CHAPTER 13
PHYSICAL CHARACTERISTICS OF THE PCB

Lee W. Ritchey
3Com Corp., Santa Clara, California

13.1 CLASSES OF PCB DESIGNS

Printed circuit boards (PCBs) or printed wiring boards (PWBs) can be divided into two general classes which have common characteristics based on their end functions. These two classes have very different materials and design requirements and functions and, as a result, need to be treated differently throughout the design and fabrication processes. The first class contains analog, RF, and microwave PCBs such as are found in stereos, transmitters, receivers, power supplies, automotive controls, microwave ovens, and similar products. The second contains digital-based circuitry such as is found in computers, signal processors, video games, printers, and other products that contain complex digital circuitry. Table 13.1 lists many of the characteristics of each class of PCBs.

TABLE 13.1 Characteristics of RF/Analog vs. Digital-Based PCBs

RF, microwave, analog PCB	Digital-based PCB
Low circuit complexity	Very high circuit complexity
Precise matching of impedance often needed	Tolerant of impedance mismatches
Minimizing signal losses essential	Tolerant of lossy materials
Small circuit element sizes often essential	Small circuit element sizes desirable
Only 1 or 2 layers	Many signal and power layers
High feature accuracy needed	Moderate feature accuracy needed
Low/uniform dielectric constants needed	Dielectric constant secondary

13.1.1 Characteristics of Analog, RF, and Microwave PCBs

As can be seen from Table 13.1, the materials, design, and fabrication needs of this class of PCBs are markedly different from those of PCBs commonly referred to as digital.

- Circuit complexity is low because most components used have two, three, or four leads. This is due to the high usage of resistors, transistors, capacitors, transformers, and inductors.

- Traces, pads, and vias often act as inductors, capacitors, and coupling elements in the actual circuit. Their shapes may have a material effect on overall circuit performance. For example, the lead inductance and capacitance in a transistor collector circuit wire may act as the resonant components for an RF amplifier or it may degrade performance if it is unwanted. Figure 13.1 shows the impedance of traces as a function of their capacitance.

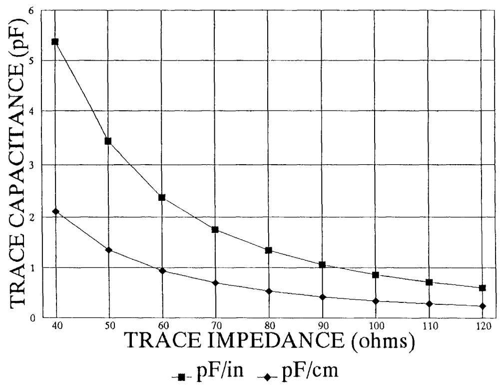

FIGURE 13.1 Trace capacitance vs. trace impedance, based on $L_0 = 8.5$ nH/in. (*Prepared by Ritch Tech, 1992.*)

- Two traces running side by side may be used to couple a signal from one circuit to another as is done in directional couplers of microwave amplifiers. (This same coupling in a digital circuit may result in a signal getting into a neighboring circuit causing a malfunction.)
- A series of conductors running side by side may function as a band-pass filter. Proper performance of filters, as well as most other wideband RF circuits, depends on all frequencies traveling with equal speed through the structures. To the extent that this is not true, frequencies that arrive later distort the signal being processed. This is called *phase distortion.*

Figure 13.2 illustrates the dielectric constant of various PCB materials as a function of frequency. Notice that some materials exhibit a dramatic decline in dielectric constant as frequency increases. The speed with which a signal travels through a dielectric is a function of the dielectric constant. Figure 13.3 illustrates signal velocity as a function of dielectric constant. From these two graphs it can be seen that using a dielectric material with a nonuniform dielectric constant in RF applications may result in severe phase distortion because the higher-frequency components arrive at the output before the lower frequencies.

- A trace in a power supply circuit may be expected to carry several amps without significant heating or voltage drop. Its resistance may even be used as a sense element to detect current flow. Similarly, handling large currents with insufficient copper in a trace may result in a voltage drop that degrades circuit performance. Figure 13.4 illustrates trace resistance of a copper trace as a function of its width and thickness. Figures 13.5 and 13.6 illustrate conductor heating as a function of width, thickness, and current flow.
- PCBs used in consumer electronics tend to share lower circuit complexity with RF and analog PCBs. However, the performance demands are far lower. The need for the lowest possible cost offsets this. To achieve the cost objectives, every effort is made to keep all connections on a single side and to form all holes in a single operation by punching. This

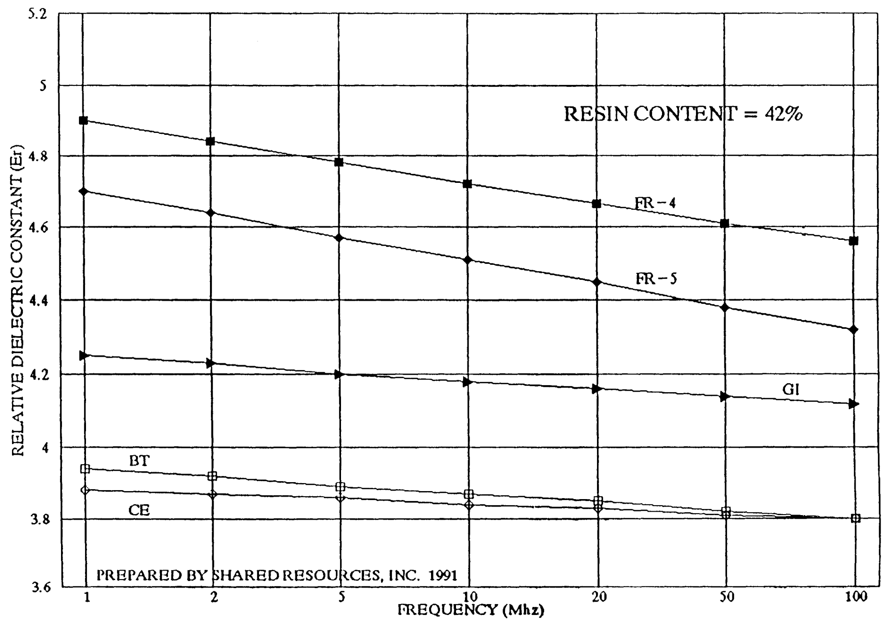

FIGURE 13.2 Dielectric constants vs. frequency of various PCB materials.

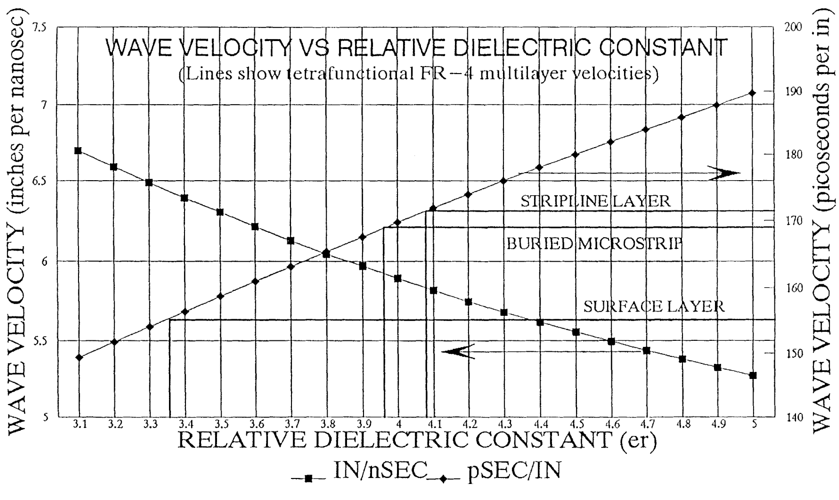

FIGURE 13.3 Signal velocity as a function of dielectric constant. (*Prepared by Shared Resources, Inc., 1991*)

eliminates both drilling and plating. The substrate material system is often resin impregnated paper, the lowest-cost substrate system for electronic packaging.

Summarizing, successful RF and analog design depends heavily on the properties of the materials used and on the physical shapes of the conductors and their proximity to each other

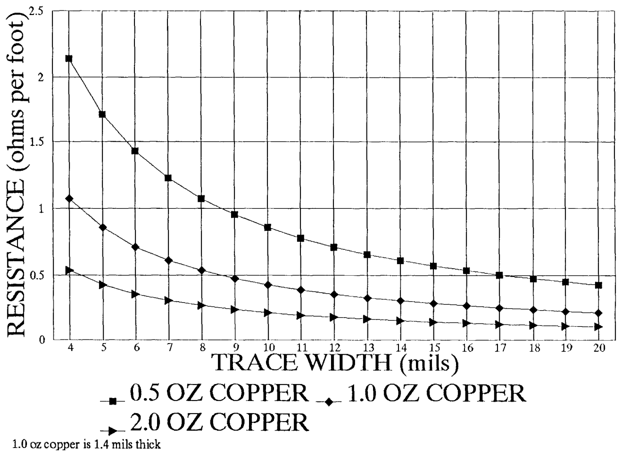

FIGURE 13.4 Trace resistance vs. trace width and thickness. (*Prepared by Ritch Tech.*)

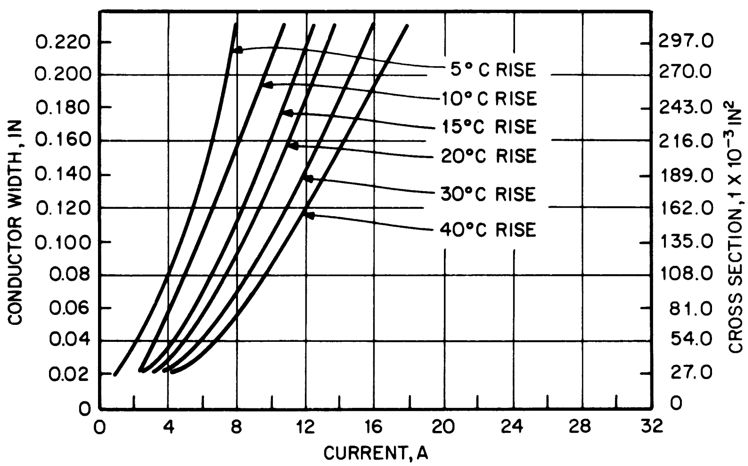

FIGURE 13.5 Temperature rise vs. current for 1-oz copper.

rather than on the ability to handle very large numbers of circuits simultaneously. Hand routing or connecting of the individual parts coupled with manipulating the shapes of individual copper features are essential parts of this design process. For these reasons, the design tools and design team must be chosen to meet these needs. Physical layout tools that provide convenient graphical manipulation of PCB shapes are a must.

13.1.2 Characteristics of Digital-Based PCBs

Compared to RF and analog PCBs, digital-based PCBs have complex interconnections, but are tolerant of rather wide feature size and materials variations.

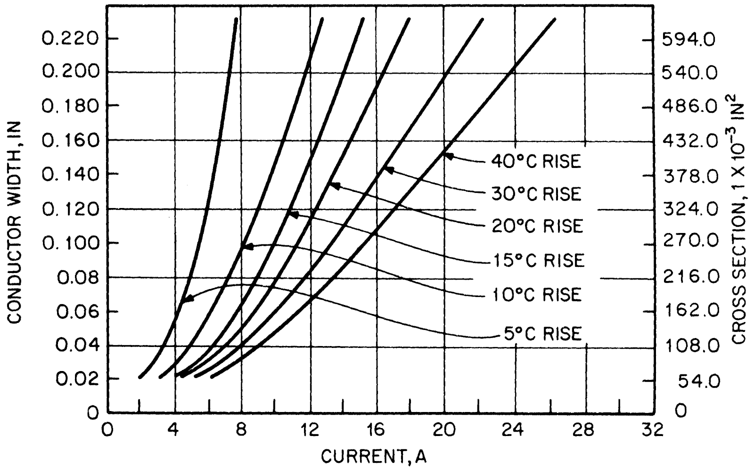

FIGURE 13.6 Temperature rise vs. current for 2-oz copper.

- They are characterized by very large numbers of components, often numbering in the hundreds and sometimes the thousands.
- Digital components often have large numbers of leads, as high as 400 or more. This high lead count stems from logic architectures that have data and address buses as wide as 128 bits or more. To connect PCBs with these wide data buses, digital systems often contain board-to-board connectors with as many as 1000 pins.
- Digital circuits have increasingly fast edges and low propagation delays to achieve faster performance. Edge rates as fast as 1 ns are now encountered in devices destined for products as common as video games. Table 13.2 lists edge speeds of some commonly used logic families, edge rate being the time required for a logic signal to switch from one logic level to the other (switching speed). Propagation delays, the time required for a signal to travel through a device, are decreasing along with edge rates.

TABLE 13.2 Typical Logic Family Switching Speeds

Logic family	Edge speed, ns	Critical length, in
STD TTL	5.0	14.5
ASTTL	1.9	5.45
FTTL	1.2	3.45
HCTTL	1.5	4.5
10KECL	2.5	7.2
BICMOS	0.7	2.0
10KHECL	0.7	2.0
GaAs	0.3	0.86

- These fast edges and short propagation delays lead to transmission line effects such as coupling, ground bounce, and reflections that can result in improper operation of the resulting PCB. Table 13.2 illustrates the degree to which a fast switching signal will couple into a neighboring line as a function of the edge-to-edge separation and the height of the signal pair above the underlying power plane. The critical length listed in Table 13.2 is the length of parallelism between two traces at which the coupling levels in Fig. 13.7 are reached.

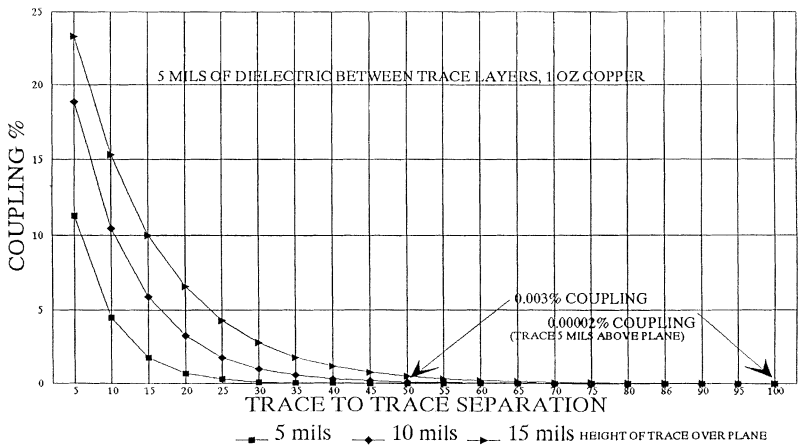

FIGURE 13.7 Trace-to-trace coupling. (*Prepared by Shared Resources, Inc.*)

The digital circuits themselves are designed to function properly with input signals that vary over a relatively wide range of values. Figure 13.8 illustrates the signal levels for a typical logic family, in this case ECL. The smallest output signal from an ECL driver is the difference between VOL_{max} and VOH_{min} or 0.99 V. The smallest input voltage to a device at which the logic part is designed to work properly is the difference between VIL_{max} and VIH_{min} or 0.37 V. The difference between these two levels, the noise margin of 0.62 V is available to counteract losses in the wiring and the dielectric and from other sources such as coupling and reflections. From this it can be seen that digital logic has a high tolerance of losses and higher immunity to noise.

This tolerance of noise and losses makes it possible to have trace features and base materials that introduce substantial losses and distortion while still achieving proper operation. It is this relatively high tolerance of distortion that makes it possible to manufacture economical digital PCBs.

Summary. The large number of connections in digital PCBs generally requires multiple wiring layers to successfully distribute power and interconnect all the devices. As a result, the design task is heavily weighted on the side of successfully making many connections in a limited number of routing layers while obeying transmission line rules. The base materials need to have characteristics that result in a PCB that is economical to fabricate and able to withstand the soldering processes while preserving high-speed performance. Compared to RF PCBs, losses in the dielectric tend to be of little concern for digital PCBs. The actual shapes of conductors, pads, holes, and other features have little effect on performance. (For detailed treatment of these topics, see Howard W. Johnson and Martin Graham, *High Speed Digital Design: A Handbook of Black Magic,* Prentice Hall, New York, 1993.)

The PCB design system and the design skill set for digital PCBs must be optimized to ensure accuracy in making large numbers of connections while successfully handling the high speed requirements of the system. Achieving this in a reasonable amount of time demands the use of a CAD system that contains an automatic router for use in connecting the wires.

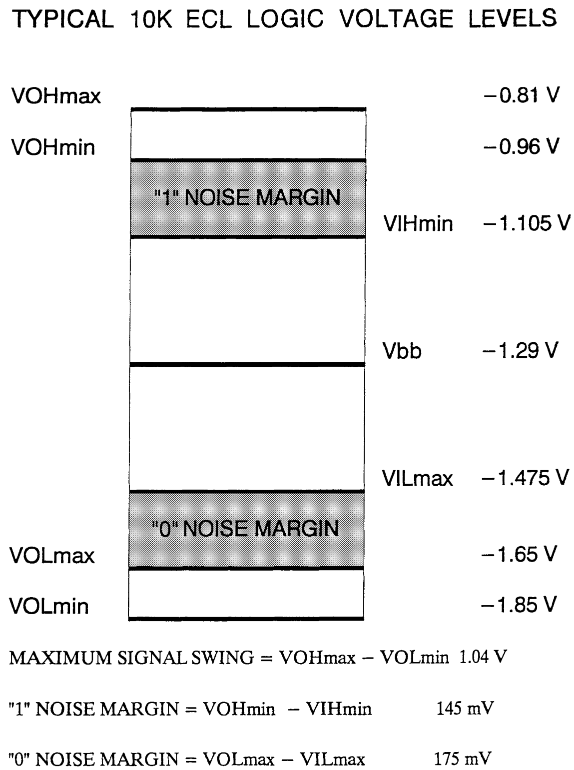

FIGURE 13.8 Noise-band chart for ECL. (*Prepared by Ritch Tech, 1992.*)

13.2 TYPES OF PCBs OR PACKAGES FOR ELECTRONIC CIRCUITS

The range of choices for packaging electronic circuits is quite broad. Some of the parameters that influence the choices are weight, size, cost, speed, ease of manufacture, repairability, and function of the circuit. The more common types are listed as follows with a brief description of their characteristics.

Levels of packaging are often used when referring to how electronics circuits are packaged. The first level of packaging is the housing of an individual component. This is usually an encapsulating coating, a molded case, or a cavity-type package such as a PGA (pin grid array). The second level of packaging is the PCB or substrate on which individual components are mounted. Third-level packaging is any additional packaging beyond these two. Most often third-level packaging takes the form of a multichip module (MCM) that has bare components mounted in it which is itself mounted on a PCB along with other components.

13.2.1 Single- and Double-Sided PCBs

These PCBs have conductor patterns on one or both sides of a base laminate with or without plated through-holes to interconnect the two sides. These are the workhorses of consumer electronics, automotive electronics, and the RF/microwave industry. They are the lowest-cost choice for consumer products. Laminate materials range from resin-impregnated paper for consumer electronics to blends of low-loss Teflon™ for RF applications.

13.2.2 Multilayer PCBs

These PCBs (see Fig. 13.9) have one or more conductor layers (usually power planes) buried inside in addition to having a conductor layer on each outside surface. The inner layers are connected to each other and to the outer layers by plated through-holes or vias. These are the packages of choice for nearly all digital applications ranging from personal computers to supercomputers. Numbers of layers range from 3 to as many as 50 in special applications. Laminate materials are nearly always some type of woven glass cloth impregnated with one of several resin systems. The resin system is chosen to satisfy requirements such as the ability to withstand high temperatures, cost, dielectric constant, or resistance to chemicals.

13.2.3 Discrete-Wire or Multiwire PCBs

This class of PCBs is a variation of the multilayer package. A circuit board is constructed by etching a pair of power layers back to back on a laminate substrate. A layer of partially cured, still sticky laminate is bonded to each side of this power plane structure. Discrete wiring is rolled into this sticky adhesive in patterns that will connect leads or serve as access to surface-mount component pads.

Once all of the wires have been rolled into place, a second layer of laminate is placed over the wires, followed by copper foil sheets. This sandwich is then laminated, drilled, and processed like any multilayer PCB. The resulting PCB has outer layers and power planes much like any multilayer PCB. The principle difference is the printed wiring signal layers have been replaced by discrete wiring layers. In some cases of very high wiring density, alternating layers of power planes between wiring layers serve as isolation.

Designing a discrete-wiring PCB (see Fig. 13.10) involves adding a special discrete-wiring router to a standard PCB design system to generate the files for the machine that rolls the wire into the dielectric. Discrete wiring once provided a faster prototyping alternative to multilayer fabrication. At the present time, both technologies are equally rapid and cost effective during prototyping. However, for modest to high-volume production, multilayer technology is more cost effective than discrete wiring.

13.2.4 Hybrids

These circuits are usually single- or double-sided ceramic substrates with a collection of surface-mount active components and screened-on resistors made from metallic pastes. They are most often found in hearing aids and other miniature devices.

13.2.5 Flexible Circuits

These circuits are made by laminating copper foil onto a flexible substrate such as Kevlar® or Kapton®. They can range from a single conductor layer up to several layers. They are most often used to replace a wiring harness with a flat circuit to save weight or space. Often, the flexible circuit will contain active and passive components. Common applications are cameras, printers, disk drives, avionics, and video tape recorders.

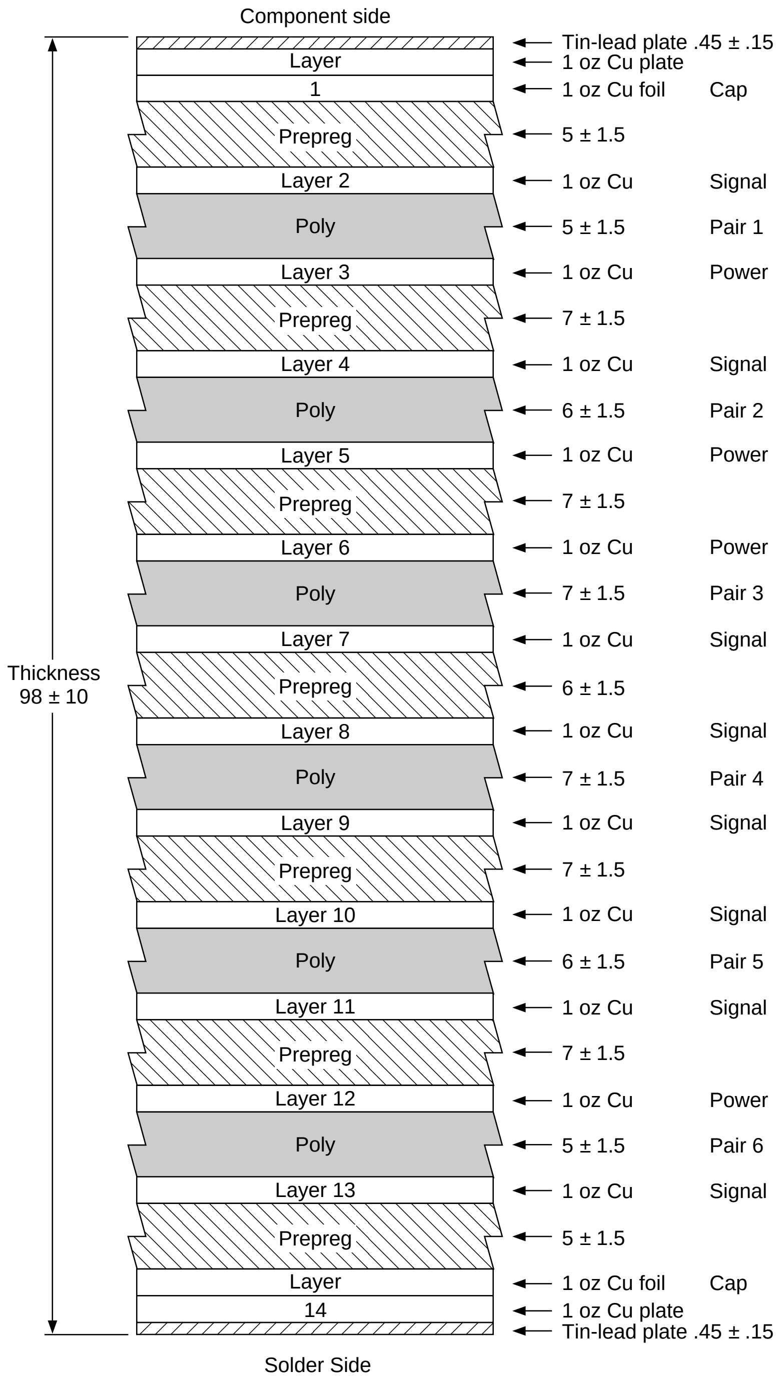

FIGURE 13.9 Cross section of 14-layer multilayer printed wiring board, showing a typical inner layer and prepreg material relationship. In this case, to reduce *z*-axis expansion, the innerlayers are polyimide, while the prepreg material is semicured polyimide. Typical signal, power, and ground layers are also indicated, as well as the thickness of the copper foil for each layer.

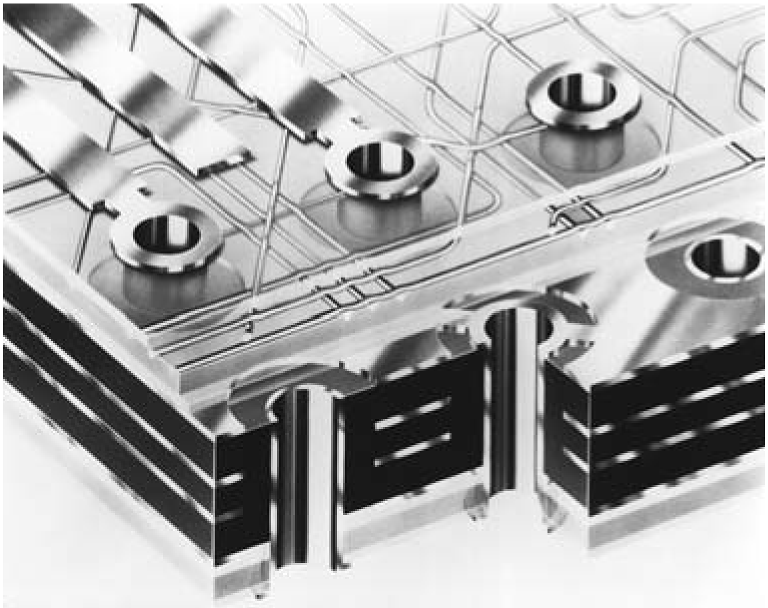

FIGURE 13.10 Cross section of a discrete-wire PCB. (*Courtesy of Icon Industries.*)

13.2.6 Flexible Rigid or Flex-Rigid

As the name suggests, these are combinations of flexible PCBs and rigid PCBs in a single unit. The flexible portion of the circuit is made first and included in the lamination process of the rigid portion of the assembly. This process eliminates wiring harnesses and the associated connectors. Applications include avionics and portable equipment such as laptop computers. As a rule, a flex-rigid assembly is more expensive than an equivalent combination of PCBs and cables.

13.2.7 Backplanes

Backplanes are special cases of multilayer PCBs. They tend to contain large quantities of connectors that have been installed using press fit pins. In addition, backplanes are used to distribute large amounts of dc power to the system. This is accomplished by laminating several power planes inside the backplane and by bolting bus bars onto the outside surfaces. Some applications require that active components, such as surface-mount ICs, be soldered to their surfaces. This greatly increases the difficulty of assembly as a result of the need to solder fine-pitch parts to a large, thick PCB.

13.2.8 MCMs (Multichip Modules)

Multichip modules are essentially miniature PCBs. Miniaturization is achieved by removing components such as ICs from their packages and mounting them directly to the substrate using wire bonds, flip chip, TAB or flip TAB. The motivation for using an MCM is miniaturization, reduction in weight, or a need to get high-speed components as close to each other as possible to achieve high-speed performance goals. MCMs usually represent a third level of packaging in a system between packaged components and the carrier PCB. As a result, this additional level of packaging virtually always results in a more expensive, more complex assembly than the equivalent circuits in standard packages. There are several types of MCM package.

13.2.8.1 MCM-L, Multichip Module, Laminate. This version of an MCM is manufactured from very thin laminates and metal layers using the same techniques employed in the manufacture of standard PCBs. Features such as holes, lands, and traces are much finer and require tooling similar to that used to manufacture semiconductors. This is the least expensive MCM type to design, tool, and manufacture. The same design tools and methodologies used for PCBs can be used.

13.2.8.2 MCM-C, Multichip Module Ceramic. This version of an MCM is manufactured by depositing conductor layers on thin layers of uncured ceramic material, punching and backfilling holes for vias, stacking the layers, and firing the total to create a hard ceramic multilayer substrate. This is the second least expensive MCM type to design, tool, and manufacture. It has been the workhorse of IBM's large mainframe computers for at least two decades. The same design tools and methodologies used for PCBs can be used.

13.2.8.3 MCM-D, Multichip Module Deposited. This version of an MCM is manufactured by depositing alternating thin films of organic insulators and thin films of metal conductors on a substrate of silicon, ceramic, or metal. The design and manufacturing techniques used for this technology resemble that used to create integrated circuit metallization. The thermal conductivity of the substrate is quite good. Design and fabrication support for MCM-D is limited.

13.2.8.4 MCM-D/C, Multichip Module Deposited and Cofired. This version of an MCM is a combination of a cofired, multilayer ceramic substrate containing the common wiring for a family of modules and deposited conductor and insulation layers containing the personality wiring. It has all of the problems of each technology it uses plus problems related to mismatches in temperature coefficient of the two materials systems.

13.2.8.5 MCM-Si, Multichip Module Silicon. As the name implies, this MCM technology starts with a silicon substrate like that used to make integrated circuits. Conductor patterns are formed using silicon dioxide (glass) as an insulator and aluminum or a similar metal for the wiring patterns in the same manner as is employed to build an integrated circuit. In fact, the same design tools and fabrication tools used to build ICs are used to build MCM-Si modules.

A significant advantage of MCM Si is the fact that the substrate is the same material as the ICs that will be attached to it. Therefore, it is thermally matched to the ICs, ensuring reliable contacts over extremes of temperature.

13.2.8.6 Summary of MCM Technologies. MCM packaging may be seen as a way to achieve higher performance from a collection of high-speed ICs than can be accomplished by mounting them onto a PCB or as a way to reduce size and weight. In reality, higher levels of integration nearly always result in a more economical solution. For all but low-volume, specialty applications, such as aerospace electronics and specialty processors for very high performance equipment, this has proven to be true for quite some time. This is likely to continue to be so for some time as semiconductor technology continues to improve the density of functionality that can be placed on a single IC. One need only examine the progression of microprocessor performance to see this phenomenon at work.

When a high-performance product requires integrated circuits made with different processing technologies, such as analog and CMOS or ECL and CMOS, integration does not represent a reasonable alternative to MCMs. Examples of this type of product are high-performance graphics products and video signal processing equipment.

13.3 METHODS OF ATTACHING COMPONENTS

A wide range of methods has evolved for attaching components to PCBs. The methods chosen as well as the combinations of methods chosen for a product have a substantial impact on the final cost, ease of assembly, availability of components, ease of test, and ease of rework. The five basic attachment combinations are: through-hole only, through-hole mixed with surface-mount on one side, surface-mount one side only, surface-mount both sides, and surface-mount both sides with through-hole.

13.3.1 Through-Hole Only

All component leads attach to the PCB by being inserted into holes that pass through the PCB. The components may be secured by wave soldering or by pressing into holes that result in an interference fit (press fit). Assembly involves a component placement operation followed by a wave-soldering operation. This method is still the workhorse of the low-cost consumer electronics industry.

13.3.2 Through-Hole Mixed with Surface-Mount

Components such as connectors and PGAs are attached to the PCB with through-hole technology. All other components are mounted using surface-mount packages. This is the most common method used to assemble electronics products. Assembly is a two-step process that involves placing all surface-mount parts and soldering them in place with a solder reflow system, then inserting all through-hole parts and soldering them in place in a wave-soldering operation. Alternatively, the through-holes may be hand-soldered if the quantity is small.

13.3.3 Surface-Mount, One Side Only

This type of package is made up of only surface-mount parts all mounted on the same side of the PCB. Assembly is a one-step process that involves placing all components and soldering them in place using some form of solder reflow.

13.3.4 Surface-Mount, Both Sides

This type of package contains surface-mount components on both sides. Assembly is a two-step process that involves placing all components on one side and reflow soldering them, followed by placing all components on the other side and reflow soldering them. In addition to more complex assembly operations, designing and testing this type of assembly is much more complex, as parts often share the same area on opposite sides of the PCB, causing conflicts when trying to locate vias and test points.

13.3.5 Surface-Mount, Both Sides with Through-Hole

As the name implies, this package type contains surface-mount parts on both sides, as well as through-hole parts such as connectors. In most cases, the surface-mount components on one side are passives, such as bypass capacitors and resistors that can withstand being passed through wave soldering. Assembly is a three-step process that involves placing the surface-mount components on the primary side and reflow soldering them. Once this is complete, the secondary-side surface-mount components are glued in place, the through-hole components are inserted, and the PCB is sent through wave soldering.

Because of the extra operations and exposure of components on the secondary side to molten solder, this type of assembly is prone to many assembly defects. It is important to note that soldering fine-pitch parts on the secondary side using wave soldering results in excessive solder bridging and should be avoided. In addition, active components, such as ICs, may be damaged by exposure to excessive heating.

13.4 COMPONENT PACKAGE TYPES

Over time, a wide assortment of packages has been developed to house components. Selecting the correct package type for each component is one of the most important parts of the design process. Package types chosen affect ease of design, assembly, test, and rework, as well as product cost and component availability.

13.4.1 Through-Hole

This class of component is characterized by parts that have wire or formed leads. These leads pass through holes drilled or punched in the PCB and are soldered to lands on the back side or to plating in the holes. This is the original package type used for electronic components. A major benefit of through-hole components is the fact that every component lead passes all the way through the PCB. Because of this, there is automatic access to any PCB layer to make connections. Further, every lead is available on the bottom of the PCB, so test tooling is easy to construct. With the advent of surface-mount components, through-hole is used primarily for connectors and pluggable devices such as microprocessors mounted in PGA packages.

Through-hole packages are often preferred for ICs and other components that dissipate large amounts of heat, because of the relative ease with which heat-sinking devices can be fitted to them. In addition, it is much easier to provide a socket for a through-hole device. This eases the task of changing programmable parts and microprocessors when it is necessary to upgrade a system.

Caution: Integrated circuits in through-hole packages are becoming difficult to find as they are displaced by surface-mount equivalents and should be avoided in new designs unless a secure supply of components is available for the production life of the design.

13.4.2 Surface Mount

This package type is the mainstream choice for packaging electronic components of every type, including connectors. Its principle characteristic is that all connections between a component lead and the PCB or substrate is made with a lap joint to a pad on the surface of the PCB. This has both advantages and disadvantages. On the advantage side, since there are no holes piercing the PCB, wiring space on inner layers and on the reverse side is not consumed with component lead holes. Because of this, it is usually possible to wire a PCB in fewer layers than would be true with through-hole parts. Another and larger benefit is the fact that surface-mount components are always smaller than their through-hole equivalent, making it possible to fit more parts in a given area.

The main disadvantages of surface-mount components stem from the fact that there are no leads to easily grip with instrumentation probes and that there may not be access to the leads from the reverse side for purposes of production testing. This gives rise to the need to add a test pad to most nets on the back side in order to perform production test. It also gives rise to the need for very expensive, complex adapters in order to provide access to leads on processors and other complex devices to probe their inputs and outputs when performing diagnostic work.

Yet another disadvantage of surface-mount parts stems from their small size. It is more difficult to remove heat from SMT packages than it is for their through-hole equivalent. In some cases, such as high-performance processors, the heat generated by the IC is too high to permit proper operation in an SMT package.

13.4.3 Fine Pitch

Fine pitch is a special class of surface-mount components. This class is characterized by lead pitches lower than 0.65 mm (25 mils). These fine lead pitches are usually driven by very high lead count ASICs (160 pins and up) or by the extreme miniaturization requirements of PCMCIA (Personal Computer Memory Card Industry Association) cards, PDAs (personal digital assistants), and other small, high-performance products. The motivation for designating a special fine-pitch-component class of surface-mount parts is the extra difficulty of successfully testing, assembling, and reworking these parts on PCBs, as well as in building PCBs with accurately formed patterns and solder masks to mate with the leads of fine-pitch parts. Fine-pitch parts are the source of most manufacturing defects in a well-run SMT assembly line. The defects stem from lack of coplanarity of the leads, bent leads, insufficient solder on the joints, and poor alignment of the leads to the patterns on the PCB.

Successful manufacture using fine-pitch components involves very tight cooperation among the PCB designer, the PCB fabricator, the component manufacturer, and the PCB assembler/tester. It almost always involves specialized assembly, test, and rework tooling. Design is most often done by convening a series of meetings of the engineering personnel of all these groups to evolve a set of rules, processes, equipment, tooling, and components. These meetings need to start at the product development stage and continue to be held until the proper pad shapes and sizes have been established and the production process is stable.

13.4.4 Press Fit

Press fit is a special form of through-hole technology. Components are fastened to the PCB by deliberately designing an interference fit between the component lead and the plated through-hole in the PCB. The principle application of press-fit technology is the attachment of connectors into backplanes. The reason for this is that early backplanes were built by wire wrapping the signal connections onto the connector pins extending out the back of the backplane. Trying to solder the connector pins to the backplane through this field of pins proved difficult, if not impossible. The solution was press fit.

Successful assembly of a press-fit backplane rests in designing a hole size small enough to create a solid connection with the pins while ensuring that the hole is large enough to permit the insertion of the pin without fracturing the hole barrel.

Caution: Hot-air leveling the solder on a backplane results in a hole with irregular diameter. This irregularity will almost certainly result in damaged hole plating when the insertion operation is done. Be sure to note on the fabrication drawing of a press-fit backplane that hot-air solder leveling is prohibited. Solder must be plated onto the backplane traces and pads and fused using IR reflow or hot-oil reflow to fuse the lead and tin into a solder alloy.

13.4.5 TAB

TAB stands for tape-automated bonding, which is a technique for attaching bare IC die to a printed circuit board. It uses a subminiature lead frame that attaches directly to the bonding pads of an IC at one end, spreads out to a much larger pitch, and attaches to pads on a PCB. The tape in the title describes the method for carrying the parts prior to assembly, which is a tape with the TAB lead frame and IC built into it. The tape is wound onto a reel for handling. The principle application of TAB components is products such as pagers and portable phones that are made in very high volume and can justify the automation involved in attaching TAB parts to substrates.

13.4.6 Flip Chip

Flip-chip technology involves plating raised pillars of metal on the bonding pads of ICs, turning them upside down, and attaching them to a matching pattern on a substrate. The substrates are most often silicon and precision ceramics. From this description, it can be seen that this is a very specialized packaging method. To succeed, a source of tested good bare die with plated-on pillars must be available. This situation occurs almost exclusively in very high performance supercomputers where the ICs have been specially designed for the application or in very high volume applications such as pagers and cellular phones.

13.4.7 BGA

BGA or ball grid array is a relatively new technology that is a cross between pin grid arrays and surface mount. High-pin-count IC die are mounted on a multilayer substrate made from ceramic or organic material. The die are connected to the substrate using standard wire-bond techniques and encapsulated in epoxy or another form of cover. The bottom side of the substrate contains an array of high-melting-point solder balls which connect to the wire-bond pads through the multilayer substrate. These balls mate with a matching array of pads on the PCB and are reflow soldered during the same operation as all other surface-mount parts. (See Fig. 13.11.)

The appeal of BGA technology is as an alternative to high-pin-count, fine-pitch SMT ICs. As mentioned earlier, assembling high-pin-count, fine-pitch SMT parts is very difficult, owing to the fragile nature of the component leads. BGAs represent a much more robust package during component-level test and during assembly.

As with most technologies, there are some disadvantages to BGAs. Among these are:

- The solder joints are hidden from view, so inspecting them requires x-rays.
- The solder joints are not accessible for rework, so the soldering process must have a very high success rate.
- Part removal requires special tooling.
- The pattern on the PCB surface is an array of pads that require one via each all the way through the PCB, hampering routing.
- All component leads are concentrated in a much smaller area than the equivalent part in a fine-pitch SMT package. This concentrates the wiring in a small area pierced with many vias. As a result, the BGA PCB will likely have more wiring layers than the equivalent SMT PCB.
- The BGA package is more expensive than the equivalent fine-pitch SMT package, by as much as three times.

13.4.8 Wire-Bonded Bare Die

As the name implies, this method of assembly involves attaching bare IC die directly to a substrate using an adhesive or reflowing solder and connecting the bonding pads to pads on the

FIGURE 13.11 Typical ball grid array package. (*Courtesy of Icon Industries.*)

PCB using wire-bond techniques. Virtually all digital watches and many other similar consumer products use this assembly technique. It is very inexpensive when used in very high volume products with only a single IC to connect.

13.5 MATERIALS CHOICES

A wide variety of materials has been developed for use in packaging electronic circuits. These can be divided into three broad classes: reinforced organics, unreinforced organics, and inorganics. These are used primarily for rigid PCBs, flexible and microwave/RF PCBs, and multichip modules, respectively. The following treatment of the available materials concentrates on the principle materials systems used in PCB design along with the properties that warrant their use. See Chaps. 5 and 8 in this handbook for detailed data provided on loss tangents, temperature of coefficient of expansion, glass transition temperature, and other electrical properties.

IPC, the Institute for Interconnecting and Packaging Electronic Circuits, publishes a comprehensive series of standards that list in detail the properties of all types of laminates, resins, foils, reinforcement cloths, and processes that are candidates for the manufacture of PCBs. These standards start with IPC-L-108B and run up to IPC-CF-152. It is recommended that copies of the applicable standards be obtained at the start of a program in order to ensure a thorough understanding of all important characteristics of the materials being considered for a design.

Properties important to the manufacture of PCBs include (see Table 13.3):

- *Glass transition temperature* T_g—The temperature at which the coefficient of thermal expansion in a resin system makes a sharp change in rate from a slow rate of change to a rapid rate of change. A high T_g is important for PCBs that are very thick to guard against barrel cracking or pad fractures during the soldering operation.

- *Coefficient of thermal expansion* T_{CE}—Surface-mount assembly process subjects the printed wiring assembly to more numerous temperature shocks than typical through-hole processes. At the same time, the increase in lead density has caused the designer to use more and more layers, making the board more susceptible to problems concerned with the base material's coefficient of thermal expansion T_{CE}. This can be a particular problem with regard to the z-axis expansion of the material, as this induces stresses in the copper-plated hole, and becomes a reliability concern. Figure 13.12 shows typical z-axis expansion for a variety of printed circuit base laminate materials.

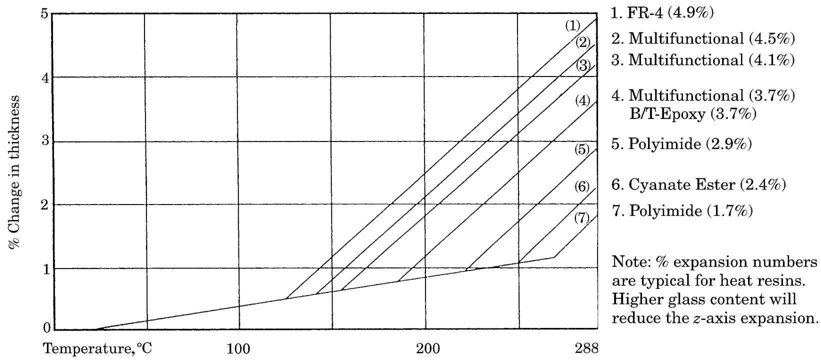

FIGURE 13.12 Typical z-axis expansion via thermal mechanical analysis. (*Courtesy of Nelco International Corp.*)

- *Relative dielectric constant* e_r—This characteristic measures the effect that a dielectric has on the capacitance between a trace and the surrounding structures. This capacitance affects impedance as well as the velocity at which signals travel along a signal line. (See Figs. 13.3 and 13.4) Higher e_r produces lower impedance, higher capacitance, and lower signal velocity.
- *Loss tangent, tan (f), or dissipation factor*—A measure of the tendency of an insulating material to absorb some of the ac energy from electromagnetic fields passing through it. Low values are important for RF applications, but relatively unimportant for logic applications.
- *Electrical strength or dielectric breakdown voltage DBV*—The voltage per unit thickness of an insulator at which an arc may develop *through* the insulator.
- *Water absorption factor WA*—The amount of water an insulating material may absorb when subjected to high relative humidity, expressed as a percent of total weight. Absorbed water increases relative dielectric constant as well as reduces DBV.

13.5.1 Reinforcement Materials

The principal reinforcement for PCB substrate materials is cloth woven from glass fibers. A variation of this glass is cloth made from quartz fibers. The resulting material has a slightly lower dielectric constant than ordinary glass, but at a substantial cost premium and a more difficult drilling cycle. Kevlar is an alternate woven reinforcement that results in a lower-weight material system with a lower dielectric constant, also at a higher cost and higher difficulty in processing.

TABLE 13.3 Properties of Some Common PCB Materials Systems*

	T_g	e_r	tan (f)	DBV, V/mil	WA, %
Std. FR4 epoxy	125C	4.1	0.02	1100	.14
Multifunctional epoxy	145C	4.1	0.022	1050	.13
Tetrafunctional epoxy	150C	4.1	0.022	1050	.13
BT/epoxy	185C	4.1	0.013	1350	.20
Cyanate ester	245C	3.8	0.005	800	.70
Polyimide	285C	4.1	0.015	1200	.43
Teflon	N.A.	2.2	0.0002	450	0.01

* All with E-glass reinforcement, except Teflon.

The original reinforcement material for PCBs was paper or cardboard in some form. Paper impregnated with a resin system is still in use in consumer applications where lowest possible cost is necessary and where performance is not an issue.

13.5.2 Polyimide Resin Systems

Polyimide resin-based laminates are the workhorse of electronics that must withstand high temperatures in operation or in assembly or repair. Common applications include down-the-hole well-drilling equipment, avionics, missiles, supercomputers, and PCBs with very high layer count. The principle advantage of polyimide is its ability to withstand high temperatures. It has approximately the same dielectric constant as epoxy resin systems. It is more difficult to work with in fabrication, is more costly than FR4 systems, and absorbs more moisture.

13.5.3 Epoxy-Based Resin Systems

Epoxy resin-based laminates are the workhorse of virtually all consumer and commercial electronic products. There are several variations of this pervasive laminate family, each developed to answer a special need. Among these are standard FR4, multifunctional epoxy, difunctional epoxy, tetrafunctional epoxy, and BT or bismaleimide triazine blends. Each of these was developed to answer the need for a resin with a successively higher T_g or glass transition temperature. Multifunctional epoxy is the most commonly used form.

13.5.4 Cyanate Ester-Based Resin Systems

This resin system is a recent entrant into the high-performance resin system category. It is said to offer processing characteristics superior to those of the FR4 blends while offering a higher T_g.

13.5.5 Ceramics

A wide variety of ceramic or alumina substrate materials have been developed for use in hybrids and multichip modules. These materials are the subject of specialized manufacturing processes beyond the scope of this handbook. The reader needing information on this group of materials is advised to contact a major manufacturer of ceramic materials.

13.5.6 Exotic Laminates

Kevlar, Kapton, Teflon, and RO 2800 are materials developed for specialty applications. The first two, in the form of thin films, are commonly used as substrates for flexible circuits. The latter two are the principal dielectrics for microwave and RF circuits. All of these materials can be used with or without reinforcements.

13.5.7 Embedded Components Materials

Specialized materials have been developed to allow construction of passive components such as resistors and capacitors into the PCB structure itself. Most of these materials are patented and available from a very small supplier base.

13.5.7.1 Embedded Resistors. These are formed by plating a very thin film of nickel or other metal onto a copper foil layer and laminating this foil, plated side in, to an FR4 or other substrate material. To form a resistor, a window is opened in the copper foil, exposing the underlying nickel resistor layer. A resistor of the appropriate value is formed in the resistance material layer. Contact is made with the resulting resistor by etching connecting pads in the copper foil layer and drilling holes through these pads and plating the holes. (See Fig. 13.13.)

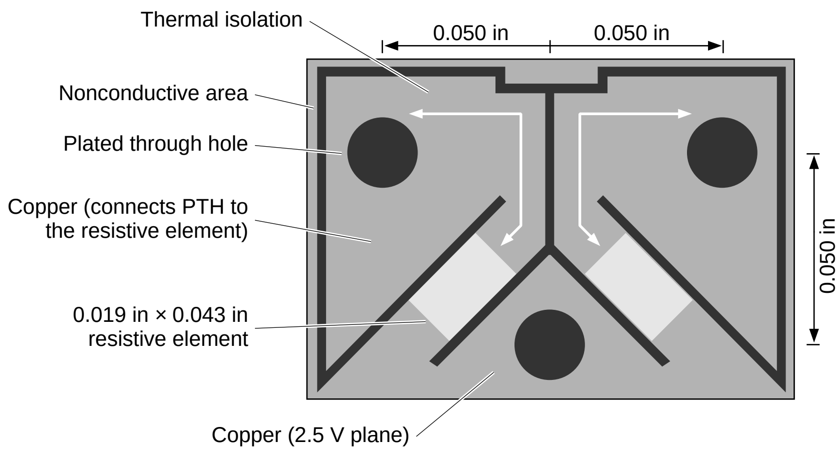

FIGURE 13.13 The two light diagonal shapes in the figure above are terminating resistors for ECL transmission lines. They are formed directly in the VTT powerplane by etching the copper away from the underlying nickel resistive layer. One end of each resistor is connected to the device terminal using a via and the other end is connected directly to the -2.5V plane. (*Courtesy of Ohmega Industries.*)

Resistive material is available in 25- and 100-Ω/square values. The principal application of embedded resistor technology is as terminating resistors for ECL transmission lines and as resistors on flexible circuits in products such as cameras and portable tape and CD players. Practical resistor values range from about 10 to 1000 Ω.

13.5.7.2 Embedded Capacitance. This is formed by placing two copper planes close to each other using very thin dielectrics (1.5 to 2.0 mils). The principal application is in the creation of very high quality, high-frequency capacitance between two power planes. This does indeed result in high-quality capacitance, but usually at a high cost resulting from the need to add a pair of extra planes to a PCB in order to create the capacitance (see Fig. 13.14).

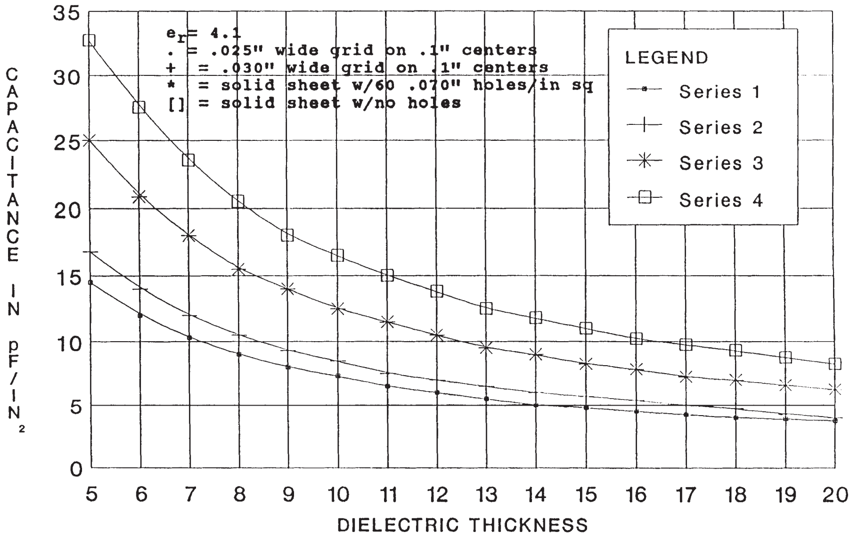

FIGURE 13.14 Capacitance per unit area vs. dielectric thickness.

13.6 FABRICATION METHODS

A wide variety of fabrication methods has been developed to meet the needs of the electronics industry. The following descriptions are quick summaries of each method intended to acquaint the reader with their advantages and disadvantages and likely applications. Detailed treatments are presented elsewhere in this book.

13.6.1 Punch Forming

Punch forming is used in the manufacture of very low cost, single-sided PCBs such as are used in many consumer electronic products. The process involves printing and etching the conductor patterns on one side of a laminate substrate, usually paper-reinforced epoxy. All holes are punched in a single stroke by a die containing a pin and opening for each hole. The PCB outline is formed in a second die that "blanks" it from the panel in which it is processed.

Often, several PCBs will be contained in a single panel sized to travel through the assembly process. After a PCB is punched out of the panel, it is forced back into the vacant hole and is held in place by interlocking fibers along the edges of the PCB. After assembly and testing is complete, each PCB is pressed from the panel. This is known as *crackerboarding* and is aimed at reducing overall manufacturing costs.

13.6.2 Roll Forming

Roll forming is a process used to manufacture flexible circuits in very large quantities. This is the lowest-cost method for the manufacture of flexible circuits. However, it involves substantial tooling, so it is applicable only for very high volume products. Examples of PCBs manufactured using this method are printer head connections, disk drive head connections, and the circuits used in cameras and camcorders. The PCBs can be single- or double-sided.

Roll forming resembles newspaper printing in that the process starts with a large roll of copper-clad laminate that is fed through a long process line containing stations which perform each operation on a continuous basis, starting with printing the conductor pattern, etching it, forming the holes, testing, and blanking from the roll itself. The process can include lamination of a cover insulator layer over the conductors as well.

13.6.3 Lamination

Lamination is the process by which PCBs of more than two layers are formed. The process begins by etching the conductor patterns of the inner layers onto thin pieces of laminate called *details*. These details are then separated by partially cured laminate called *prepreg* and stacked in a *book* with layers of prepreg on the top and bottom and foil sheets on the outside. This stack is placed into a press capable of heating the combination to a temperature that causes the prepreg resin to reach the liquid state. The liquefied resin flows into the voids in the copper patterns to create a *solid* panel upon cooldown. Once cooled, the panel is sent through the drilling and plating operations much like a two-sided PCB.

Note that some materials, such as polyimide, do not have a prepreg form to act as the glue during lamination. In these cases, a special glue sheet must be used during lamination to fasten the individual layers together.

13.6.4 Subtractive Plating

Subtractive plating is a method of forming traces and other conductive patterns on a PCB by first covering a sheet of laminate with a continuous sheet of copper foil. A layer of etch resist is applied such that it covers the copper pattern that is desired. The panel with protective coating is passed through an etcher that removes (subtracts) the unwanted copper, leaving behind the desired patterns. This is the dominant, almost only, method in common usage in the printed circuit industry today.

13.6.5 Additive Plating

As the name implies, this method of forming conductor patterns involves beginning with a bare substrate and plating on the conductor patterns. There are two methods for doing this: electroless plating on areas sensitized to accept electroless copper and electroplating by first applying a very thin coating of electroless copper over the entire surface to act as a conductive path, followed by electroplating to full thickness.

Additive plating is seen as a method for reducing the amount of chemicals required to manufacture PCBs, and this is true. However, the process does not yield copper sufficiently robust to withstand the handling of normal assembly and rework. As a result, it is not commonly available in production.

13.6.6 Discrete Wire

Discrete wire is a method for forming the wiring layers by rolling round wire into a soft insulating material coated onto the outsides of power plane cores. This method is often referred to as *multiwire*. It is available from only a very small number of manufacturers and offers few advantages over conventional multilayer processing. It is described more fully in Sec. 13.2.3.

13.7 CHOOSING A PACKAGE TYPE AND FABRICATION VENDOR

A key part of arriving at a successful design is choosing PCB materials, component-mounting techniques, and fabrication methods that meet the performance needs of the product being designed while achieving the lowest possible costs. Among the decisions that are part of this process are deciding whether to package a product on one large PCB or several smaller PCBs, whether to spread components out to hold the layer count down or increase layers, move components closer together, and design a smaller PCB, whether to package some components in a multichip module that is then mounted on the PCB, as well as other issues.

Part of this decision-making process is arriving at an overall package choice that can be manufactured by the mainstream fabricators and assemblers. Failure to do this will result in excessively high prices and long lead times stemming from the lack of a competitive supplier base from which to choose. At the extreme, where some of the more exotic materials systems are used, there may be as few as one supplier to turn to. In markets where there is substantial price pressure, such as with disk drives and PCs, it imperative that the design choices be made such that the PCBs can be manufactured at offshore fabricators. Not doing this will place the product at a competitive disadvantage.

13.7.1 Trading Off Number of Layers Against Area

Cost of the bare PCB is often a significant contributor to the overall cost of an assembly. As the number of layers in a PCB increases, the cost increases. A standard practice is to spread components out to make room for the connecting wiring as a way to avoid adding additional wiring layers. As might be expected, there is a point at which PCB size grows to where a smaller PCB with more layers yields a more economical solution. Determining where this breakpoint is requires some knowledge about the PCB fabrication process.

Table 13.4 shows typical costs of four-, six-, and eight-layer 18 in by 24 in standard process panels built at offshore manufacturers. This table can be used to calculate the relative cost of PCBs as layer count is increased to reduce area. While the absolute costs in the table are based on Spring 1995 pricing for Pacific Rim fabricators, the percentage relationships between the costs of panels of various layer counts are a good indicator of relative costs for deciding when to increase layer count and reduce area.

13.7.1.1 Background Information. Multilayer PCBs of six or more layers are normally built on standard 18 in by 24 in panels using pin lamination. Many four-layer PCBs are built offshore using a process called mass lamination with panels sizes of 36 in by 48 in (four times a "standard" panel). The pricing of individual PCBs is based on how many PCBs fit on a stan-

TABLE 13.4 PCB Panel Process Cost vs. Layer Count
Price per panel, $ U.S., Spring 1995

Number of layers	Panels per mo. 100	Panels per mo. 250	Panels per mo. 1000	Panels per mo. 5000
4 mass lam*	$260	$250	$240	$231
4 pin lam†	84	80	77	74
6 pin lam†	113	108	104	100
8 pin lam†	147	140	135	130

To determine the number of PCBs that will fit onto a panel, allow 0.125 in between PCBs and allow 0.75-in margin on all four sides. Net areas are 34.5 in by 46.5 in and 16.5 in by 22.5 in.
For gold plating on connector tips, add up to $2 per PCB.
* 36 in × 48 in panel.
† 18 in × 24 in panel.

dard panel. Therefore, designers need to choose finished PCB sizes with this in mind (assuming the size is negotiable).

The pricing matrix in Table 13.4 is in price per panel based on the following:

- Solder mask over bare copper, silk screen one side only
- Standard multifunctional FR4 laminate
- 1-oz copper inner layers, ½-oz outer layer foil plated up to 1.5 oz nominal
- Thickness accuracy: ±10% overall, ±1.5 mil any dielectric layer
- No controlled impedance requirement or testing
- 1-mil minimum copper plating in holes
- No vias smaller than 13 mils
- Traces and spaces: 7 mil, 7 mil
- Standard delivery: (no acceleration premium)
- Fabrication site: **Pacific Rim Fabricators**
- No gold plating
- Tested to CAD netlist

Since PCB fabrication is based on standard panels, the cost of each PCB is affected by how much of each panel is used to form the PCBs built on it and the amount of the panel that is scrapped. Clearly, the more of each panel that is usable, the lower the average cost of each PCB will be. As one chooses the size and form factor of each PCB, attention should be paid to how many will fit onto a standard panel in order to minimize the scrap material created.

13.7.2 One PCB vs. Multiple PCBs

One way to keep individual PCB layer counts down is to divide the circuitry among several smaller, simpler PCBs. There is a hidden cost associated with doing this. The hidden cost is spread across several organizations, ranging from the design activity, through manufacturing, and into the sales and service organization. The costs are those associated with handling multiple assemblies, such as managing multiple designs and their documentation, managing the procurement, inventory, testing and stocking of multiple assemblies, and the cost of interconnecting the multiple assemblies. In almost all cases, these costs exceed any savings that might be realized by the creation of multiple assemblies.