.



Contents

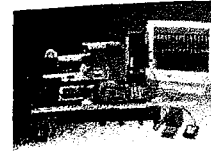
Preface

xxi

1. Basics of Printed Circuit Boards

1

- 1.1 Connectivity in Electronic Equipment 1
 - 1.1.1 Advantages of Printed Circuit Boards 1
- 1.2 Evolution of Printed Circuit Boards 2
- 1.3 Components of a Printed Circuit Board 4
- 1.4 Classification of Printed Circuit Boards 5
 - 1.4.1 Single-sided Printed Circuit Boards 5
 - 1.4.2 Double-sided Printed Circuit Boards 6
 - 1.4.3 Multi-layer Boards 7
 - 1.4.4 Rigid and Flexible Printed Circuit Boards 9
- 1.5 Manufacturing of Basic Printed Circuit Boards 10
 - 1.5.1 Single-sided Boards 10
 - 1.5.2 Double-sided Plated Through-holes 14
 - 1.5.3 Multi-layer Boards 16
 - 1.5.4 Flexible Boards 16
- 1.6 Challenges in Modern PCB Design and Manufacture 17
- 1.7 Major Market Drivers for the PCB Industry 19
- 1.8 PCBs with Embedded Components 21
- 1.9 Standards on Printed Circuit Boards 23
- 1.10 Useful Standards 24



Basics of Printed Circuit Boards



1.1 Connectivity in Electronic Equipment

Electronic equipment is a combination of electrical and electronic components connected to produce a certain designed function. In the era of vacuum tubes and even later, electronic equipment was constructed by hand wiring and by point-to-point soldering. The wires were stripped of their insulation, tinned and soldered. Each discrete component was installed by hand, electrically and mechanically. The equipment was obviously large, awkward and bulky. It was difficult to meet the demanding requirements for the use of this equipment in aircrafts, the health sector and home emergency uses, thereby necessitating the development of smaller and more compact electronic equipment.

A natural evolution took place in several areas. Smaller components were developed and modular design became popular, basically intended to decrease the time between unit failure and repair due to easy replaceability. The use of miniaturization and sub-miniaturization in electronic equipment design gave birth to a new technique in inter-component wiring and assembly that is popularly known as the *printed circuit board*. The printed circuit board provides both the physical structure for mounting and holding electronic components as well as the electrical interconnection between components.

Printed circuit board is usually abbreviated as PCB and quite often referred to as *board*. However, in the USA, the term PWB (*Printed Wiring Board*) is more often used instead of PCB.

1.1.1 Advantages of Printed Circuit Boards

There are many good reasons for using printed circuit boards instead of other interconnection wiring methods and component mounting techniques, some of which are as follows:

- i. The size of component assembly is reduced with a corresponding decrease in weight.
- ii. Quantity production can be achieved at lower unit cost.
- iii. Component wiring and assembly can be mechanized.
- iv. Circuit characteristics can be maintained without introducing variation in inter-circuit capacitance.
- v. They ensure a high level of repeatability and offer uniformity of electrical characteristics from assembly to assembly.
- vi. The location of parts is fixed, which simplifies identification and maintenance of electronic equipment and systems.
- vii. Inspection time is reduced because printed circuitry eliminates the probability of error.
- viii. Printed wiring personnel require minimal technical skills and training. Chances of mis-wiring or short-circuited wiring are minimized.

1.2 Evolution of Printed Circuit Boards

The history of development of printed circuit boards is not very old. They have been in commercial use only since the early 1950s, even though their concept originated nearly 50 years prior to their commercial use.

Frank Sprague, the founder of Sprague Electric, while still an apprentice, had the idea, in 1904, of eliminating point-to-point wiring. When he conferred with his mentor, Thomas Edison, for implementing his concept, it was suggested that silver reduction, as used in mirror manufacture or the printing of graphite pastes on linen paper, may prove to be suitable to achieve the objective. Subsequently, a number of pioneering efforts were made for the development of printed circuit boards, out of which the following events occupy place of eminence.

The first significant contribution came from Mr. Charles Ducas, who filed a patent application at the US Patent Office on March 2, 1925 for his proposal to mount electrical metal deposits in the shape of conductors directly onto the insulation material to simplify the construction of electrical appliances (Fjelstad, 2001). He used a stencil to form the conductors on the surface of insulation material and applied a conductive paste in the lines desired. After removal of the stencil, the lines were reinforced to the desired thickness by electrolytic metal deposition. This development considerably simplified the manufacture of electrical appliances because the electrolytic metal deposition, being a simple process, could be carried out by unskilled operators.

Just 17 days later, Mr. Francis T. Harmann filed a patent for the so-called subtractive method of making PCBs. This development could be considered as the forerunner of etching technology. In April 1926, a patent was granted to Mr. Cesar Pasolini in France for his invention of the additive way of making electrical connections. Mr. Samuel Charles Ryder filed an Australian patent in September 1928, related to the manufacture of inductance coils for use in radio tuning devices or other such applications. He proposed to print or spray the substrate directly with conductive paint

during manufacture. Similarly, the patent application filed by Mr. Herbert C. Arlt in July 1935 in America again emphasized the avoidance of wires as the basic purpose of the development.

The major contribution towards the development of modern printed circuit technology was made by Dr. Paul Eisler, who proposed copper clad insulation material in sheet form for use as the base material in circuit board manufacture. According to him, "a resist in the shape of the circuit pattern is printed onto the surface of the copper cladding with the uncovered metal being removed by etching". He also proposed the generation of conductors on both sides of the copper clad base material, with connection between conductors on both sides being made through eyelets. Eisler's work not only gave birth to a method of mass production and an assembly scheme but also offered economy in weight and space, which is especially important in military equipment. Little wonder that Dr. Eisler is often called the Father of printed circuit board technology. However, he gave preference to the use of eyelets rather than the plated-through hole technology, which, with time, became an essential process for the manufacture of double-sided and multi-layer printed circuit boards.

Towards the end of World War II, a technology developed by the US National Bureau of Standards was used in the volume production of US army VT proximity fuses for rockets. Unlike the Print and Etch technique of Eisler, this technology used printed silver paste conductors and graphite resistors that were screen printed onto ceramic substrate. This technique is more commonly associated with today's hybrid circuit technology. It was this technique that ushered in the commercial use of printed circuits.

After World War II, fascinating developments took place in the field of electronics, resulting in a high demand for consumer products like radio and television and simultaneously, the use of electronics for military applications. These developments resulted in the need for reliable circuit boards with increasing complexity. After attaining the level of maximum density, based on contemporary fabrication limitations, single side boards were replaced by double side boards, which allowed wires to cross over each other without the need for additional special jumpers. This was accomplished finally by plated-through holes.

During 1953–55, Motorola introduced the copper metal plating process to provide interconnection between two sides of a board, which was found to be more suitable for the mass manufacturing process. In the 1960s, the electroless method was introduced using catalyst activators, while Photo Circuits, USA, developed the fully additive process in 1964. In this method, the base material does not have copper on it and the copper is plated selectively on the required places for interconnections.

The late 1960s witnessed phenomenal growth in the field of consumer electronics, which necessitated the introduction of automation in fabrication and in testing of bare board/populated circuit boards. Soon thereafter in the 1970s, printed circuit boards were firmly entrenched in the consumer electronic, scientific equipment, medical equipment, air and space, defence, and in nearly all branches of electronics, which later culminated in the personal computer industry. Several new processes were subsequently developed for applications such as photo film lamination, dry film and wet film resist, solder masking, legend printing and CNC drilling, etc. The size of the printed board got considerably reduced with the manufacture of multi-layer and rigid-flexible circuit boards which made use of buried and blind vias plated through hole connection and wet process chemistry.

New developments in component technology, especially in the area of surface mount technology, resulted in several innovations in PCB materials and processes, and today there are constant pressures for improvements in PCB technology in all its aspects. The continuing trend towards high functionality integrated circuit (IC) components with higher input-output (I/O) pin counts of the IC packages has resulted in increased demand for fine-featured PCBs, giving rise to "high density interconnect structures (HDIS)" which are now manufactured by a large number of companies. High frequency electronic systems, with their high speed operations, create a demand for PCBs with lower electrical losses. In addition, higher operating voltages require PCBs with greater resistance to voltage breakdown, high voltage tracking and corona.

PCBs constitute a very important strategic component for electronic products. It is therefore no surprise that the PCB industry worldwide is a booming market, and an annual growth rate of 7 per cent is expected in this industry up to the year 2010. Therefore, printed circuits are likely to continue to be the icon of the electronic industry well through the next decade, which will of course, bring higher functionality/density, improved cost and reliability through more tightly controlled cost-effective processes. The subtractive process-based PCB world is likely to have a transition to an additive process. We can also expect an increase in the use of flexible circuits and a move towards more environment-friendly PCBs without the use of lead. Predictably, the various kinds of printed circuit boards, both those that are currently existing and those to be developed, will retain their function as an essential part of electronics and, in some cases, may even achieve a more significant place in the electronics industry.

1.3 Components of a Printed Circuit Board

The essential components of a printed circuit board are:

- The *base*, which is a thin board of insulating material, rigid or flexible, which supports all conductors and components; and
- The *conductors*, normally of high purity copper in the form of thin strips of appropriate shapes firmly attached to the base material.

The *base* provides mechanical support to all copper areas and all components attached to the copper. The electrical properties of the completed circuit depend upon the dielectric properties of the base material and must therefore, be known and appropriately controlled.

The *conductors* provide not only the electrical connections between components but also solderable attachment points for the same.

When the completed board provides mechanical support and all necessary electrical connections to the components, it is essentially a Printed Wiring Board or Printed Circuit Board. The term *printed* became popular because the conductive areas are usually generated by means of a printing process like screen printing or photo-engraving, which are commonly used to print drawings or inscriptions.

1.4 Classification of Printed Circuit Boards

Printed Circuit Boards may be classified according to their various attributes, often with ambiguous results. They were traditionally divided into three classes according to their use and applications, and were commonly referred to as consumer, professional and high reliability boards.

Consumer PCBs were generally used in consumer products such as radio, television, and cheap test and measuring equipment. They used less expensive base material and allowed greater tolerances for manufacture to keep the cost low. Much importance was not given to good and consistent electrical properties.

Professional boards were made of better quality material to achieve tighter electrical and environmental specifications using controlled fabrication techniques. Higher reliability boards, normally used in strategic applications, were meant to provide the best of electrical properties through the use of high quality base material and tightly controlled manufacturing processes.

The above classification might have been applicable two or three decades ago, but presently, the distinction between consumer and professional markets has disappeared. Many consumer products like compact discs, camcorders or cameras have become more complex, reliable and demanding than what was hitherto considered as professional equipment like personal computers. The advent of surface mount technology and developments in automatic assembly techniques requires that the boards even for the cheapest product must be manufactured to strict mechanical tolerances.

A more simple and understandable classification is now used, which is based on the number of planes or layers of wiring, which constitute the total wiring assembly or structures, and to the presence or absence of plated-through holes. This method of classifying boards has the advantage of being related directly to the board specifications. The important distinguishing constructions of PCBs are detailed below.

1.4.1 Single-sided Printed Circuit Boards

'Single-sided' means that wiring is available only on one side of the insulating substrate. The side which contains the circuit pattern is called the 'solder side' whereas the other side is called the 'component side'. These types of boards are mostly used in case of simple circuitry and where the manufacturing costs are to be kept at a minimum. Nevertheless, they represent a large volume of printed boards currently produced for professional and non-professional grades. Figure 1.1 shows the arrangement of a single-sided board.

The single-sided boards are manufactured mostly by the 'print and etch' method or by the 'die-cut' technique by using a die that carries an image of the wiring pattern; and the die is either photo-engraved or machine-engraved.

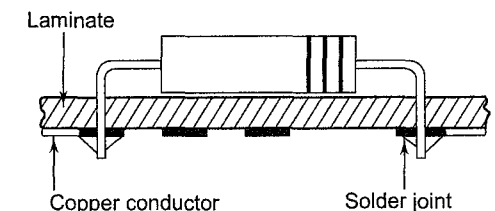


Fig. 1.1 Single-sided PCB

A variable resistor can be used either as a rheostat or potentiometer. Figure 2.14 shows the difference in the two applications. When used as a voltage divider, the resistor element is connected to a voltage reference source and the slide arm, which is used as the pick-off point, and can be moved to obtain the desired voltage.

For a variable resistor configuration, the resistor element is connected to the circuit at each end and the slide arm is connected to one of the ends. Alternatively, the entire resistance is in series and the slider is connected to an external circuit. Even in this configuration, it may be used like a potentiometer.

Variable resistors can be constructed to follow one of the following laws:

- **Linear:** The resistance of the pot is distributed evenly over its entire length.
- **Log:** The resistance of the pot varies so as to follow the logarithmic law. In these pots, when the wiper is turned, the resistance increases (from zero) very slowly and gradually until about the half way mark. From then onwards, as the wiper shaft is turned further, the resistance will increase much more rapidly in comparison with the first half of the pot-rotor rotation.

Figure 2.15 shows the change of resistance value with the angle of rotation for linear and logarithmic type of variable resistors.

- **Sine-Cosine Potentiometers:** As the name implies, the variation of resistance over the track, when the wiper moves, follows the sine-cosine law. The total operative track length over 360 degrees of rotation is divided into four quadrants of 90 degrees each.

Figure 2.16 shows the package shapes of commonly used variable resistors.

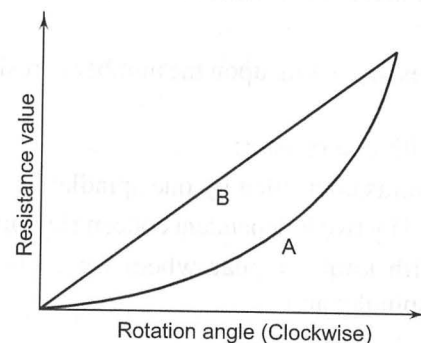


Fig. 2.15 Change of resistance value with rotation angle

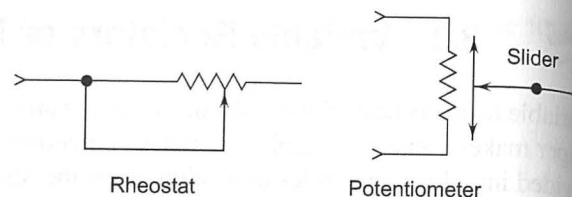


Fig. 2.14 Difference between a potentiometer and rheostat operation

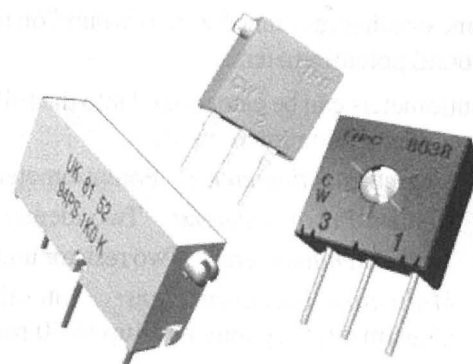


Fig. 2.16 Package shapes of variable resistors

2.4 Light-dependent Resistors (LDRs)

Light-dependent Resistors are made of cadmium sulphide. They contain very few free electrons when kept in complete darkness and therefore, exhibit very high resistance. When subjected to light, the electrons are liberated and the material becomes more conducting. When the light is switched off, the electrons are again recaptured and the material becomes less conducting or an insulator. The typical dark resistance of LDRs is 1 MOhms to 10 MOhms. Its light resistance is 75 to 300 ohms. The LDRs take some finite time to change its state and this time is called the *recovery time*. The typical recovery rate is 200 kOhms/sec.

2.5 Thermistors

Thermistors are resistors with a high temperature co-efficient of resistance. Thermistors with negative temperature co-efficient (fall in resistance value with an increase in temperature) are the most popular. They are oxides of certain metals like manganese, cobalt and nickel. Thermistors are available in a wide variety of shapes and forms suitable for use in different applications. They are available in the form of disks, beads or cylindrical rods. Thermistors have inherently non-linear resistance-temperature characteristics. However, with a proper selection of series and parallel resistors, it is possible to get a nearly linear response of resistance change with temperature over a limited range.

Thermistors with a positive thermo-resistive co-efficient are called *posistors*. They are made from barium titanate ceramic and are characterized by an extremely large resistance change in a small temperature span.

Thermistors have various applications such as excess current limiters, temperature sensors, protection devices against over-heating in all kinds of appliances such as electric motors, washing machines and alarm installations, etc. They are also used as thermostats, time delay devices and compensation resistors. Depending upon the application, the thermistor beads need to be properly protected by sealing them into the tip of a glass tube or placing them inside a stainless steel cover.

2.6 Capacitors

A capacitor, like a resistor, is also a passive component, which can be used to store electrical charge. Capacitors find widespread applications in the electrical and electronics fields in the form of:

- Ripple filters in power supplies;
- Tuning resonant circuits, oscillator circuits;
- Timing elements in multi-vibrators, delay circuits;
- Coupling in amplifiers;
- De-coupling in power supplies and amplifiers; and
- Spark suppression on contacts on thermostats and relays.

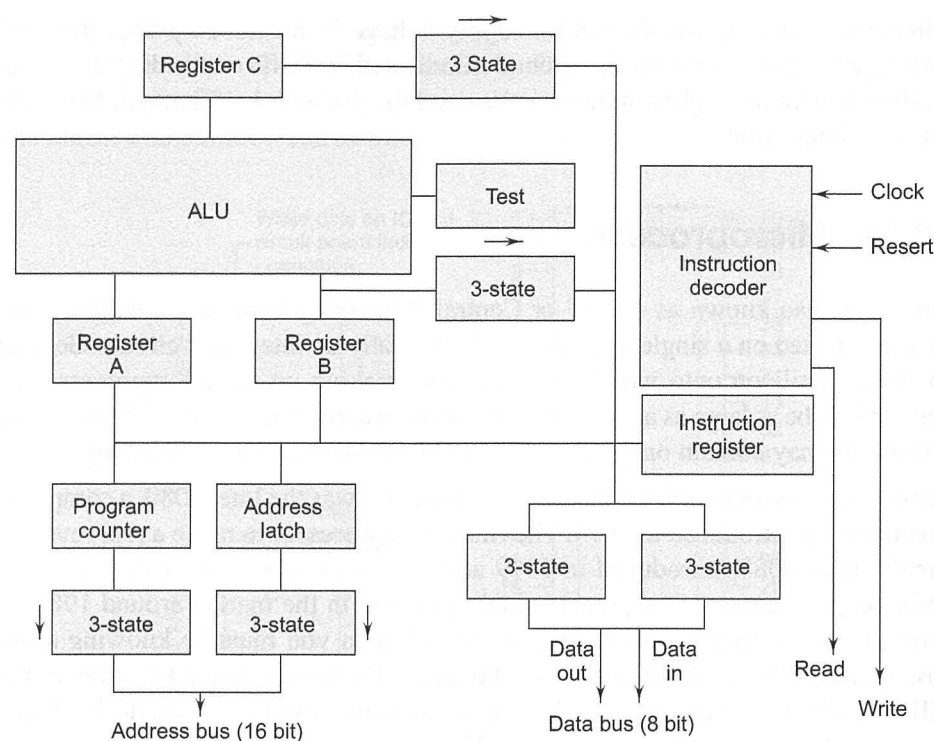


Fig. 2.69 Simplified block diagram of a microprocessor

Most practical microprocessors use 40 or more pins, and even keeping to that number requires that the eight data pins be used for both reading and writing. Figure 2.70 shows the representation of a typical CPU and the different types of packages in which they are available.

The function of the microprocessor is to manipulate data in accordance with the instructions stored in the memory. For this, the microprocessor transfers data and internal state information via an 8-bit bi-directional three-state data bus. Memory and peripheral device addresses are transmitted over a separate 16-bit three-state address bus. Timing and control outputs are given out for synchronization and control inputs like reset, hold and, ready and interrupt are used to perform specific functions.

In the context of microprocessors, the following terms must be understood:

- *Microns:* This refers to the width, in microns, of the smallest wire on the chip. For comparison, a human hair is 100 microns thick. As the feature size on the chip goes down, the number of transistors rises. Modern microprocessors are based on 0.25 micron or less technology.
- *Clock speed:* This is the maximum rate that the chip can be clocked. The clock speed of modern microprocessors is over 2 GHz.

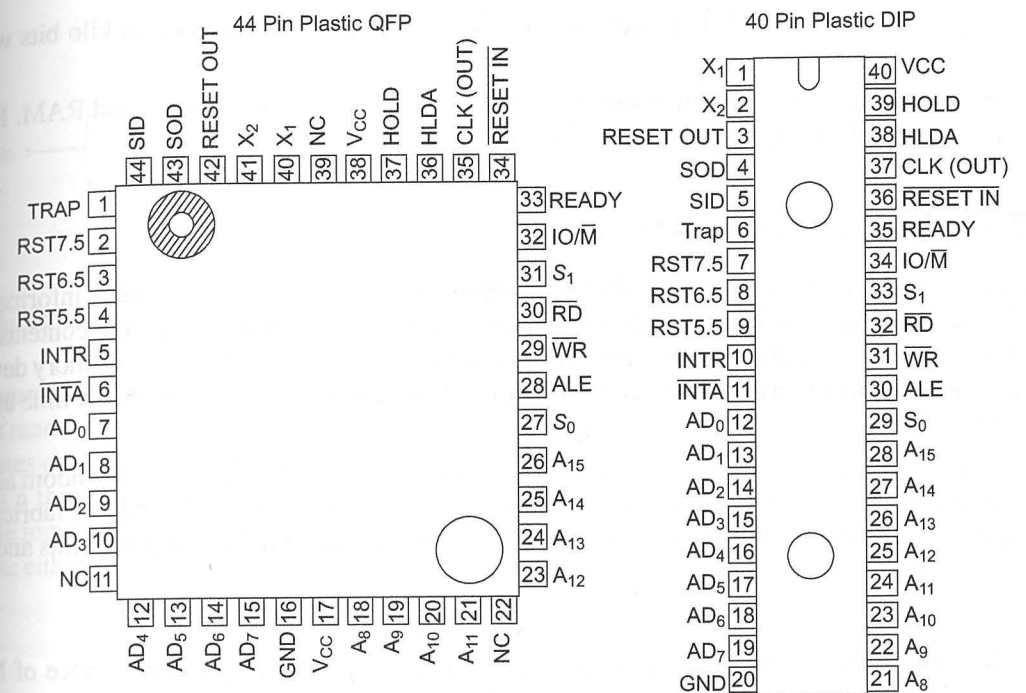


Fig. 2.70 Typical CPU pin-configuration (Intel 80C85) and different packages of 80C85

- *Data Width:* This is the width of the ALU. An 8-bit ALU can add/subtract/multiply/etc. two 8-bit numbers, while a 32-bit ALU can manipulate 32-bit numbers. An 8-bit ALU would have to execute four instructions to add two 32-bit numbers, while a 32-bit ALU can do it in one instruction. In many cases, the external data bus is of the same width as the ALU, but not always. The 8088 had a 16-bit ALU and an 8-bit bus, while the modern Pentiums fetch data 64 bits at a time for their 32-bit ALUs.
- *MIPS:* This stands for Millions of Instructions Per Second, and is a measure of the performance of a CPU. The MIPS value of modern processors is around 1000, as compared to 1 MIPS for an 80286 processor.

2.17 Semiconductor Memories

Memory technically implies any form of electronics storage. A digital memory is an array of binary storage elements arranged in a manner that it can be externally accessed. The memory array is organized as a set of memory words. Each word consists of a number of single bit storage elements called *memory cells*. The word length of a memory word is typically one, four or eight memory cells. Therefore, 1 bit, 4 bits or 8 bits (byte) of information can be stored by the memory word respectively. The memory capacity is the product of the number of memory words and the number



2.18 Microcontrollers

A microcontroller is basically a single-chip microcomputer provided in a single integrated circuit package, which contains a CPU, clock circuitry, ROM (flash), RAM, serial port, timer/counter and I/O circuitry. As such, unlike conventional microprocessors it does not require a host of associated chips for its operation.

Most of the microcontrollers come in 40-pin DIP packages; the pin-out consists essentially of upto 32 I/O lines with the remainder being used for power, reset interrupt and timing. The instruction set of a single chip microcomputer generally bears a close resemblance to that of the microprocessor family to which the microcontroller belongs.

Microcontrollers have been available for a long time. Intel introduced 8048 (MCS-48, an 8-bit microcontroller) in 1976. It has developed upwards and its new version is the 8051 family, which is a high performance 40-pin DIP package. Microcontroller's functionality has increased tremendously in recent years. Today, one gets microcontrollers which are stand-alone for applications in data acquisition systems and control. They have A/D converters on the chip which enable their direct use in instrumentation.

One of the popular microcontrollers available in the CMOS technology is 89C51, which has 4K bytes of Flash programmable and erasable read-only memory (EEPROM). Its instruction set and pin-out are compatible with industry standard MCS-51. Flash allows the programme memory to be re-programmed in-system or by a conventional non-volatile memory programmer. This microcontroller is available in both DIP or QUAD package.



2.19 Surface Mount Devices

The conventional through-hole mounting technology used for printed circuit assemblies is being increasingly superseded by surface mount technology. Instead of inserting leaded components through the holes, special miniaturized components are directly attached and soldered to the printed circuit board. The surface mounted components and their packing are particularly suitable for automatic assembly. The advantages of surface mounting are rationalized production, reduced board size and increased reliability. The prime motivation for introducing surface mount technology (SMT) is, thus, density increase and board area reduction due to continuous demand and market trend for miniaturization in electronic assembly, particularly in portable products.

Figure 2.76 shows the conventional through-hole technology in which the components are placed on one PCB side (component side) and soldered on the other side. In surface mount technology, the components can be assembled on both sides of the board (Figure 2.77). The components are attached to the PCB by solder paste or non-conductive glue and then soldered.

In addition, there are hybrid circuits consisting of thick and thin film circuits, which are basically leadless components. They are reflow soldered on to the ceramic or glass substrate in addition to the components already integrated on the substrate. The mounting of the hybrid circuits is shown in Figure 2.78.

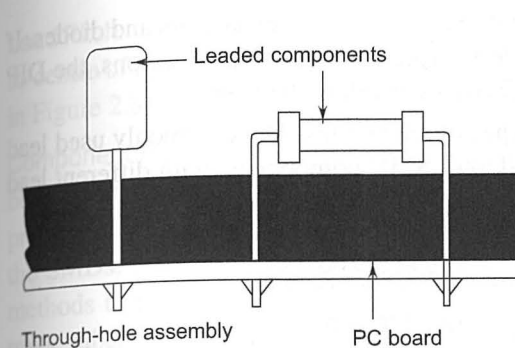


Fig. 2.76 Conventional through-hole technology-components are placed on one side and soldered on the other

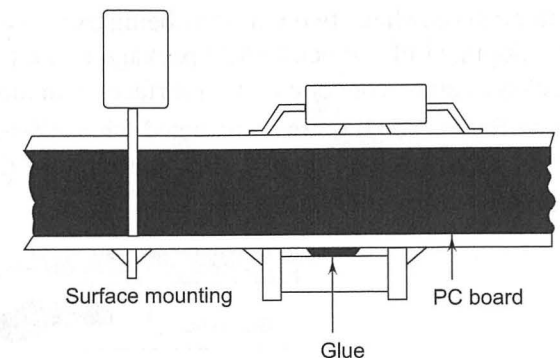


Fig. 2.77 Surface-mount technology-components are mounted on both the sides

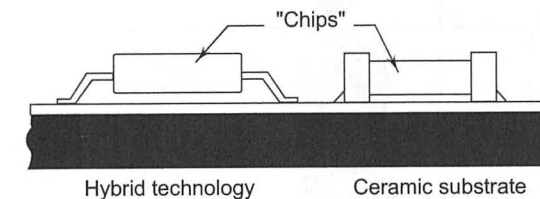


Fig. 2.78 Hybrid technology-consisting of components integrated on the substrate and soldered chips

Since not all component types are available in the surface mount version, a combination of leaded and surface mounted components is generally seen in manufactured electronic equipment.

The 100 mil lead spacing used in DIL packages has a severe limitation in terms of the number of pins that could be accommodated in high density VLSI circuits, with 64 being the limit. The development of miniature systems has forced new package types to appear specially in the surface mount type. The small outline (SO) package with gull leads is available in which pins are on a 50 mil grid. For higher pin counts, the quad package with J leads has become a popular package in the industry. With the pin numbers going up to 300+, the pin spacing has been decreasing from 50 to 30 and then to 25 mil.

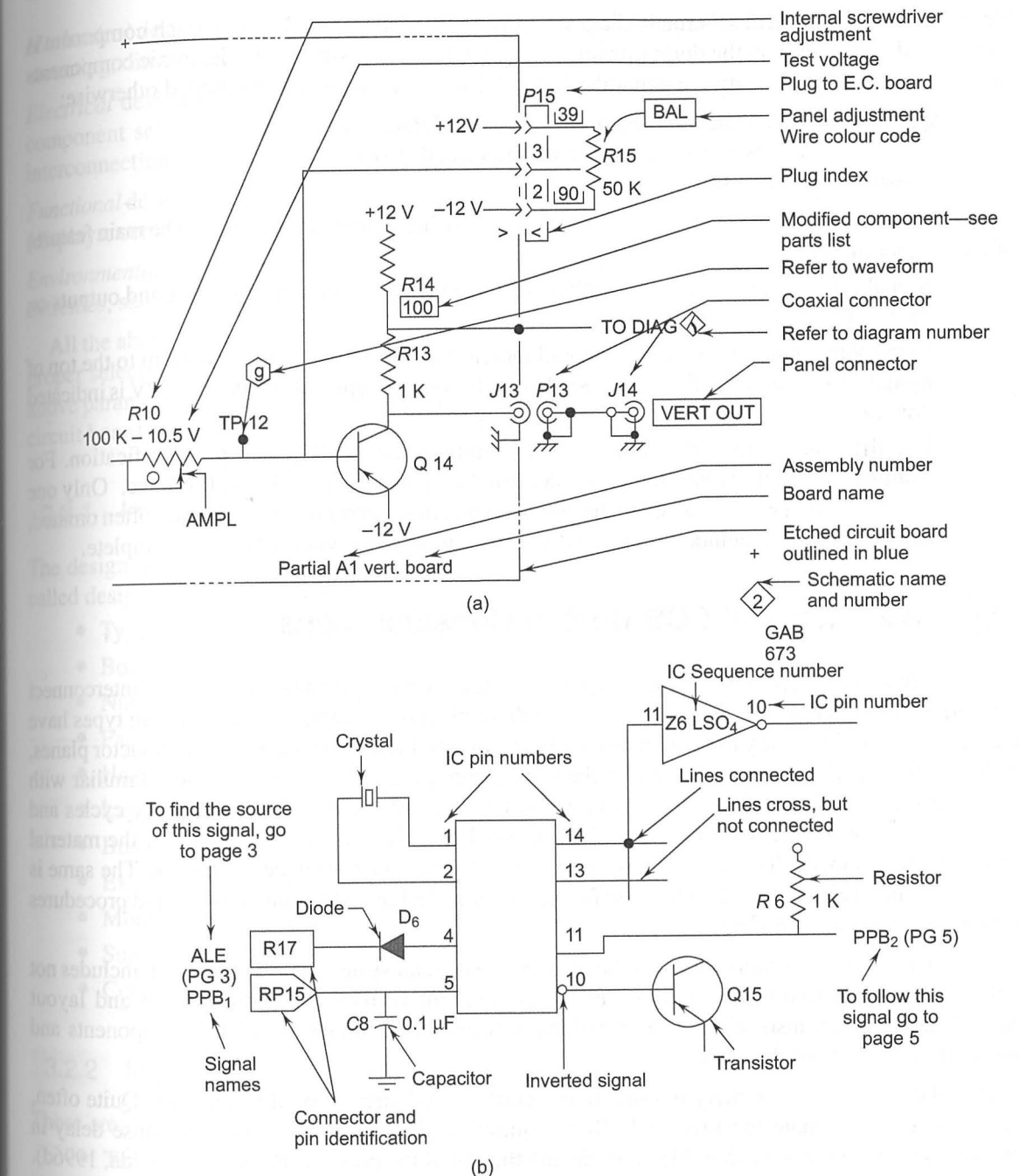
2.19.1 Surface Mount Devices

The abbreviation SMD for Surface Mount Device is the most common designation for the components used in surface mount technology. SMDs are designed with soldering pads or short leads, and are much smaller than comparable leaded components. In contrast to conventional components, the leads of which must be inserted into holes, SMDs are directly attached to the surface of the PCB and then soldered.

Resistors, ceramic capacitors and discrete semiconductors represent 80 per cent of the total available SMDs. Normally, in the SMDs, the cubic shape prevails over cylindrical versions, as the

Table 3.1 Reference Designators (Reference: Component Identification, IPC-DRM-18 F, Desk Reference Manual)

Component	ANSI/IEEE	IE C	Component	ANSI/IEEE	IEC
Amplifier	AR	A	Jumper	W,P or R	
Capacitor	C		Microprocessor	U	
Pack	C				G
network	C				
Polarized	C		Oscillator	Y (crystal or G (other)	
Variable	C		Relay	K	
Connector	J or P		Resistor	R	
Crystal	Y	B	Pack	R	
			network	R	
Delay Line	DL		Potentiometer	R	
			Variable	R	
Diode	D or CR	V			
Light Emitting Diode	DS (display)	E	Thermistor	RT	
Voltage Rectifier	D or CR	V	Varistor		
Zener Diode	D or VR	V	Asymmetrical	D or CR	
			Symmetrical	RV	
Filter	FL	Z	Socket	X, XAR, XU, XQ, etc.	
Fuse	F				
Header	J or P		Switch	S	
Inductor, Choke	L		Test Point or Pin	TP	V
Integrated Circuit	U, IC		Transistor	Q	
Insulated Jumper	W or P		Transformer	T	
			Voltage Regulator	VR	
Battery	BT				
Meter, Instrument	M		Antenna, Spark Gap, Shield	E	
Plug, Connector Male	P		Attenuator	AT	
Power Supply	PS		Motor, Fan, Synchro	B	
Test Point	TP				

**Fig. 3.2** Schematic circuit with symbols as per the American National Standard Institute (a) Circuit with discrete components (b) Circuit with integrated circuit

stray capacitance is unimportant. On the other hand, in case of high impedance signal lines, where stray capacitance is required to be limited to a low value, narrow conductors are used. Normally, the conductor width should be as generous as possible to take care of any variation in the etching process as well as any scratches in the artwork caused unintentionally. Figure 3.10 shows the current carrying capacity of etched copper conductors for rigid boards. Generally, for 35 μm and 70 μm thickness conductors, a nominal 10 per cent de-rating is allowed to provide for normal variations in etching methods.

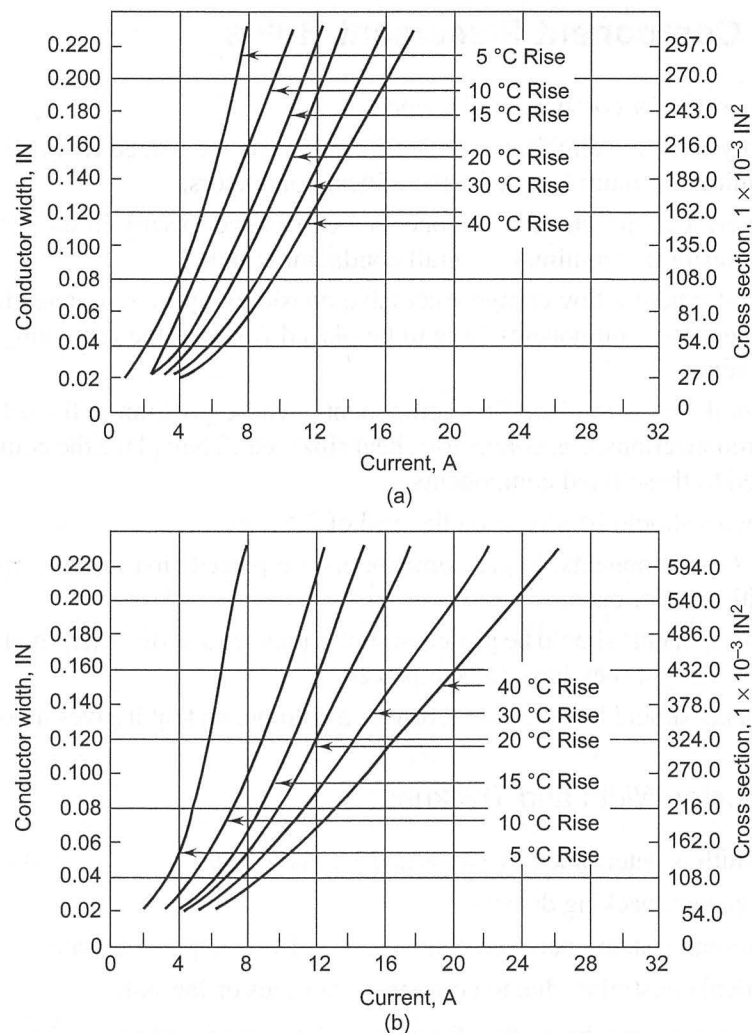


Fig. 3.10 Temperature rise versus current for (a) 1 oz copper (b) 2 oz copper

The chemical and photographic processes used to produce a PCB put requirements on the minimum width of trace (conductor) and the minimum spacing between traces. If a trace is made smaller than this minimum width, there is a chance that it will open (no connection) when manufactured. If two traces are closer together than the minimum spacing, there is some chance that they will short when manufactured. These parameters are usually specified as X/Y rules, where X is the minimum trace width and Y is the minimum trace spacing. For example, 8/10 rules would indicate 8 mil minimum trace width and 10 mil minimum trace spacing. Typical modern process rules are 8/8 rules with values as small as 2/2 rules being available.

3.6.2 Conductor Spacing

The conductor spacing considerations are generally based on voltage breakdown or flashover between adjacent conductors. The conductor spacing is determined by the peak voltage difference between adjacent conductors, capacitive coupling parameters and the use of a coating. The recommended conductor spacing designs are:

General designs: 0.025 mm voltage limitation 400 V dc high density:

Preferred ac or peak minimum is 0.015 mm, voltage limitation 50V dc or ac peak.

The minimum by exception is 0.010 mm.

In case of preferred minimum or minimum by exception designs, the printed board assembly must be conformal coated or a suitable solder mask must be applied.

The minimum permissible spacing should be applied only where there is no way to avoid it, otherwise higher spacing should be given. This is to minimize the reject rate during production.

Additional considerations in deciding conductor spacing are:

- Critical-impedance or high frequency components should be placed very close together to reduce critical stage delay.
- Transformers and inductive elements should be isolated to prevent coupling.
- Inductive signal paths should cross at right angles.
- Components which produce any electrical noise from movements of magnetic fields should be isolated.

There are no standards for minimum spacing between any two leads that are not electrically connected. However, an average 2 mm is generally taken where voltages are in excess of 30 volts or where a short will cause failure of critical nature or expensive components, when an increase in spacing becomes necessary.

Most modern circuits make use of semiconductor devices operating at voltages normally upto 24 volts. Therefore, the problem of breakdown due to narrow spacing between conductors almost does not exist. However, for a normal uncoated epoxy card, operating at an altitude of less than 1000

The various methods used for artwork preparation are discussed below.

5.2.1 Ink Drawing on White Card Board Sheets

This is the simplest and earliest method used for PCB artwork design. The materials required are a white cardboard paper, good quality Indian ink and an ink pen. A paper with blue lines grid is usually preferred as these lines do not get reproduced in the photographic process for film master production. Suitable polyester foil, which is dimensionally more stable than cardboard paper, can also be used for drawing the ink pattern. However, the one side of the foil should have a rough or mat surface so that ink will stick to it.

In this technique, circumferences of the solder pads and the centre holes are drawn with a drawing compass and the spaces in between are filled with ink. Conductors are drawn with an ink pen either by directly providing the desired width or as double lines, which are thereafter filled with ink.

Templates in the scale of 1:1, 2:1 and 4:1 are available, which serve as useful aids for drawing of the solder pads. Since this is a totally manual method, it suffers from drawing inaccuracies. For example, conductor widths may vary at least by 0.1-0.2 mm and solder pad locations and conductors can easily be displaced by 0.3-0.5 mm. Also the dimensional instability of the cardboard, 0.01 % °C with respect to temperature and 0.005 %RH (relative humidity) with respect to changes in relative humidity, could be a source of error in high accuracy boards. The method is now rarely used because of the poor stability and precision results.

5.2.2 Black Taping on Transparent Base Foil

Manual artwork generation has become very convenient since the advent of self-adhesive or transfer type pads and precision tapes, and no longer relies on the high drawing skills and patience of the draftsman.

Self-Adhesive Tapes

They are available in a wide range of widths, which can be selected depending upon the application. They are supplied in rolls with standard widths of 0.5 mm, 1 mm, 2 mm, 4 mm and 8 mm; etc. (Figure 5.1a). The width tolerance can be 0.05-0.1 mm depending upon the tape width and the manufacturer.

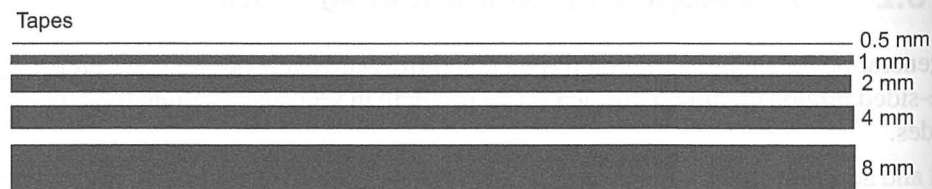


Fig. 5.1(a) Sizes of tapes

Self-Adhesive Pads

Pads are available in standard shapes such as donut, square, hex, oval, and tear drop and triangular. These are shown in Figure 5.1b. The two different types of pads which are commonly used are: Self-adhesive type and the Transfer Type.

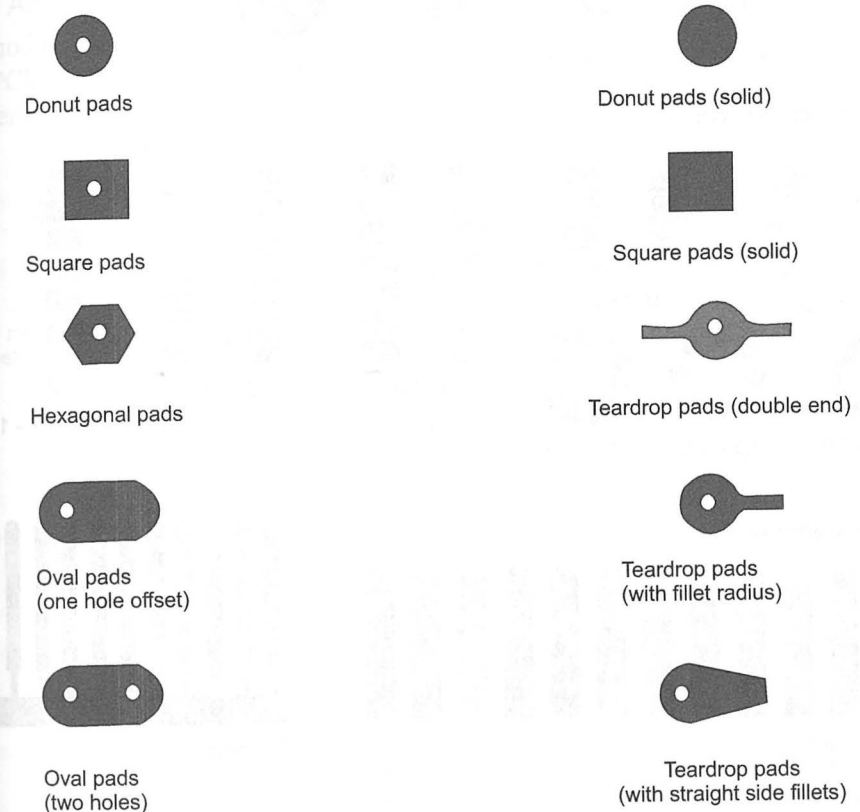


Fig. 5.1 (b) Standard pad shapes, all pads are 1.27 mm

Transfer Pads

They are printed on a thin adhesive film of typically 10 µm thickness, which is mounted on the top side against a transparent carrier strip. The pad can be transferred from the carrier strip onto the artwork base (polyester base sheet) by just rubbing with a wooden stick end or pencil. The carrier strip can then be lifted from the artwork base leaving behind the pad.

In order to minimize the artwork preparation time and to achieve accuracy in positioning, pads in the form of solder pads, pre-arranged patterns like integrated circuit footprints, contiguous patterns for connectors and special patterns for TO casing components can be used. These pre-arranged patterns are usually for the multi-lead components and are generally referred to as footprints: for

Table 5.4 Recommended Nominal Values (mm) for BGA (Courtesy Micron Technology Inc.)

BGA pitch	BGA pad size	Ball diameter*	PCB pad size	Foil thickness	Square aperture
1.25	0.60	0.75	0.60	0.127	0.60
1.00	0.33	0.40	0.33	0.102	0.33
0.80	0.33	0.40	0.33	0.102	0.33
0.75	0.27	0.35	0.27	0.102	0.27

Note *Ball diameter measured prior to mounting and re-flow.

5.3.5 The Square Land/Pad

The square land/pad is a common method to designate the polarity or orientation of a component. For polarized components that have positive or anode lead marked, the square land typically indicates where the positive lead should be placed. For components which have the negative or cathode lead marked, such as diodes or LEDs, the square land indicates where the marked negative lead should be placed.

Similarly, the square land is often used to indicate where the marked lead or pin-1 of a multi-pinned component should be placed. This facilitates proper orientation of the components on the surface board.

In case of integrated circuits, the orientation symbol is usually Pin-1 of the IC. In such cases matching orientation marks are made on the printed circuit board with silk screened symbols and markings.



5.4 Artwork Generation Guidelines

5.4.1 No Conductor Zone

Traces should not come any closer to the edge of the board than 0.50 inches (12.5mm) at a scale of 1:1 and if possible, at least 0.10 inches (2.5 mm) should be left. This gives the necessary tolerance that is required to shear the board to the specified size; otherwise a portion or all the trace could get sheared off or shorted out at the next assembly.

If metal or electrically conductive card guides are used, then the gap from the edge of the card guide to the edge of the nearest conductor should be the standard required air gap 1.25 mm (0.05 in. or greater). This is shown in Figure 5.15.

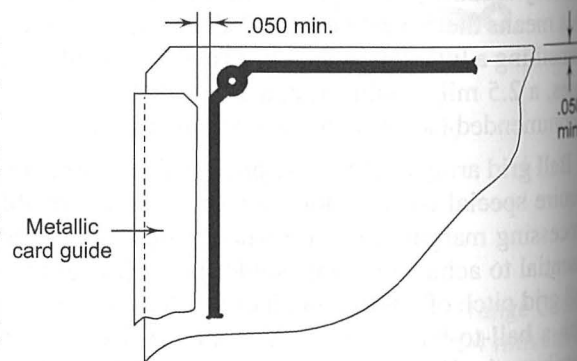


Fig. 5.15 Spacing for conductive card guides

5.4.2 Pad Centre Holes

The pad centre holes are always kept free, i.e. they should not be covered by tapes. Only this can enable them to act as registration marks and in artwork design and as a centring aid in the drilling operation.

5.4.3 Conductor and Solder Pad Joints

For ensuring reliability of the solder joint, it is important that the pattern around hole should be maintained as uniform and as small as possible to enable symmetrical solder joints. The overlap should be such that even if the tape creeps, there is no gap between the pad and the condition. Figure 5.16 shows examples of some bad solder joints and the suggested improvements.

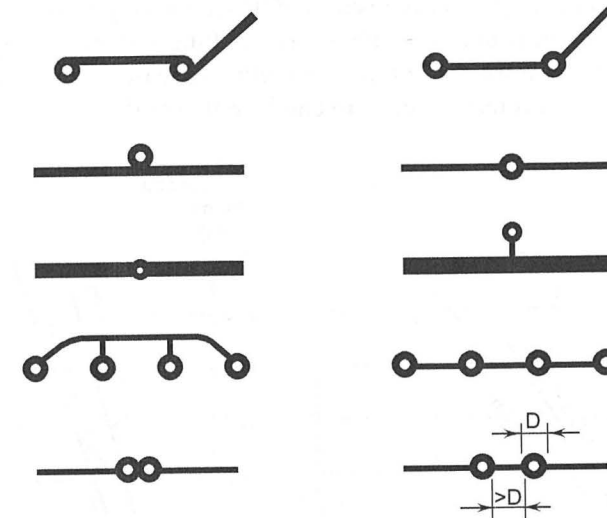


Fig. 5.16 Guidelines for the design of reliable solder joints

5.5 Film Master Preparation

The components and the various connections according to circuit design are done in the artwork. Film master is prepared from the artwork. It is the film negative or film positive, which is finally used for the direct exposure of the photo-resist coated PCB or the light-sensitized screen. In order to ensure PCBs of high quality, the film master must have high dimensional accuracy, sharpness and wear-out resistance. The imperfections of the film master get materially reflected in each PCB made with it afterwards. The final quality of the film master, in general, depends upon the chemicals, the film emulsion and exposure units such as cameras.

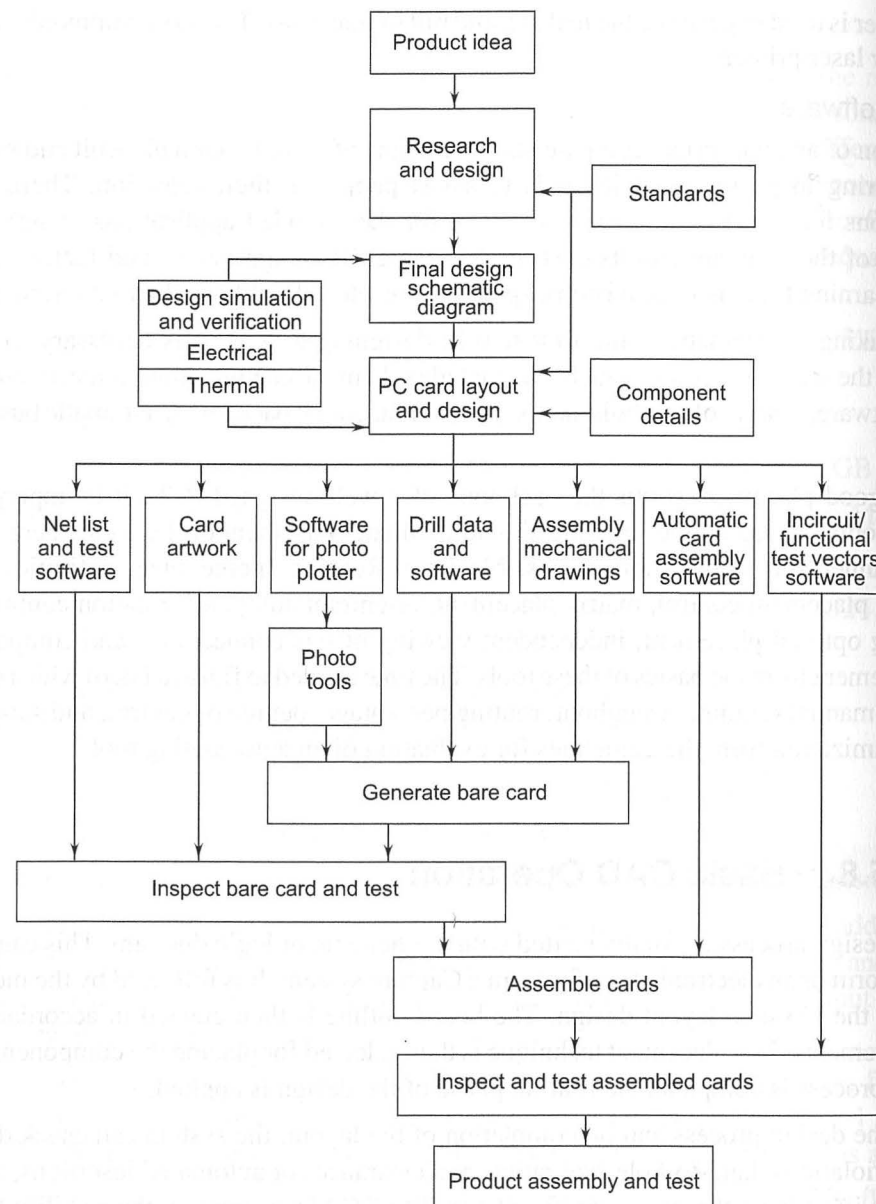


Fig. 5.26 Steps in the design and production of a board showing the wide ranging information required from a designer to manufacture and assemble a board

Schematic Diagram: This provides the functional flow and graphical representation of an electronic circuit and consists of electrical connections (nets) and junctions. The schematic diagram consists of:

- Symbols representing discrete components like resistors, capacitors, transistors and integrated circuits; symbols to follow international standards;
- Input/output connectors;
- Buses;
- Power and ground symbols;
- Component reference names; and
- Text supporting the diagram.

Layout: This involves decision-making pertaining to:

- Surface mounts or mixed technology;
- Single, double or multi-layer boards;
- PTH or non-PTH;
- Placement of components, vias, lands, test pads and device footprints; and
- Generation of interconnection tracks, etc.

Design: Design considerations should be decided in parallel with the layout. The following factors must be taken care of:

- Track widths to ensure that the current densities are not excessive;
- Adequate track separation so that there is no electrical breakdown;
- Proper thermal design to avoid any hot spots on the board; and
- Effect of stray parasites, particularly stray capacitances in case of high frequency circuits, etc.

Netlist: This includes:

- Generation of software list to show all interconnection paths; and
- Comparison with the netlist generated by the schematic diagram to check the accuracy of interconnection lines.

Card Artwork: This involves:

- Test plot of the final card layout;
- Manual inspection of components and visual inspection of the card; and
- Generation of bare board testing information from via information and the netlist.

Photo-tooling: This implies:

- Generation of photo negative or positive.

Drill data: This involves:

- Specifying drill sizes;
- Ensuring that the drills are in the correct position; and
- Software to drive numerically controlled (NC) drilling machine.

Assembly Drawing: This is required to:

- Provide mechanical design information, specifically card material, thickness, overall dimension and shape;

- Check the datum position; and
- Fix the component sizes and their positions.

Automatic Card Assembly: This is needed to:

- Drive radial, axial and integrated circuit insertion machines; and
- Pick and place machines.

In circuit/function Testing: This involves:

- Test vector generation; and
- Final testing and verification.

Figure 5.27 shows the traditional design flow in CAD systems.

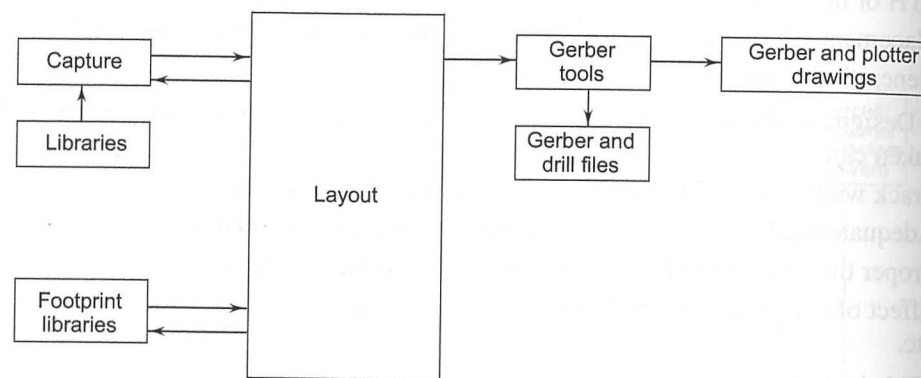


Fig. 5.27 Typical design flow in CAD systems

5.8.1 Layout Procedure

Before the layout procedure is started, complete and detailed specifications on the electronic circuit are required. This includes the following material:

- Schematic diagram with component details, interconnections, and edge connector specifications;
- Component list including component name, type specifications, type number, manufacturer;
- Mechanical specifications such as board size and shape, mounting holes, identification of restricted areas with respect to component height, edge connector location, etc;
- PCB specifications, whether single-sided; double-sided or multi-layer board, plated-through holes;
- Pattern specifications on pad type and size, conductor widths, spacing;
- Electrical specifications like restrictions on component placement because of heat production, capacitive or inductive coupling, ground planes, critical-length interconnections, etc.; and
- Data operation.

First of all, a component library is prepared describing each component package type used with its outline, pad type, and size and pad position. In the complete component list, the package type, component name orientation and side of interconnection on the PCB are identified for each component used. The connection list gives an exact account of all the interconnections in point-to-point form. The list of board details finally contains the board information and the x/y co-ordinates of the board corners. The correct entry of these data and, in particular, of the connection list, is of utmost importance. In many CAD systems, there is a provision to enter two connection lists for the same circuit but prepared by two independent operators. Any further processing of the data is possible only after the discrepancies between the two connections lists are sorted out.

5.8.1.1 Entry of Schematic Diagram

As explained in Chapter 3, a schematic diagram provides the functional flow and the graphical representation of an electronic circuit. The entry of the schematic diagram is the first step in PCB design using a CAD system. A schematic diagram consists of:

- Electrical connections (nets);
- Junctions;
- Integrated circuits symbols;
- Discrete component symbols like resistors, capacitors and transistors, etc.;
- Input/output connectors;
- Power and ground symbols;
- Buses;
- No-connection symbols;
- Component reference names; and
- Text.

It is desirable that the symbols used in a schematic diagram follow the international standards.

In schematic capture, each design that you open is in a separate *project manager window*. If you need to work simultaneously with several designs, you can open them all, and each will have its own project manager window. The project manager is used to collect and organize all the resources you need for your project. These resources include schematic folders, schematic pages, part libraries, parts, and output reports such as bills of materials and netlists. A project manager doesn't actually contain all the resources. It merely "points to" the various files that the project uses. For this reason, be sure that you don't move or delete any files referenced by a project. If you do, the project won't be able to find them.

The project file is saved as an ASCII-file (American Standard Code for Information Interchange), and can be viewed in any text editor.

5.8.2 Library Manager

The library manager contains all the files for the component footprints. These footprints are used to design PCB. The layout provides the facility to create a footprint library for component footprints. Libraries also contain a variety of symbols that you can use in your boards. The layout has more than 30,000 footprints.

Footprints:

Footprints refer to the physical description of components. A footprint generally consists of the following three types of objects:

- Pad stacks;
- Obstacles, representing the physical outline of the component, silkscreen outline, assembly outline, and placement and insertion outline; and
- Text.

A library footprint can be viewed as a graphical display wherein you can perform various actions on the footprint, such as editing, saving, copying and deleting it.

Creating New Library Modules

Using the library manager, you can create a new library module by saving new or existing footprints to a library that you name. You can then add other footprints by selecting them in the footprints list and saving them to the newly created library module. You can also create new footprints and add them to the libraries.

Similarly, you can add pins to a footprint. Pins can be numeric or alphanumeric, and placed in any order. By default, layout names the pins in numerical order beginning with the number 1. You must change the pin names in the layout to match the pin numbers in the schematic, or change them in the schematic library.

Assigning Pad Stacks to Footprint Pins

Pad stacks define the pins on each layer of the footprint. Pad stacks show the shape and size of each pin. You can assign the same pad stack to all the pins in the footprint using the edit footprint dialog box.

5.8.3 Component Placement

After the input data have been fully accepted by the computer, component placement is carried out. Those components for which the position has earlier been determined by the designer will get their place accordingly. The other components are usually placed in a manner that ensures minimum overall conductor length. Two placement procedures are possible: either the complete placement can be fully guided by the operator, or it can just be attempted by the automatic placement routine with the intervention of the operator, wherever necessary.

This part of the software enables one to place as well as route the components and set unit of

measurements, grids, and spacing. For the placement and routing of the components, the Auto-placement and Auto-routing facility is normally used. However, in many softwares, some critical signals have to be routed manually before Auto-routing. The following steps are taken for board design:

- Create a netlist from the schematic diagram by using capture. Netlist file is a document file which contains information about the logical interconnections between signals and pins. A Netlist file consists of nets, pins, components, connectors, junctions, no-connection symbols, and power and ground symbols. Before you create a netlist file, be sure that your project is completed, annotated and free from electrical rule violations.
- Software includes design rules in order to guide logical placement and routing, which means, that you must load the netlist into layout to create the board by specifying the board parameters.
- Specify board parameters. Specify the global setting for the board, including units of measurements, grids, and spacing, create a board outline and define the layer stacks, pad stacks and vias.
- Place components. Use the component tool in order to manually place the components which are fixed by the system designer on the board or otherwise use Auto-placement.
- Route the board. Use different routing technologies to route the board and take advantage of push-and shove (a routing technology), which moves tracks to make room for the track that you are currently routing, and you can also auto-route the board.
- Provide finishing of the board. The system supplies an ordered progression of commands on the Auto menu for finishing your design. These commands include Design Rule Check, Clean-up design, Re-name Components, Back Annotate, Run Post Processor, and Create Reports.

5.8.3.1 Manual Placement

The graphic display shows the board contour as specified in the board list. At the same time, all the components are shown with their outlines in one corner of the display including the straight pin-to-pin interconnections.

With the help of the light pen, components can be called up one by one, and moved across the display to the position found to be the most suitable. The light pen is a hand-held electronic device. If held close to the display, its position can be detected by the computer with a special program. When components are moved, the interconnections simultaneously follow in straight lines like elastic bands.

5.8.3.2 Automatic Placement with Operator's Interaction

With the computer's placement routine, it is possible to attempt the optimum component placement by automatic means. The components having the most interconnections are placed together in groups while the components with pre-determined locations are placed accordingly. The component allocation occurs now by minimizing the overall conductor length and can be observed on the display. The huge amount of data to be processed and considered by the automatic placement routine (Figure 5.28) makes it very difficult to find the optimum solution. Since the designer has the option to interact, an acceptable solution can be found with combined efforts.

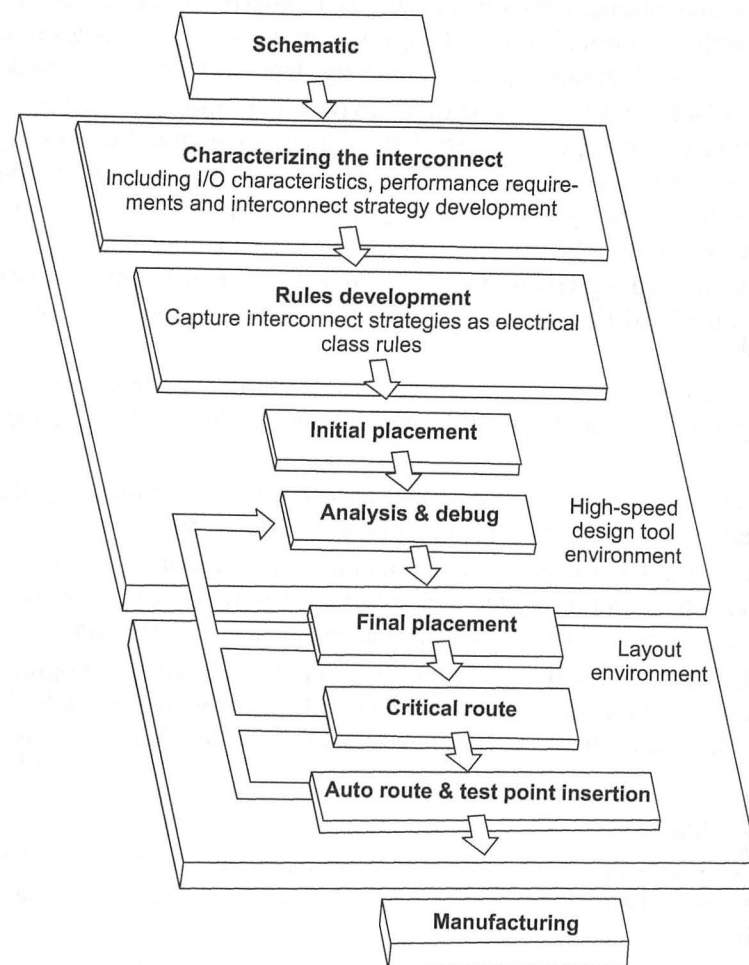


Fig. 5.28 High-speed design tools: physical design process (after Isaac, 1995)

Most automatic placement routines or algorithms automatically place the components at strictly defined locations on the printed board. Other systems allow random positioning of components, restricting only component body interference. All systems attempt to minimize the length of the connection paths (Ginsberg, 1992a). The most commonly used automatic placement techniques are:

- **Swapping:** A possible feature of CAD systems is the ability to swap the positions of similar components so the possibility of 100 per cent interconnection is enhanced. Some auto-placement packages provide the capability to swap parts, gates and pins as shown in Figure 5.29a.

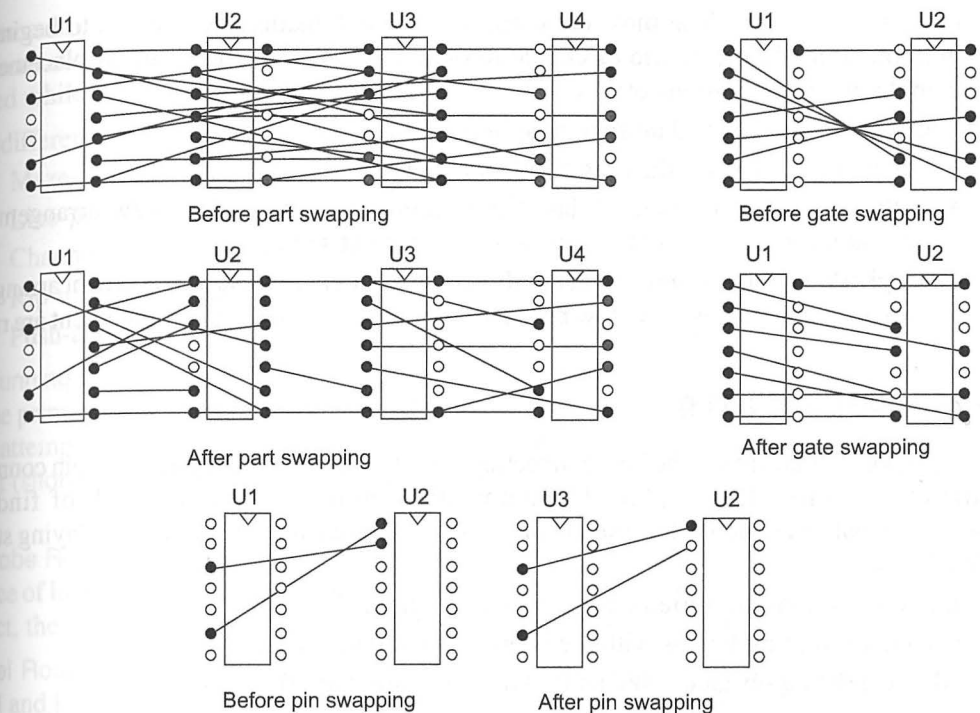


Fig. 5.29(a) Swapping of parts, gates and pins in CAD systems (after Ginsberg, 1992 a)

Components and pins can be swapped automatically to obtain the shortest possible overall wire length; components, gates and pins can be swapped interactively to minimize routing complexity and to shorten overall wire length.

- **Moving/rotating:** This is usually an interactive design tool accessed from a menu during the design placement phase. It enables the designer to move and/or rotate any individual part in the design. Smarter software packages also have the ability to move and rotate a complete group, or to rotate a complete component type.
- **Partitioning:** In this technique, the design process is reduced to a sequence of partitioned problems. First, a vertical boundary line splits the set of component locations into left and right halves. The set of components is then partitioned into halves; the partitioning procedure swaps component groups to minimize the crossing count for the vertical boundary line.

Next, the set of locations is split into upper and lower halves separated by a horizontal boundary line, and the same component swapping method is used. Each component now has been placed in either the left-upper, right-upper, left-lower or right-lower quadrant. The partitioning technique is applied to each of the quadrants in turn, and then to the resulting octant until each region contains one board location.

- *Iterative placement*: The most basic approach to automatic placement is to begin with a placement, apply the metric of choice to obtain a "score" for the current placement, and then iterate the operations of:
 - Choosing a new, legal arrangement of devices;
 - Obtaining a score for the new arrangement; and
 - If the new score is "better" than the current score, making the new arrangement the current arrangement and the new score the current score.

Through this technique the designer is able to get a better component placement arrangement. However, the process is very slow because whole choices for a new arrangement are random.

5.8.4 Conductor Routing

After the component placement, the interconnections are still shown as straight pin-to-pin connections in all directions across the display. The automatic routine now has the task of finding an interconnection pattern which is feasible on the PCB. This procedure is executed by laying stress on the following priorities:

- Interconnections are sorted out according to length.
- Conductor routing begins with the shortest interconnections.
- All the existing obstacles like pads and copper areas are sorted out.
- Via holes are minimized by transferring conductors or portions thereof to the optimum board side. (A via hole is a plated through-hole of usually a smaller diameter which serves as an interconnection between the circuit pattern on the two sides of the board, but it is not used for component lead mounting)
- All the information in the computer memory and on the display is continuously updated.

On the display, the patterns of the two different board sides are distinguishable according to different light intensity. The software will attempt to place all the conductors in x direction on one board side while conductors in y direction are placed on the opposite side.

The computer as such will hardly be able to place all the conductors onto the board or it might also find solutions with very long tracks which are very risky because of the possibility of an unwanted coupling effect. Unfinished interconnections (usually about 20 per cent) are displayed as dotted lines. Since the operator has a better overview on the pattern, he can interfere and finish the conductor routing.

5.8.4.1 Auto-Routing

Nowadays CAD systems are able to handle fine line technology, variable trace widths, buried and blind vias as well as multiple land sizes on the same layer with a variable grid. A good routing program attempts to make all the required connections by using the shortest grid. A good routing program attempts to make all the required connections by using the shortest total connection paths and at the same time reducing the number of vias. Many of the Auto-routers are restricted to route between pairs of layers only. For efficient routing, all layers must be taken into consideration

simultaneously. The Auto-router of the particular software should be capable of constructing an intelligent routing pattern while constantly checking clearances and connectivity. All this must be achieved while keeping both routing and circuit costs to a minimum.

The different categories/variations of automatic routers for PCB design in use include:

- Maze-running routers;
- Line-probe routers;
- Channel routers;
- Rip-up routers; and
- Push-and-shove routers, as in 'OrCAD-Layout Plus'.

Maze-running Routers: In this technique, the router makes interconnections by expanding out from the primary grid connection point until the secondary point is found, and connected. The maze runner attempts to complete every possible path, compare all the successful connections and pick the best (shortest path and/or least vias) available. The method usually requires longer processing time.

Line-probe Routers: Here, the router connects a pair of points by constructing simultaneously or sequence of line segments (probes) out from each point to be connected. When the two-line sequences intersect, the connection path is complete. The method gives over 95 per cent completion rates.

Channel Routers: This method involves placing circuit components in such a way as to leave vertical and horizontal channels between the components through which the connection paths may be run. Any two points to be connected must lie in the same channel, column or row, for the router to make the connection. The router proceeds by connecting the shortest paths first and progresses until it has completed all the connections within the channel. It will then move to the next channel and eventually to the other side. This is because the basic router algorithm operates on two-circuit layers, making vertical connections on one side and horizontal connections on the other layer.

Rip-up Routers: The algorithm used in this method divides the circuit board into a grid with each connection point residing in a grid cell. The cells containing connections are called *obstacles* and those without connections are known as *free cells*. The algorithm keeps a track of available cells and finds the shortest path between two connections. The process continues until all connections have been routed. (Ginsberg, 1992c).

Push-and-shove Routers: With this algorithm, the completed paths which form obstacles to subsequent paths are not removed but are re-routed. Instead, the path is completed by pushing or shoving small sections of previously routed conductors in order to make additional routing channels. This technique is commonly used by human designers and is shown in Figure 5.29b.

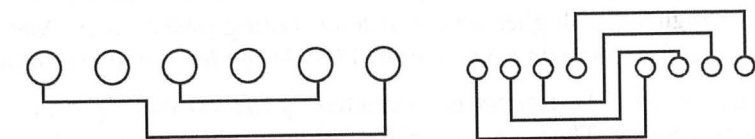


Fig. 5.29 (b) Push and shove router

The performance of an auto-routing algorithm is usually measured against the percentage of routing completed or in other words, the amount of manual effort needed to complete a job. The execution time and system resources required are also important considerations. Figure 5.30 illustrates completion rates based on density and routing completion time. The best algorithms are designed to balance both economic and technical factors. The system usually allows the user to complete the unfinished connections, while it monitors his action against the ground rules for interconnections and signals him when any rules are violated. The re-entry capability helps the designer to make changes on previously laid out designs in terms of movements, and the deletion and addition of conductors.

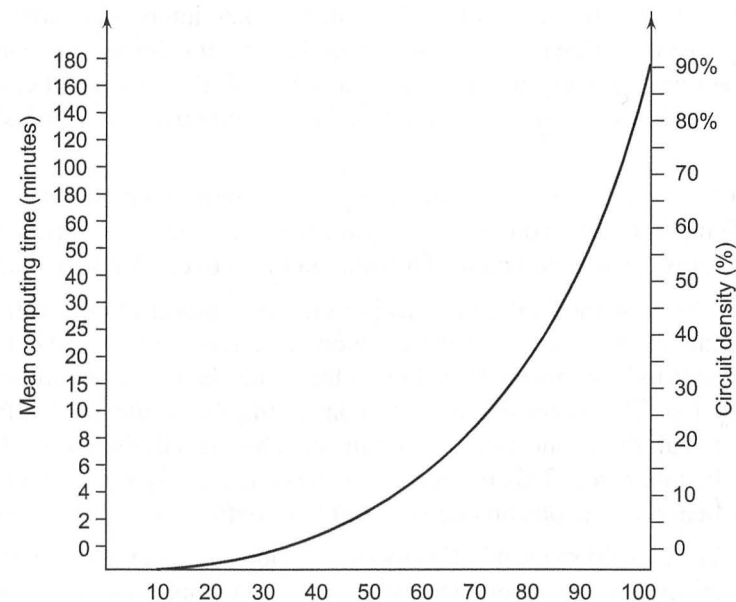


Fig. 5.30 Completion rates based on circuit density and routing completion time (redrawn after Herrmann, 1992)

As no router gives 100 per cent auto-routing on every board, this needs to be completed by manual means. Generally, the last few routes are the most difficult to connect and require changes in earlier placement. Figure 5.31 illustrates the manual work required as opposed to the number of open connections after auto-routing. It may however, be noted that faster and more powerful microcomputers are bringing artwork routing systems into an affordable price range, which will eventually result in higher and higher levels of auto-routing possibilities. Mantay, et al. (1991) describe a simple model for optimizing auto-routing PCB design for manufacturability and routability.

The wiring space used by the routers is connected by the various types of space organization methods used in auto-routing. These are the grided, gridless, plastic grid, and channel methods and combinations of these (Ginsberg, 1992a).

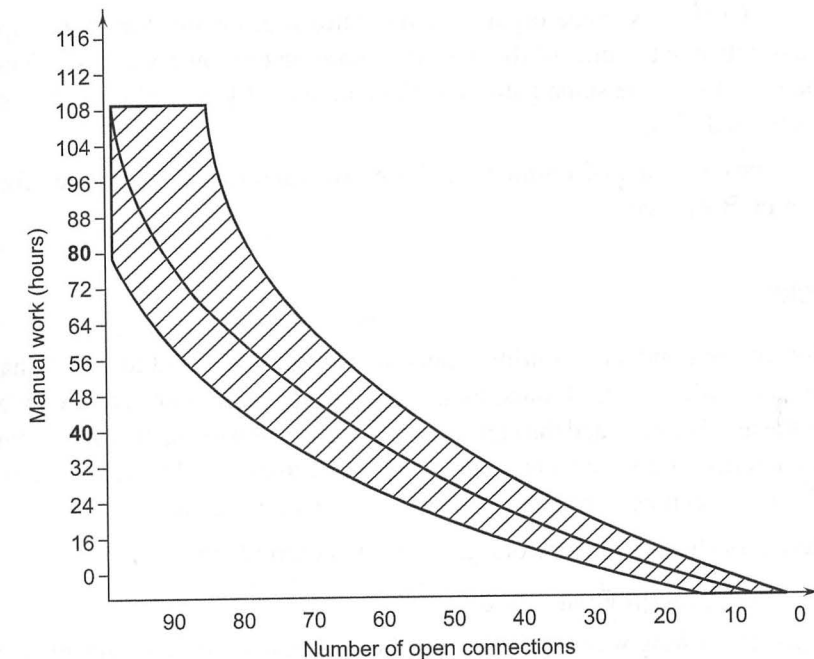


Fig. 5.31 Completion curve: manual work vs. number of open connections after auto-routing (redrawn after Herrmann, 1992)

Grid arrangement: In this method, the conductors can be placed on any gridline that does not cause interference with a land, component lead, via or keep-out area. Grided routers work best with designs consisting mainly of small-scale (SSI) and medium scale (MSI) integrated circuits. The higher the component density becomes, the slower the router runs because it has to manipulate a larger number of co-ordinate vectors.

Gridless arrangement: Gridless routers also rely on design rules, but are not constrained by fixed grid or cell dimensions. Instead, they assess conductor width, conductor spacing and via size for the net currently being routed, in order to vary the grid size on the fly. This technique allows the stacking of conductors and spaces of different widths across a span surface while dodging obstacles such as pins and vias. The advantage of gridless routers is that they adapt easily to changes in topological and packaging technology such as the requirements of surface mount devices. However, the method is inherently slow.

Plastic grid arrangement: Plastic grid or flexible field surface organization permits each routing grid to be of a different size. Run before the actual routing, the plastic grid algorithm establishes a grid pattern of various sized rectangles, customized to the components and other restrictions on the printed circuit board. As such, dense component placements and extraordinary land spacing, such as off-grid, are easily accommodated, with connections to the centre of all lands. This process is also inherently slower than a grided solution.

Channel routers: Channel surface organization creates routing surfaces made up of a series of channels that pass between the pins of the devices placed on the routing surface. Vias are placed in between the channels using the same pattern as the component leads. This allows for pre-reserved space for both wire and vias.

There are also possibilities of combining the above-mentioned techniques for proper space utilization on the PCB surface.

5.8.5 Checking

After completion of the conductor routing, checking routines are used to check that the design is fully in accordance with specified standards. Such check parameters can be rules on spacing, minimization of the number of plated through-holes, special design rules, the current-carrying capacity of the conductor, opened or disconnected pins, positional errors, etc. The errors are indicated on the display by flashing and can be corrected by the designer's interaction.

After completion of the design, artwork generation is carried out.

5.8.5.1 Verification/Design Rule Check

In manual designs, everything was required to be checked as a possible source of error. Component sizes, hole sizes, conductor widths and clearance, land-to-hole ratio, board areas to be free of components, clearance to the edges, positional accuracy and, of course, electrical interconnections had to be personally reviewed with a great deal of care. Automated design also requires no less attention as many types of checking are performed by the software during the design cycle.

After completing the design of the printed circuit board with the help of CAD Tool, a designer again has to verify the PCB in order to find out errors. Such type of verification/design rule check entails:

- General verification;
- Physical verification; and
- Electrical verification.

General Verification: General verification checks are broad checks carried out to establish that the basic requirements have been met by the design. The checks seek answers to the following questions:

- Has the circuit been analysed and divided into basic areas for a smooth signal flow?
- Has a basic grid pattern been fully utilized?
- Are the board size and the number of layers selected for low cost manufacturing?
- Are preferred conductor widths and spacing used wherever possible?
- Are preferred pad and hole sizes used?
- If jumpers are used, are they kept to a minimum?
- Are the larger areas of copper broken up to prevent blistering?

- Have mounting holes been provided or not?
- Are dummy areas inserted where necessary?
- Has shielding been effectively used where necessary?

Physical Verification: In physical verification, checks are carried out on the component placement and various other violations are identified. This verification attempts to answer the following questions:

- Are terminals and control locations compatible with the total assembly; i.e., does the concerned part work in conjunction with other parts?
- Will the mounted board meet shock and vibration requirements?
- Are specified standards (like: IPC 2221, IPC 2222) used?
- Are unstable or heavy parts adequately retained?

CAD systems provide for verifications of placement violations, including component-to-component spacing violations and other placement errors like insertion outline or grid restrictions.

Electrical Verification

The assignment of signal paths generally follows a priority system which starts with the highest priority signal such as of low level and descends to those signals which can tolerate greater movement on the layout. The ground signal is usually placed at the outer extremity of the printed circuit format because it allows for easy connection of internal ground buses to the main bus. The ground and power signals are the first to be assigned to ensure a good, clean path for the ground signals. The ground signal has to be of sufficient width to more than adequately carry the intended current for the board. Mostly, ground and power signals are put between the rows of pins of the ICs on the component side of the board. This will allow for all cross traces to be on the wiring side of the board where they are clearly visible after assembling. This allows the service engineer to easily follow a signal path during troubleshooting.

It is important to check the following from the point of view of electrical considerations:

- Have conductor resistance, capacitance and inductance effects been analysed? This is especially important for the critical voltage drops and grounds.
- Are the conductor and hardware spacing and shape compatible with insulation requirements considered?
- Have controlled impedance for frequencies above 100 MHz digital and 300 MHz analog signals been provided?
- Are polarities adequately identified?
- Have the dielectric changes related to surface coating been evaluated?

The tools available for checking electrical violations are designed to verify adherence to spacing criteria as listed in the route spacing spreadsheet. Any problem found by the route spacing violations is thus identified.

5.8.5.2 Design Rule Violations Check

The CAD tool scans schematic designs and checks for conformance to basic design and electrical rules. These checks usually are easiest for continuity verification and clearance violation checking. Clearance checking has assumed great importance because of tight tolerances on technologically advanced devices. The following checks are made:

- Conductor-to-conductor spacing;
- Conductor-to-land spacing;
- Land-to-land spacing;
- Via hole-to-via hole; and
- Mechanical clearance (to edge of board, around on-board connectors, around mounting holes).

The basic design rule checks involve checking of pad-to-pad, pad-to-trace, trace-to-trace and pad-to-via gaps, as shown in Figure 5.32.

5.8.5.3 General Guide Lines for Avoiding Design Errors

The following factors must be considered to avoid design errors:

- The design must have a complete, closed board outline. A design without a board outline or with only a partial outline is not valid and acceptable. There cannot be more than one board outline in the design.
- When placing components, enough clearance must be left between the connected pads and the board outline to avoid spacing violations during routing. Edge finger connections are a frequent offender in this category.
- Component pads outside the board outline to which connections are attached, will be rejected.
- Do not try to sub-divide plane layers with copper, outlines or details until after routing.

5.8.5.4 Identification Checks

For the sake of proper documentation and identification, the following checks are made on the artwork:

- The PCB number;
- Assembly title and number;
- Revision information and number; and
- Serial number.

Most of the CAD systems are capable of performing many of the design checks including connection and dimensional checks. The netlist driver CAD systems usually provide for checking circuit continuity versus the layout and vice versa.

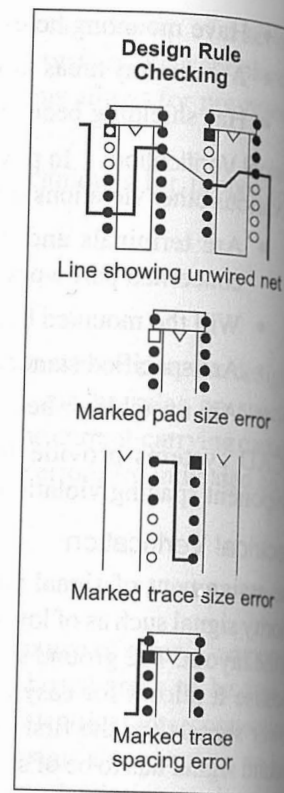


Fig. 5.32 Examples of design rule violations (IPC-D-390)

The most pressing problems faced by CAD software developers and users today are those of board size and density. With most systems, the packaging design engineer can find ways to work around these problems by fooling the system, changing the grid or splitting the board. Also, higher technology designs involving extremely dense analog circuitry typical of high speed RF designs, fine line technology, surface mount technology, flexible circuitry and thick film design, pose a considerable challenge to CAD software developers, and to the packaging designers that must work with existing CAD systems.

5.9 Design Automation

A design automation system is one in which the computer acts as the designer rather than just as an aid to the design engineer. Because of the large amount of data to be processed, high capability computers are utilized for this purpose.

While it is typical in designing with CAD systems that only about 80-90 per cent of the conductor routing is automatically accomplished, design automation results in nearly 100 per cent complete conductor routing. This is achieved by iterative routing programs which also replace previously routed conductor tracks and via holes. Iterative routing requires considerable processing time since a new step can only start after the previous run has been completed. Hence, only computers with high processing capacity are used in design automation.

Many of the digital IC packages house two to eight similar gates within the same package. The software does not only optimize the location of the package on board but can also move gates from one package to another in order to keep the overall length of interconnections and via holes at a minimum. An automatic check certifies that the fan-out capacity of the gates is taken care of.

Design automation software also provides conductor routing and component placement programs, which develop the board design simultaneously on all the conductive layers of the board. This eliminates the possibility of a dense pattern on the first layer and a far more sparse pattern on the final layer. This requirement is of special importance to the multi-layer design.

Design automation is an extremely efficient tool, especially for the design of highly complex double-sided and multi-layer boards with a large number of integrated circuits. If compared with CAD, the overall system flexibility in design automation is limited since it does not utilize human ingenuity which is still superior to computer algorithms. Design automation can be successfully applied where this high flexibility is not required and the tasks follow a certain standardization, as for instance in a digital system design. Standardization in this context is related to mechanical board outlines and the types of components/packages used. For standard designs, design automation can usually complete the design job without human involvement. But now and then, the need for an interaction by the designer is still required, particularly for more complex tasks.

Design modifications are also possible with design automation; as compared to CAD, they need more time and are costlier because the whole design gets involved. In CAD, the modifications are carried out manually on the display which works more in a straightforward way.

The advantages of CAD in providing production documentation can be further enhanced in design automation. It may even include engineering report facilities; giving an account of spare gates and unutilized pins or thermal placement.

The most challenging problems faced by the developers of EDA (Electronic Design Automation) systems are diminishing board sizes and increasing packaging density. Also, the need for higher technology designs involving extremely dense analogue circuitry typical of high speed high frequency designs, fine line technology, surface mount technology and flexible circuitry are some of the pressing problems being faced by the system developers and PCB designers. The ingenuity of the designer lies in finding ways to work around and find solutions by changing the grid or splitting the board, etc.

A number of companies are now commercially selling PCB-EDA software systems. They have similar features, with small variations here and there.

5.9.1 How to Judge CAD Systems?

A number of CAD systems are available and it is quite difficult to select the right tool you need. The early simple programs that ran on personal computers have now become quite sophisticated. Examples of such programs are SMARTWORK, PROTEL, P-CAD and CAD-STAR. Some of the latest CAD Systems are quite commendable. They include:

- Mentor Expedition 2000;
- Cadence Allegro V13.6;
- Innoveda Power PCB V3.6; or
- even much cheaper system like EDW 2000.

The standard elements are schematic capture, simulation, library, placement, auto-routing, post-processing and documentation. Evaluate each of these areas.

Demo Board: Avoid the demo board. It is too easy and doesn't represent the real world and will always look good. Use a board you have already designed as a comparison (take your toughest board).

Demo Licence: This is the best way to evaluate the learning curve, but will cause you to spend the maximum time to evaluate the tool and see how it integrates into your environment.

EMI, Thermal, Geometric and Other Technological Restrictions: Certain issues such as RF, EMI, thermal-, geometric and other technological restrictions are often better done on systems that specialize in these topics.

Auto-router: The percentage of completion and amount of time taken to complete the routing can be used as a criterion to judge which tool has the better router, provided that the quality is similar.

Metric-system: Is the software truly 100 per cent metric compatible? Designers and component manufacturers need to drop the English measurement system and embrace the metric system.

Library: Does the component library, provided by the software vendor, mostly contain old and outdated parts with no support documentation?

What kind of information is included in the library? A good library should meet all manufacturing requirements like:

- | | |
|------------------------------|------------------------------|
| • Component Families | • Component Geometry |
| • Documentation | • Tolerances |
| • Silkscreen outline | • Links or Search Engines |
| • Naming convention | • Creation of components |
| • Polarity Marking | • Mounting and Tooling Holes |
| • Transition to Metric Units | • Land patterns |

Standards Interface: There is a time when we need a standard interface such as "GENCAM". Check if the CAD software is GENCAM import and export compatible.

5.10 Manual Versus Automation in PCB Design

The decision to go in for automation in PCB design and artwork generation, and to what extent, depends upon a number of factors. Each method has its area wherein it can be justified as a method of preference.

Manual Design and Artwork Generation

This is a preferred method for simpler boards like single-sided or double-sided PCBs. It can also be successfully employed for higher circuit complexity in single or low-volume production quantities. Manual designing offers high flexibility and gives all possibilities for human ingenuity. However, digital boards of high complexity, especially those involving more than 100 ICs are difficult to design manually. Other limitations of manual methods relate to quality, time and the need for trained personnel.

Worldwide, a large percentage of the PCB design and artwork generation is still done manually. The fully manual method does not require any investment and may therefore, be used for some more time though its share gets continuously reduced especially for digital PCB designs.

Design Automation

The full automation of PCB design and artwork generation is a valuable tool wherein the input constraints can be standardized and reduced to a smaller number of straightforward approach rules. It is a desirable tool for the design of the highly sophisticated digital boards with more than 150 ICs, and for complicated multi-layer boards wherein it is extremely difficult, even for an experienced design engineer, to keep an exact overview on the design. Here the total design time can be reduced from several weeks to a few days while giving near optimum results. A tight delivery schedule is

important for a considerable number of PCB designs while simultaneously the need for de-bugging and rectifications should be minimum, CAD is often a preferred choice. Automated artwork draughting also provides a more precise artwork than manual draughting or taping.

Design automation is normally not used for analog PCBs because it is difficult to reduce the various design constraints for a wider range of analog circuit boards to a list of a view straightforward approach rules as it is normally done with digital circuit boards.

The large capital investment into CAD facilities will always call for a full utilization of the system. CAD can hardly be justified for boards with less than 20 digital ICs, with discrete component contents of more than 50 per cent or where only a very small number of PCBs is normally required of such designs.



5.11 Photoplotter

The processed digital data from CAD systems are converted back to graphics by photographic means with the help of an equipment called a 'photoplotter'. In this equipment, the drawing head which is a light spot projector changes its relative position towards the drawing medium which is the photographic film. Depending upon the design of the photoplotter, the drawing head or drawing table or both of them are moving parts.

A photoplotter, as the name implies, is a plotter that writes by using light and accordingly has to be programmed for the tool to be used and the path to be followed for drawing a trace. For a photoplotter, tool means especially shaped apertures through which light passes to create a given shape on film. An aperture can be used without movement to make a shape or with movement to make a line or an arc.

There are two major types of photoplotters, 'vector' and 'raster' (or laser). Each of these types handles the apertures differently. The vector photoplotter has become almost obsolete and is being replaced by the laser photoplotter. Laser plotters operate much quicker than vector machines. For example, a complex plot that required hours on a vector machine can usually be performed in ten minutes or less on a laser photoplotter. This decrease in turnaround time has also brought down the photoplotting cost considerably. The photoplotter includes three main features

- A photoplotting head, which comes with a set of apertures;
- Film plotter; and
- Software.

The main advantages of photoplotting as compared to pen plotting; are that the line quality is sharp, consistent and accurate. There are no pens to skip or dry up. The photoplotting head has more than 24 apertures thereby allowing the user to have several line widths and pad shapes. The details of the two types of photoplotters are available (Ward, 1992):

5.11.1 Vector Photoplotter

The computer numeric control (CNC) vector machines produce photoplots by moving a projection head across a sheet of photographic film. When stationary, the projection head could "flash" the light source to image a pad or target of any fixed shape. When moving, the projection head could "draw" circuit lines by leaving the light source on.

The projection heads that the vector plotters use are mechanically sophisticated and so the size and shape of the projected light is controlled by using apertures. These apertures could either be fixed or variable.

Aperture Wheels: Traditionally, the photoplotters counterpart to a pen plotter's pen rack has been the aperture wheel. The aperture wheel is a disk with 24 or 70 apertures arrayed radially along its circumference. When the photoplotter selects an aperture, the aperture wheel is rotated to place the desired aperture between the light source and the film. Apertures are themselves pieces of film and can be made to any shape required, though in practice, this is a time-consuming process and there is a physical limitation in size.

Flash and Draw Apertures: In order to achieve constant exposure on a vector photoplotter, apertures used for flashing pads are filtered differently than those used for drawing traces. Therefore, Flash and Draw apertures cannot be used interchangeably without the risk of localized over-exposure and under-exposure.

The set-up of an aperture wheel is an exacting and time-consuming process since each aperture in the wheel must be hand-mounted and aligned. In order to avoid repeated set-up costs, designers have the photoplotting vendor keep a wheel in file and are forced to always use that same set of apertures. This has obvious drawbacks, both in terms of design flexibility and the ease of migration to other vendors.

5.11.2 Raster (Laser) Plotters

In the laser raster plotting technique, the instructions which are to be executed are completely read into and interpreted by a raster image processor (RIP) before the machine plots. The RIP usually sorts the commands into some intermediate form which is then processed in real time to produce a stream of on/off commands to a laser modulator.

A raster plotter produces images as a series of tiny dots clustered together to form the lines and pads called from a vector plotter file. Generally the smaller the dots (pixels), the better the image. Also, it is a general rule that the smaller the pixels, the longer it takes the plotter to plot.

The SMDs require a wide variety of pad geometries and orientations. There are two methods of dealing with this requirement on CAD systems. When many different pad shapes are needed, one choice is to define a symbol for each shape. The easier and more economical way of producing many pad shapes is to define an aperture for each shape.