Electronic components are rarely useful in isolation, and it is usually necessary to connect a number of them together in order to achieve a desired effect. Early electronic circuits were constructed using discrete (individually packaged) components such as transistors, resistors, capacitors, and diodes. These were mounted on a non-conducting board and connected using individual pieces of insulated copper wire. The thankless task of wiring the boards by hand was time-consuming, boring, prone to errors, and expensive.

The First Circuit Boards

The great American inventor Thomas Alva Edison had some ideas about connecting electronic circuits together. In a note to Frank Sprague, founder of Sprague Electric, Edison outlined several concepts for printing additive traces on an insulating base. He even talked about the possibility of using conductive inks (it was many decades before this technology—which is introduced later in this chapter—came to fruition).

In 1903, Albert Hanson (a Berliner living in London) obtained a British patent for a number of processes for forming electrical conductors on an insulating base material. One of these described a technique for cutting or stamping traces out of copper foil and then sticking them to the base. Hanson also came up with the idea of double-sided boards and through-holes (which were selectively connected by wires).

In 1913, Arthur Berry filed a British patent for covering a substrate with a layer of copper and selectively etching parts of it away to leave tracks. In another British patent issued in 1925, Charles Ducas described etching, plating up, and even multi-layer circuit boards (including the means of interconnecting the layers). For the next few decades, however, it was easier and cheaper to wire boards manually. The real push into circuit boards only came with the invention of the transistor and later the integrated circuit.
PCBs and PWBs

By the 1950s, the interconnection technology now known as printed wire boards (PWBs) or printed circuit boards (PCBs) had gained commercial acceptance. Both terms are synonymous, but the former is more commonly used in America, while the latter is predominantly used in Europe and Asia. These circuit boards are often referred to as laminates because they are constructed from thin layers or sheets. In the case of the simpler boards, an insulating base layer has conducting tracks formed on one or both sides. The base layer may technically be referred to as the substrate, but this term is rarely used in the circuit board world.¹

The original board material was Bakelite, but modern boards are predominantly made from woven glass fibers which are bonded together with an epoxy. The board is cured using a combination of temperature and pressure, which causes the glass fibers to melt and bond together, thereby giving the board strength and rigidity. These materials may also be referred to as organic substrates, because epoxies are based on carbon compounds as are living creatures. The most commonly used board material of this type is known as FR4, where the first two characters stand for flame retardant, and you can count the number of people who know what the “4” stands for on the fingers of one hand.

To provide a sense of scale, a fairly representative board might be 15 cm × 20 cm in area and in the region of 1.5 mm to 2.0 mm thick, but they can range in size from 2 cm × 2 cm or smaller (and thinner) to 50 cm × 50 cm or larger (and thicker).²

Subtractive Processes

In a subtractive process, a thin layer of copper foil in the order of 0.02 mm thick is bonded to the surface of the board. The copper’s surface is coated with an organic resist, which is cured in an oven to form an impervious layer (Figure 18-1).

---

¹ See also the glossary definition of “substrate.”

² These dimensions are not intended to imply that circuit boards must be square or even rectangular. In fact, circuit boards may be constructed with whatever outline is necessary to meet the requirements of a particular enclosure: for example, the shape of a car dashboard.
Next, an optical mask is created with areas that are either transparent or opaque to ultraviolet light. The mask is usually the same size as the board, and can contain hundreds of thousands of fine lines and geometric shapes.

The mask is placed over the surface of the board, which is then exposed to ultraviolet (UV) light. This ionizing radiation passes through the transparent areas of the mask to break down the molecular structure of the resist. After the board has been exposed, it is bathed in an organic solvent, which dissolves the degraded resist. Thus, the pattern on the mask has been transferred to a corresponding pattern in the resist (Figure 18-2).
A process in which ultraviolet light passing through the transparent areas of the mask causes the resist to be degraded is known as a *positive-resist* process; *negative-resist* processes are also available. The following discussions assume positive-resist processes unless otherwise noted.

After the unwanted resist has been removed, the board is placed in a bath containing a cocktail based on sulfuric acid, which is agitated and aerated to make it more active. The sulfuric acid dissolves any exposed copper not protected by the resist in a process known as *etching*. The board is then washed to remove the remaining resist. Thus, the pattern in the mask has now been transferred to a corresponding pattern in the copper (Figure 18-3).

![Figure 18-3. Subtractive process: removing the unwanted copper](image)

This type of process is classed as *subtractive* because the board is first covered with the conductor and any unwanted material is then removed, or subtracted. As a point of interest, much of this core technology predates the modern electronics industry. The process of copper etching was well known by the printing industry in the 1800s, and opto-lithographic techniques involving organic resists were used to create printing plates as early as the 1920s. These existing processes were readily adopted by the fledgling electronics industry.

As process technologies improved, it became possible to achieve ever-finer features. By 2002, a large proportion of boards were using lines and spaces of

---

3 In a *negative-resist* process the ultraviolet radiation passing through the transparent areas of the mask is used to cure the resist. The remaining uncured areas are then removed using an appropriate solvent. Thus, a mask used in a negative-resist process is the photographic negative of one used in a positive-resist process to achieve the same effect; that is, the transparent areas are now opaque and vice versa.
5 mils or 4 mils, with some as small as 3 mils (one mil is one thousandth of an inch). In this context, the term “lines” refers to the widths of the tracks, while “spaces” refers to the gaps between adjacent tracks.

**Additive Processes**

An additive process does not involve any copper foil being bonded to the board. Instead, the coating of organic resist is applied directly to the board's surface (Figure 18-4).

![Resist on FR4](image)

**Figure 18-4. Additive process: applying resist to bare board**

Once again the resist is cured, an optical mask is created, ultraviolet light is passed through the mask to break down the resist, and the board is bathed in an organic solvent to dissolve the degraded resist (Figure 18-5).

![Ultraviolet radiation](image)

**Figure 18-5. Additive process: degrading the resist and exposing the FR4**

---

Note that a mask used in an additive process is the photographic negative of one used in a subtractive process to achieve the same effect; that is, the transparent areas are now opaque and vice versa.
After the unwanted resist has been removed, the board is placed in a bath containing a cocktail based on copper sulfate where it undergoes a process known as *electroless plating*. Tiny crystals of copper grow on the exposed areas of the board to form copper tracks. The board is then washed in an appropriate solvent to remove the remaining resist (Figure 18-6).

A process of this type is classed as *additive* because the conducting material is only grown on, or added to, specific areas of the board. Additive processes are increasing in popularity because they require less processing and result in less wasted material. Additionally, fine tracks can be grown more accurately in additive processes than they can be etched in their subtractive counterparts. These processes are of particular interest for high-speed designs and microwave applications, in which conductor thicknesses and controlled impedances are critical. Groups of tracks, individual tracks, or portions of tracks can be built up to precise thicknesses by iterating the process multiple times with selective masking.

**Single-sided Boards**

It probably comes as no great surprise to find that *single-sided* boards have tracks on only one side. These tracks, which may be created using either subtractive or additive processes, are terminated with areas of copper known as *pads*. The shape of the pads and other features are, to some extent, dictated by the method used to attach components to the board. Initially, these discussions will assume that the components are to be attached using a technique known as
through-hole, which is described in more detail below. The pads associated with the through-hole technique are typically circular, and holes are drilled through both the pads and the board using a computer-controlled drilling machine (Figure 18-7).  

![Figure 18-7. Single-sided boards: drilling the holes](image)

Once the holes have been drilled, an electroless plating process referred to as tinning is used to coat the tracks and pads with a layer of tin-lead alloy. This alloy is used to prevent the copper from oxidizing and provides protection against contamination. The deposited alloy has a rough surface composed of vertical crystals called spicules, which, when viewed under a microscope, resemble a bed of nails. To prevent oxygen from reaching the copper through pinholes in the alloy, the board is placed in a reflow oven where it is heated by either infrared (IR) radiation or hot air. The reflow oven causes the alloy to melt and form a smooth surface (Figure 18-8).

After the board has cooled, a layer known as the solder mask is applied to the surface carrying the tracks (the purpose of this layer is discussed in the next section). One common technique is for the solder mask to be screen printed onto the board. In this case, the screen used to print the mask has patterns that leave the areas around the pads exposed, and the mask is then cured in an

---

5 This is usually referred to as an NC drilling machine, where NC stands for “Numerically Controlled.”
oven. In an alternative technique, the solder mask is applied across the entire surface of the board as a film with an adhesive on one side. In this case, a further opto-lithographic stage is used to cure the film with ultraviolet light. The optical mask used in this process contains opaque patterns, which prevent the areas around the pads from being cured; these areas are then removed using an appropriate solvent (Figure 18-9).
Beware! Although there are relatively few core processes used to manufacture circuit boards, there are almost endless variations and techniques. For example, the tracks and pads may be created before the holes are drilled or vice versa. Similarly, the tin-lead alloy may be applied before the solder mask or vice versa. This latter case, known as *solder mask over bare copper* (SMOBC), prevents solder from leaking under the mask when the tin-lead alloy melts during the process of attaching components to the board. Thus, the tin-lead alloy is applied only to any areas of copper that are left exposed by the solder mask, such as the pads at the end of the tracks. As there are so many variations and techniques, these discussions can only hope to offer an overview of the main concepts.

**Lead Through-Hole (LTH)**

Prior to the early 1980s, the majority of integrated circuits were supplied in packages that were attached to a circuit board by inserting their leads through holes drilled in the board. This technique, which is still widely used, is known as *lead through-hole (LTH)*, *plated through-hole (PTH)*, or, more concisely, *through-hole*. In the case of a single-sided board, any components that are attached to the board in this fashion are mounted on the opposite side to the tracks. This means that any masks used to form the tracks are actually created as mirror-images of the required patterns (Figure 18-10).

![Figure 18-10. Lead through-hole (LTH)](image-url)
The act of attaching components is known as *populating* the board, and the area of the board occupied by a component is known as its *footprint*. Early manufacturing processes required the boards to be populated by hand, but modern processes make use of automatic insertion machines. Similarly, component leads used to be hand-soldered to the pads, but most modern processes employ automatic *wave-soldering* machines (Figure 18-11).

A wave-soldering machine is based on a tank containing hot, liquid *solder*. The machine creates a wave (actually a large ripple) of solder which travels across the surface of the tank. Circuit boards populated using the through-hole technique are passed over the machine on a conveyer belt. The system is carefully controlled and synchronized such that the solder wave brushes across the bottom of the board only once.

The solder mask introduced in the previous section prevents the solder from sticking to anything except the exposed pads and component leads. Because the solder is restricted to the area of the pads, surface tension causes it to form

---

6 An alloy of tin and lead with a relatively low melting point.
good joints between the pads and the component leads. Additionally, capillary action causes the solder to be drawn up the hole, thereby forming reliable, low-resistance connections. If the solder mask were omitted, the solder would run down the tracks away from the component leads. In addition to forming bad joints, the amount of heat absorbed by the tracks would cause them to separate from the board (this is not considered to be a good thing to happen).

**Surface Mount Technology (SMT)**

In the early 1980s, new techniques for packaging integrated circuits and populating boards began to appear. One of the more popular is *surface mount technology* (SMT), in which component leads are attached directly to pads on the surface of the board. Components with packages and lead shapes suitable for this technology are known as *surface mount devices* (SMDs). One example of a package that achieves a high lead count in a small area is the *quad flat pack* (QFP), in which leads are present on all four sides of a thin square package.\(^7\)

Boards populated with surface mount devices are fabricated in much the same way as their through-hole equivalents (except that the pads used for attaching the components are typically square or rectangular and do not have any holes drilled through them).\(^8\) However, the processes begin to diverge after the solder mask and tin-lead plating layers have been applied. A layer of solder paste is screen-printed onto the pads, and the board is populated by an automatic *pick-and-place machine*, which pushes the component leads into the paste (Figure 18-12).

Thus, in the case of a single-sided board, the components are mounted on the same side as the tracks. When all of the components have been attached, the solder paste is melted to form good conductive bonds between their leads and the board’s pads. The solder paste can be melted by placing the board in a *reflow oven* where it is heated by infrared (IR) radiation or hot air. Alternatively, the solder may be melted using *vapor-phase soldering*, in which the board is lowered into the vapor-cloud above a tank containing boiling hydrocarbons.

---

*Other packaging styles such as *pad grid arrays* (PGAs) and *ball grid arrays* (BGAs) — which are also suitable for surface mount technology — are introduced in more detail in Chapter 20.*

*Actually, the pads may have holes in the case of the microvia technologies discussed later in this chapter.*
However, vapor-phase soldering is becoming increasingly less popular due to environmental concerns.

Surface mount technology is well suited to automated processes. Due to the fact that the components are attached directly to the surface of the board, they can be constructed with leads that are finer and more closely spaced than their through-hole equivalents. The result is smaller and lighter packages which can be mounted closer together and occupy less of the board’s surface area, which is referred to as real estate. This, in turn, results in smaller, lighter, and faster circuit boards. The fact that the components do not require holes for their leads is also advantageous, because drilling holes is a time-consuming and expensive process. Additionally, if any holes are required to make connections through the board (as discussed below), they can be made much smaller because they do not have to accommodate component leads.  

9 Typical pad and hole diameters are presented in the Holes versus Vias section later in this chapter.
**Double-sided Boards**

There is a simple game played by children all over the world. The game commences by drawing three circles and three squares on a piece of paper, and then trying to connect each circle to each square without any of the connecting lines crossing each other (Figure 18-13).

Children can devote hours to this game, much to the delight of their parents. Unfortunately, there is no solution, and one circle-square pair will always remain unconnected. This simple example illustrates a major problem with single-sided boards, which may have to support large numbers of component leads and tracks. If any of the tracks cross, an undesired electrical connection will be made and the circuit will not function as desired. One solution to this dilemma is to use wire links called *jumpers* (Figure 18-14).
Unfortunately, the act of inserting a jumper is as expensive as for any other component. At some point it becomes more advantageous to employ a double-sided board, which has tracks on both sides.

Initially, the construction of a double-sided board is similar to that for a single-sided board. Assuming a subtractive process, copper foil is bonded to both sides of the board, and then organic resist is applied to both surfaces and cured. Separate masks are created for each side of the board, and ultraviolet light is applied to both sides. The ultraviolet radiation that is allowed to pass through the masks degrades the resist, which is then removed using an organic solvent. Any exposed copper that is not protected by resist is etched, the remaining resist is removed, and holes are drilled. However, a double-sided board now requires an additional step. After the holes have been drilled, a plating process is used to line them with copper (Figure 18-15).

Instead of relying on jumpers, a track can now pass from one side of the board to the other by means of these copper-plated holes, which are known as vias.10,11 The tracks on one side of the board usually favor the y-axis (North-South), while the tracks on the other side favor the x-axis (East-West).

10 The term via is taken to mean a conducting path linking two or more conducting layers, but does not include a hole accommodating a component lead (see also the following section entitled Holes versus Vias).

11 There are a number of alternative techniques that may be used to create circuit board vias. By default, however, the term is typically understood to refer to holes plated with copper as described here.
The inside of the vias and the tracks on both sides of the board are plated with tin-lead alloy, and solder masks are applied to both surfaces (or vice versa in the case of the SMOBC-based processes, which were introduced earlier).

Some double-sided boards are populated with through-hole or surface mount devices on only one side. Some boards may be populated with through-hole devices on one side and surface mount devices on the other. And some boards may have surface mount devices attached to both sides. This latter case is of particular interest, because surface mount devices do not require holes to be drilled through the pads used to attach their leads (they have separate fan-out vias as discussed below). Thus, in surface mount technology, it is possible to place two devices directly facing each other on opposite sides of the board without making any connections between them.

Having said this, in certain circumstances it may be advantageous to form connections between surface mount devices directly facing each other on opposite sides of the board. The reason for this is that a through-board connection can be substantially shorter than an equivalent connection between adjacent devices on the same side of the board. Thus, this technique may be of use for applications such as high-speed data buses, because shorter connections result in faster signals.

**Holes versus Vias**

Manufacturers of circuit boards are very particular about the terminology they use, and woe betide anyone caught mistakenly referring to a hole as a via, or vice versa. Figure 18-16 should serve to alleviate some of the confusion.

In the case of a single-sided board (as illustrated in Figures 18-7 and 18-14), a hole that is used to accommodate a through-hole component lead is simply referred to as a hole. By comparison, in the case of double-sided boards (or multilayer boards as discussed below), a hole that is used to accommodate a through-hole component lead is plated with copper and is referred to as a plated through-hole. Additionally, a hole that is only used to link two or more conducting layers, but does not accommodate a component lead, is referred to as a via (or, for those purists among us, an interstitial via). The qualification attached to this latter case is important, because even if a hole that is used to accommodate a component lead is also used to link two or more conducting layers, it is still referred to as a plated through-hole and not a via (phew!).
Due to the fact that vias do not have to accommodate component leads, they can be created with smaller diameters than plated through-holes, thereby occupying less of the board’s real estate. To provide a sense of scale, the diameters of plated through-holes and their associated pads are usually in the order of 24 mils (0.6 mm) and 48 mils (1.2 mm), respectively, while the diameters of vias and via pads are typically 12 mils (0.3 mm) and 24 mils (0.6 mm), respectively.

Finally, in the case of surface mount devices attached to double-sided boards (or multilayer boards as discussed below), each component pad is usually connected by a short length of track to a via, known as a fan-out via, which forms a link to other conducting layers (an example of a fan-out via is shown in Figure 18-16). However, if this track exceeds a certain length (it could meander all the way around the board), an otherwise identical via at the end would be simply referred to as a via. Unfortunately,

---

12 Some engineers attempt to differentiate vias that fall inside the device’s footprint (under the body of the device) from vias that fall outside the device’s footprint by referring to the former as fan-in vias, but this is not a widely used term.
there is no standard length of track that differentiates a fan-out via from a standard via, and any such classification depends solely on the in-house design rules employed by the designer and board manufacturer.

**Multilayer Boards**

It is not unheard of for a circuit board to support thousands of components and tens of thousands of tracks and vias. Double-sided boards can support a higher population density than single-sided boards, but there quickly comes a point when even a double-sided board reaches its limits. A common limiting factor is lack of space for the necessary number of vias. In order to overcome this limitation, designers may move onwards and upwards to *multilayer* boards.

A multilayer board is constructed from a number of single-sided or double-sided sub-boards. The individual sub-boards can be very thin, and multilayer boards with four or six conducting layers are usually around the same thickness as a standard double-sided board. Multilayer boards may be constructed using a double-sided sub-board at the center with single-sided sub-boards added to each side (Figure 18-17a). Alternatively, they may be constructed using only double-sided sub-boards separated by non-conducting layers of semi-cured FR4 known as *prepreg* (Figure 18-17b).

---

13 The term “sub-board” is not an industry standard, and it is used in these discussions only to distinguish the individual layers from the completed board.

14 This technique is usually reserved for boards that carry only four conducting layers.
After all of the layers have been etched to form tracks and pads, the subboards and prepreg are bonded together using a combination of temperature and pressure. This process also serves to fully cure the prepreg. Boards with four conducting layers are typical for designs intended for large production runs. The majority of multilayer boards have less than ten conducting layers, but boards with twenty-four conducting layers or more are not outrageously uncommon, and some specialized boards like backplanes (as discussed later in this chapter) may have 60 layers or more!

**Through-Hole, Blind, and Buried Vias**

To overcome the problem of limited space, multilayer boards may make use of through-hole, blind, and buried vias. A through-hole via passes all the way through the board, a blind via is only visible from one side of the board, and a buried via is used to link internal layers and is not visible from either side of the board (Figure 18-18).

Unfortunately, although they help to overcome the problem of limited space, blind and buried vias significantly increase the complexity of the manufacturing process. When these vias are only used to link both sides of a subboard, that board must be drilled individually and a plating process used to line its vias with copper.
Similarly, when these vias only pass through a number of sub-boards, those boards must be bonded together, drilled, and plated as a group. Finally, after all of the sub-boards have been bonded together, any holes that are required to form plated through-holes and vias are drilled and plated. Blind and buried vias can greatly increase the number of tracks that a board can support but, in addition to increasing costs and fabrication times, they can also make it an absolute swine to test.

**Power and Ground Planes**

The layers carrying tracks are known as the *signal layers*. In a multilayer board, the signal layers are typically organized so that each pair of adjacent layers favors the y-axis (North-South) and the x-axis (East-West), respectively. Additionally, two or more conducting layers are typically set aside to be used as *power* and *ground* planes. The power and ground planes usually occupy the central layers, but certain applications have them on the board’s outer surfaces. This latter technique introduces a number of problems, but it also increases the board’s protection from external sources of noise such as electromagnetic radiation.

Unlike the signal layers, the bulk of the copper on the power and ground planes remains untouched. The copper on these layers is etched away only in those areas where it is not required to make a connection. For example, consider a through-hole device with eight leads. Assume that leads 4 and 8 connect to the ground and power planes respectively, while the remaining leads are connected into various signal layers (Figure 18-19).

For the sake of simplicity, the exploded view in Figure 18-19 only shows the central sub-board carrying the power planes; the sub-boards carrying the signal layers would be bonded to either side. Also note that the holes shown in the prepreg in the exploded view would not be drilled and plated until all of the sub-boards had been bonded together.

In the case of component leads 1, 2, 3, 5, 6, and 7, both the power and ground planes have copper removed around the holes. These etched-away areas, which are referred to as *anti-pads*, are used to prevent connections to the planes when the holes are plated. Similarly, the power plane has an anti-pad associated with lead 4 (the ground lead), and the ground plane has an anti-pad associated with lead 8 (the power lead).
The power plane has a special pattern etched around the hole associated with lead 8 (the power lead), and a similar pattern is present on the ground plane around the hole associated with lead 4 (the ground lead). These patterns, which are referred to as thermal relief pads, are used to make electrical connections to the power and ground planes. The spokes in the thermal relief pads are large enough to allow sufficient current to flow, but not so large that they will conduct too much heat.

Thermal relief pads are necessary to prevent excessive heat from being absorbed into the ground and power planes when the board is being

---

The pattern of a thermal relief pad is often referred to as a “wagon wheel,” because the links to the plated-through hole or via look like the spokes of a wheel. Depending on a number of factors, a thermal relief pad may have anywhere from one to four spokes.
soldered.\textsuperscript{16} When the solder is applied, a surface-tension effect known as capillary action sucks it up the vias and plated through-holes. The solder must be drawn all the way through to form reliable, low-resistance connections. The amount of copper contained in the power and ground planes can cause problems because it causes them to act as \textit{thermal heat sinks}. The use of thermal relief pads ensures good electrical connections, while greatly reducing heat absorption. If the thermal relief pads were not present, the power and ground planes would absorb too much heat too quickly. This would cause the solder to cool and form plugs in the vias resulting in unreliable, high-resistance connections. Additionally, in the case of wave soldering, so much heat would be absorbed by the power and ground planes that all of the layers forming the board could separate in a process known as \textit{delamination}.

A special flavor of multilayer boards known as \textit{Padcap} (or “Pads-Only-Outer-Layers”) are sometimes used for high-reliability military applications. Padcap boards are distinguished by the fact that the outer surfaces of the board only carry pads, while any tracks are exclusively created on inner layers and connected to the pads by vias. Padcap technology offers a high degree of protection in hostile environments because all of the tracks are inside the board.

\textbf{Microvia, HDI, and Build-up Technologies}

One exciting recent circuit board development is that of \textit{microvia} technology, which officially refers to any vias and via pads with diameters of 6 mils (0.15 mm) and 12 mils (0.3 mm)—or smaller—respectively. By 2002, boards using 4 mil, 3 mil, and 2 mil diameter microvias were reasonably common, and some folks were even using 1 mil microvias.

In a typical implementation, one or two microvia layers are added to the outer faces of a standard multiplayer board, which is why the term \textit{buildup technology} is also commonly used when referring to microvia boards. Just to make things even more confusing, the term \textit{high density interconnect (HDI)} is also commonly used. In reality, the terms \textit{microvia}, \textit{HDI}, and \textit{buildup technology} are synonymous.

\textsuperscript{16} For future reference, the term \textit{pad-stack} refers to any pads, anti-pads, and thermal relief pads associated with a particular via or plated-through hole as it passes through the board.
Microvias—which are actually blind vias that just pass through one or more of the buildup layers on the outer faces of the main board—may be created using a variety of techniques. One common method is to use a laser, which can "drill" 20,000+ microvias per second.

The reason microvias are so necessary is largely tied to recent advances in device packaging technologies. It's now possible to get devices with 1,000 pins (or pads or leads or connections or whatever you want to call them), and packages with 2,000 and 4,000 pins are on the way. These pins are presented as an array across the bottom of the device. The pin pitch (the distance between adjacent pins) has shrunk to the extent that it simply isn't possible to connect the package to a board using conventional via diameters and line widths (there just isn't enough space to squeeze in all of the fan-out vias and route all of the tracks). The use of microvia technology alleviates this problem, making it possible to place a microvia in the center of a component pad, thereby eliminating the need for fan-out vias.

As a simple example, consider a simple 8-pin TSOP type package. Using conventional technologies, each of the component pads would be connected to a fan-out via (Figure 18-20a). (This is obviously a much-magnified view, and this simple component would actually be only a few millimeters in size.) Compared to the footprint of the device itself, having these fan-out vias means
that this component now occupies a substantial amount of the board’s real
estate. By comparison, placing vias in the component’s pads using microvia
technology means that the component occupies much less space on the board
(Figure 18-20b).

Although using microvia technology isn’t cheap per se, it can actually end
up being very cost effective. In one example of which the author is aware, the
use of microvias enabled an 18-layer board to be reduced to 10 layers, made the
board smaller, and halved the total production cost.

**Discrete Wire Technology**

Discrete wire technology is an interesting discipline that has enjoyed only
limited recognition, but its proponents continue to claim that it’s poised to
emerge as the technology-of-choice for designs that demand the highest signal
speeds and component densities. Circuit boards created using this technology
are known as **discrete wired boards** (DWBs).

**Multiwire Boards**

The earliest form of discrete wire technology, commonly known as
multiwire,\(^{17,18}\) was developed in the late 1960s, and had gained both military
and commercial acceptance by the late 1970s. The discrete wire process
commences with a conventional FR4 base layer with copper foil bonded to
both sides to form the power and ground planes.\(^{19}\) After any thermal relief
pads and anti-pads have been etched into the copper using conventional
opto-lithographic processes (Figure 18-21), a layer of insulating prepreg is
bonded to each side and cured. This is followed by a layer of adhesive,\(^{20}\) or
wiring film, which is applied using a hot roll laminating machine.

---

\(^{17}\) Multiwire is a registered trademark of Advanced Interconnection Technology (AIT), Islip,
New York, USA.

\(^{18}\) Many thanks for the wealth of information on discrete wire technology, which was provided
by Hitachi Chemical Electro-Products Division (Atlanta, Georgia, USA), I-CON Industries
(Euless, Texas, USA), and MW Technologies (Aldershot, Hampshire, England).

\(^{19}\) These discussions concentrate on a reasonably simple implementation: more complex
boards with multiple and/or split power and ground planes are also available.

\(^{20}\) Manufacturers of discrete wire boards are trying to discard the term “adhesive” on the basis
that this layer is not actually “sticky.”
Chapter Eighteen

Thermal relief pad

Anti-pad

Copper

power plane

FR4

(Only the power plane is shown for clarity. The ground plane would be bonded to the bottom of the board.)

Figure 18-21. Multiwire boards: preparing the core

Wire cropped over the center of this anti-pad

Wire passes directly over the center of this anti-pad

Wiring film

Prepreg

Power plane

FR4

Copper wire

Polyimide insulation

Crossover point (no connection is made between the wires because they are insulated)

Figure 18-22. Multiwire boards: ultrasonically bonding the wire
A special computer-controlled wiring machine\textsuperscript{21} is used to ultrasonically bond extremely fine insulated wires\textsuperscript{22} into the wiring film. The wire is routed through all the points to which it will eventually be connected. When the last point of a particular net is reached, the wiring machine cuts the wire and positions the wiring head at the next starting point (Figure 18-22).

The wire has an insulating coat of polyimide and can be wired orthogonally (North-South and East-West), diagonally, or as a combination of both. Due to the fact that the wires are insulated, they can cross over each other without making any unwanted electrical connections. After all the wires have been applied to one side of the board, the board is inverted and more wires can be applied to the other side. The majority of such boards have just these two wiring layers, one on each side. If necessary, however, additional layers of prepreg and wiring film can be bonded to the outer surfaces of the board and more wiring layers can be applied. It is not unusual for a very dense multiwire board to support four wiring layers, two on each side. Supporters of this technology claim that an equivalent multilayer board might require twenty or more signal layers to achieve the same effect.

After all of the wiring layers have been created, a final layer of prepreg and copper foil are laminated onto the outer surfaces of each side of the board, and any necessary holes are drilled through the board (Figure 18-23).

The drilling process leaves the wire ends exposed in the holes. The board is now exposed to a polyimide-selective etchant, which etches away the insulation at the end of wires. Although the insulation is only etched away to a depth of approximately 0.05 mm, this is sufficient for the subsequent plating process to form a wrap-around joint, which provides mechanical reliability as opposed to a simple electrical connection.

The holes are plated with copper to form plated through-holes and vias, and then the outer surfaces of the board are etched to form any via pads and component mounting pads (Figure 18-24). (Note that the pads shown in this illustration indicate that the board is to be populated with surface mount devices.)

\textsuperscript{21} This is usually referred to as an NC wiring machine, where NC stands for “Numerically Controlled.”

\textsuperscript{22} The wires used in the original process were 0.16 mm in diameter. Later processes made use of additional wire diameters of 0.10 mm and 0.06 mm.
Figure 18-23. Multiwire boards: drilling the holes

Figure 18-24. Multiwire boards: plating and etching
Finally, the board is tinned, solder masks are applied, and components are attached in the same way as for standard printed circuit boards. In fact, a completed multiwire board appears identical to a standard printed circuit board.

**Microwire Boards**

If multiwire is a niche market, then its younger brother, *microwire*,\(^2^3\) forms a niche within a niche. Microwire augments the main attributes of multiwire with laser-drilled blind vias, allowing these boards to support the maximum number of wires and components.\(^2^4\) Due to the large numbers of components they support, microwire boards typically have to handle and dissipate a great deal of heat, so their cores may be formed from sandwiches of copper-Invar-copper\(^2^5\) or similar materials. This sandwich structure is used because the resulting coefficient of thermal expansion of the core combined with the other materials used in the board is almost equal to that of any components with ceramic packages that may be attached to the board. The coefficient of thermal expansion defines the amount a material expands and contracts due to changes in temperature. If materials with different coefficients are bonded together, changes in temperature will cause shear forces at the interface between them. Engineering the board to have a similar coefficient to any ceramic component packages helps to ensure that a change in temperature will not result in those components leaping off the board.

To provide a sense of scale, in one of the more common microwire implementations each copper-Invar-copper sandwich is 0.15 mm thick. The board's central core is formed from two of these sandwiches separated by a layer of insulating prepreg, where the upper sandwich forms the power plane and the lower sandwich forms the ground plane (Figure 18-25).\(^2^6\)

\(^{2^3}\) Microwire is a registered trademark of Advanced Interconnection Technology (AIT), Islip, New York, USA.

\(^{2^4}\) Actually, as multiwire boards can also be created with blind vias, and as the diameter of these vias can be as small as their microwire equivalents, someone less trusting than the author might wonder whether the main justification for microwire was the fact that some people simply wanted to play with lasers.

\(^{2^5}\) Invar is an alloy similar to bronze.

\(^{2^6}\) More complex structures are also possible, including multiple and/or split power and ground planes.
Anti-pad

Thermal relief pad

Bounce pad

Power plane (copper-Invar-copper sandwich)

FR4

Figure 18-25. Microwire boards: preparing the core

Wire cropped over this anti-pad

Wire passes directly over the center of this bounce pad

Wiring film

Cured epoxy

Power plane

FR4

Copper wire

Polyimide insulation

Cured epoxy

Wiring film

Figure 18-26. Microwire boards: ultrasonically bonding the wire
Thermal relief pads and anti-pads are etched into the upper and lower copper-Invar-copper sandwiches in the same manner as for multiwire boards. The etch is performed through all of the layers forming the sandwich, all the way down to the surface of the prepreg in the center. In addition to the standard thermal relief and anti-pads, microwire boards also contain special pads known as bounce pads for use during the laser drilling process discussed below.

Following the etching process, a layer of liquid epoxy is applied to the outer surfaces. After this epoxy has been cured, a layer of wiring film is applied into which wires are ultrasonically bonded as for multiwire boards. However, in the case of microwire, the wires are always 0.06 mm in diameter (Figure 18-26). As with multiwire, the majority of microwire boards have just two wiring layers, one on each side. However, if necessary, additional layers of liquid epoxy and wiring film can be bonded to the outer surfaces of the board and more wiring layers can be applied.

After all of the wiring layers have been created, a final encapsulation layer of liquid epoxy is applied to the outer surfaces of the board and cured. Now comes the fun part of the process, in which blind vias are drilled using a carbon-dioxide laser (Figure 18-27). The laser evaporates the organic materials and strips the polyimide coating off the top of wire, but does not affect the wire itself. The laser beam continues down to the bounce pad, which reflects, or bounces, it back up, thereby stripping the polyimide coating off the bottom of the wire. To complete the board, holes for through-hole vias and, possibly, plated through-holes are created using a conventional drilling process. The through-hole vias, laser-drilled blind vias, and plated through-holes are plated with copper, along with any via pads and component pads which are plated onto the board’s outer surfaces (Figure 18-28), and standard tinning and solder mask processes are then applied.

---

27 Liquid epoxy is used to cover the wires because the glass fibers in prepreg would be impervious to the subsequent laser-drilling operations.
Mechanically drilled through-holes.

Laser-drilled blind hole. The laser beam strips the polyimide insulation off the wire.

Cured epoxy
Wiring film
Cured epoxy
Power plane
FR4

Bounce pad reflects laser beam

Figure 18-27. Microwire boards: laser-drilling the blind vias

Anti-pad prevents this through-hole via's plating from connecting into the power plane

Component pads plated onto surface of board

Thermal relief pad connects this through-hole via's plating into the power plane

Plating the vias connects the pads to the wires

Ring etched around the bounce pad acts like an anti-pad and prevents this blind via's plating from connecting into the power plane

Figure 18-28. Microwire boards: plating the pads and vias
The Advantages of Discrete Wire Technology

Since its inception, discrete wire technology has attracted a small number of dedicated users, but has never achieved widespread recognition. Both multiwire and microwire processes are more expensive than their conventional printed wire counterparts, and therefore they are normally reserved for the most dense and electrically sophisticated applications. In fact, this discipline was originally regarded by many designers as simply being a useful prototyping technology, which offered fast turnaround times on design modifications. However, as many of its devotees have known for a long time, discrete wire technology offers a number of advantages over traditional printed wire boards. One obvious advantage is that discrete wired boards with only two wiring layers (one on each side) can provide an equivalent capability to ten or more signal layers in their multilayer counterparts. Thus, discrete wire technology can offer substantially thinner and lighter circuit boards.

Additionally, discrete wire technology is particularly advantageous in the case of high-speed applications. One major requirement of high-speed designs is to control the impedance (capacitance and inductance) of the interconnect, where the impedance is a function of the distance of the tracks from the power and ground planes. In the case of multilayer boards, the tracks keep on changing their distance from the planes as they pass through vias from one layer to another. To compensate for this, additional ground planes must be added between each pair of signal layers, which increases the thickness, weight, and complexity of the boards. By comparison, the wires on discrete wire boards maintain a constant distance from the planes.

Another consideration is the vias themselves. Transmitting a signal down a conducting path may be compared to shouting down a corridor, where any sharp turns in the corridor cause reflections and echoes. Similarly, the vias in a multilayer board have significant capacitive and inductive parasitic effects associated with them, and these effects cause deterioration and corruption of the signals being passed through them. In the case of discrete wire technology, the points at which wires cross over each other have similar effects, but they are orders of magnitude smaller.

Yet another consideration is that, as the frequency of signals increases, electrons start to be conducted only on the outer surface, or skin, of a conductor. This phenomenon is known as the skin effect. By the nature of their construction,
the etched tracks on multilayer boards have small imperfections at their edges. These uneven surfaces slow the propagation of signals and cause noise. By comparison, the wire used in discrete wire technology is uniform to ±0.0025 mm, which is smoother than etched tracks by a factor of 10:1.

Last but not least, the optimal cross-section for a high-speed conductor is circular rather than square or rectangular, and a constant cross-section should be maintained throughout the conductor's length. Both of these criteria are met by discrete wire technology, which, in fact, offers interconnections that are close to ideal transmission lines. The end result is that discrete wire boards can operate at frequencies that are simply not possible with their traditional multi-layer counterparts. It is for this reason that discrete wire technology may yet be poised to emerge as the technology-of-choice for designers of high-speed circuit boards.

**Backplanes and Motherboards**

Another flavor of multilayer boards, known as backplanes, have their own unique design constraints and form a subject in their own right. Backplanes usually have a number of connectors into which standard circuit boards are plugged (Figure 18-29).

Backplanes typically do not carry any active components such as integrated circuits, but they often carry passive components such as resistors and capacitors. If a backplane does contain active components, then it is usually referred to as a motherboard. In this case, the boards plugged into it are referred to as daughter boards or daughter cards.

Because of the weight that they have to support, backplanes for large systems can be 1 cm thick or more. Their conducting layers have thicker copper plating than standard boards and the spaces between adjacent tracks are wider to reduce noise caused by inductive effects.

Backplanes may also support a lot of hardware in the form of bolts, earth straps, and power cables. It is not unusual for a backplane to have multiple power planes such as +5 volts, −5 volts, +12 volts, −12 volts, +24 volts, and −24 volts. Each power plane typically has an independent ground plane associated

---

28 Discrete wire boards can be constructed using microcoaxial cable, and such boards can operate at frequencies that would make your eyes water!
with it to increase noise immunity. So, our hypothetical backplane would require 12 conducting layers for the power and ground planes alone. Additionally, systems containing both analog and digital circuits often require independent power and ground planes for purposes of noise immunity. Backplanes also require excellent thermal tolerance, because some of the more heroic systems can consume upwards of 200 amps and generate more heat than a rampaging herd of large electric radiators.

Backplanes may be constructed using multilayer techniques or discrete wire technology. In fact, discrete wire technology is starting to see increased use in backplane applications because of its high performance, and also due to the resulting reduction in layers, thickness, and weight.

**Conductive Ink Technology**

The underlying concept of conductive ink technology is relatively simple. Tracks are screen-printed onto a bare board using a conducting ink, which is then cured in an oven. Next, a dielectric, or insulating, layer is screen-printed over the top of the tracks. The screen used to print the dielectric layer is patterned so as to leave holes over selected pads on the signal layer. After the dielectric layer has been cured, the cycle is repeated to build a number of signal layers separated by dielectric layers. The holes patterned into the dielectric layers are used to form vias between the signal layers (Figure 18-30).
Finally, plated through-holes and vias can be created, and components can be mounted, using the standard processes described previously. The apparent simplicity of the conductive ink technique hides an underlying sophistication in materials technology. Early inks were formed from resin pastes loaded with silver or copper powder. These inks required high firing temperatures to boil off the paste and melt the powder to form conducting tracks. Additionally, the end product was not comparable to copper foil for adhesion, conductivity, or solderability.

In the early 1990s, new inks were developed based on pastes containing a mixture of two metal or alloy powders. One powder has a relatively low melting point, while the other has a relatively high melting point. When the board is cured in a reflow oven at temperatures as low as 200°C, a process called sintering occurs between the two powders, resulting in an alloy with a high melting point and good conductivity.

Conductive ink technology has not yet achieved track widths as fine as traditional circuit board processes, but it does have a number of attractive features, not the least of which is that it uses commonly available screen-printing equipment. Modern inks have electrical conductivity comparable to copper and they work well with both wave soldering and reflow soldering techniques. Additionally, these processes generate less waste and are more cost-effective and efficient than the plating and etching of copper tracks.
Chip-On-Board (COB)

Chip-on-board (COB) is a relatively modern process that only began to gain widespread recognition in the early 1990s, but which is now accepted as a common and cost-effective die attachment technique. As the name implies, unpackaged integrated circuits are mounted directly onto the surface of the board. The integrated circuits are mechanically and electrically connected using similar wire bonding, tape-automated bonding, and flip-chip techniques to those used for hybrids and multichip modules. The final step is encapsulation, in which the integrated circuits and their connections are covered with “glob” of epoxy resin or plastic, which are then cured to form hard protective covers (Figure 18-31).

![Figure 18-31. Chip-on-board (COB)](image)

There are a number of variations of chip-on-board. For example, the designer may wish to maintain an extremely low profile for applications such as intelligent credit cards. One way to achieve this is to form cavities in the board into which the integrated circuits are inserted. Compared to surface mount technology, and especially to through-hole technology, chip-on-board

---

29 Hybrids and multichip modules are introduced in Chapters 19 and 20, respectively.

30 In addition to mechanical and environmental protection, the encapsulating material is also used to block out light.
offers significant reductions in size, area, and weight. Additionally, this technique boosts performance because the chips can be mounted closer together, resulting in shorter tracks and faster signals.

**Flexible Printed Circuits (FPCs)**

Last, but not least, are *flexible printed circuits* (FPCs), often abbreviated to *flex*, in which patterns of conducting tracks are printed onto flexible materials. Surprisingly, flexible circuits are not a recent innovation: they can trace their ancestry back to 1904 when conductive inks were printed on linen paper. However, modern flexible circuits are made predominantly from organic materials such as *polyesters* and *polyimides*. These base layers can be thinner than a human hair, yet still withstand temperatures up to approximately 700°C without decomposing.

There are many variants of flexible circuits, not the least being *flexing*, or *dynamic flex*, and *non-flexing*, or *static flex*. Dynamic flex is used in applications that are required to undergo constant flexing such as ribbon cables in printers, while static flex can be manipulated into permanent three-dimensional shapes for applications such as calculators and high-tech cameras requiring efficient use of volume and not just area (Figure 18-32).
As well as single-sided flex, there are also double-sided and multilayer variants. Additionally, unpackaged integrated circuits can be mounted directly onto the surface of the flexible circuits in a similar manner to chip-on-board discussed above. However, in this case, the process is referred to as chip-on-flex (COF).

A common manifestation of flex technology is found in hybrid constructions known as rigid flex, which combine standard rigid circuit boards with flexible printed circuits (Figure 18-33).

In this example, the flexible printed circuit linking two standard (rigid) boards eliminates the need for connectors on the boards (which would have to be linked by cables), thereby reducing the component count, weight, and susceptibility to vibration of the circuit, and greatly increasing its reliability.

While the use of flexible circuits is relatively low, it is beginning to increase for a number of reasons. These include the ongoing development of miniaturized, lightweight, portable electronic systems such as cellular phones, and the maturing of surface mount technology, which has been described as the ideal packaging technology for flexible circuits. Additionally, flexible circuits are amenable to being produced in the form of a continuous roll, which can offer significant manufacturing advantages for large production runs.