Chapter 2
Practical Matters: Circuit Boards and Debugging

Background and Objectives

This chapter is a bit of a detour. We will review the basic information that one needs to design and understand circuit boards. The reason we are doing this is so that when we get into concepts like transmission lines and ground bounce, we can use examples from the world of circuit boards.

When this chapter is finished, you should be able to:

• Understand basic packages for components.
• Understand the basic parts of circuit boards.
• Understand how circuit boards are designed and manufactured.
• Have a toolkit of basic strategies for debugging boards.

Components and Component Packages

Before we discuss circuit boards, one must understand the components that go on them.

Circuit components come in a variety of package styles. Components include not only passive components, like resistors, capacitors, inductors, and diodes, but also integrated circuits and transistors. Passive components typically have two terminals or wires, and integrated circuits can have dozens or even hundreds of connections.

Sometimes the component has little legs (or stiff wires) that stick through the board. The component’s legs sit in plated through holes (discussed below) and are then soldered from below. This type of part is called a through-hole component (Fig. 2.1).

The pins or wires add inductance to the signals, and often parts have to be placed by hand. (Machines can easily bend the pins.)
Sometimes the part is basically flat and just sits on the top of the board. This is called a surface mount part (Fig. 2.2).

These parts are small and place on a circuit board by machine. The smaller packages are less expensive, and the smaller size leads to lower inductance.

*Engineer’s Notebook: More About Component Packaging*

The simplest packages are the leaded packages used for resistors and capacitors. In the case of resistors, these are the brown resistors with colored bands that most students use in the first electrical engineering lab. They are called “leaded” because they have leads. (Note that this is the term “lead” with a long e, meaning a wire that sticks out, and not lead with a short e, meaning the toxic metal.) Capacitors can
have leads sticking out both ends (axial) and both leads sticking out of one end (radial).

Chips that are leaded come in packages with pins that stick out of the bottom. The simplest is the dual-inline package or DIP, which is the package that is used in most introductory digital labs. There are other through-hole chip packages, but they are increasingly rare.

Most connectors come in through-hole packages, so that the mechanical strain of inserting and removing the mating connector is transferred to the circuit board.

Surface-mount packages, as mentioned above, are designed to sit on top of a circuit board.

The simplest are the two-terminal packages for resistors and capacitors. They look like little shoeboxes with metal on both ends. The package sizes are called out in hundredths of an inch, and so a 1206 package is 0.12 inches by 0.06 inches. Common sizes today are 0805, 0402, and 0201. (0402 is considered the smallest that can be soldered by hand, and even then only with a very steady hand.) 01005 is on the horizon, which is 0.01 inches by 0.005 inches.

Chips come in packages with leads that stick down and then out. The result is a set of pads that lie horizontally, with the number of pads equal to the number of circuit connections that are needed. They range from 3-pin packages, such as an SOT-23, up to hundreds of pins, so-called quad flat packs or QFPs.

In newer versions, there are no leads that stick out; rather the package is arranged so that there is a tiny square of metal next to each integrated-circuit pad. These are sometimes called chip-scale packages and are very, very small. (Most cannot be soldered by hand.)

Chips with more connections come in packages with an entire array of “dots” on the bottom. The dots look like little balls, and so the package is called a ball-grid array or BGA.

Smaller packages transmit heat better, allow more circuits to be crammed into a tinier space, and are more difficult to manufacture and rework. (Some rework is always necessary because the assembly machines are not perfect.) There is no clear correlation of package size to price – it really depends on the manufacturer’s volume – but it is generally the case that more pins or balls equals more cost.

What Is a Circuit Board?

We have all seen circuit boards, normally when something breaks. It is the board, usually green, that holds all the circuits that make a device, like a computer or smartphone, work. These boards are always designed by engineers, and, as an electrical or computer engineer, you will need to know how they work to design them yourself.

The board that holds all the components is called a “circuit board” or “printed circuit board” or “PCB” or “printed wiring board” or “PWB.” We will just call it a circuit board. It is made of an insulating material, for obvious reasons. The board
holds wiring, both in the form of little wires (also called “traces”) and in the form of planes or sheets of conductive material. One main purpose of the board is to hold all of the wiring needed to connect all of the components. Rather than doing all the wiring by hand (which was done at one point; it was called “wire wrap”), a circuit board is fabricated in an optical-etching process that makes all the wires at the same time. It is much more repeatable, much faster, and much more compact than wiring by hand and results in better (i.e., better signal integrity) wiring. (The other main purpose of the board is structural – it provides mechanical strength and rigidity so that, e.g., plugging in a connector does not break the connector.)

On a typical board today, the insulating material is FR-4 fiberglass, and the wires are made of copper. Silver is slightly more conductive, but much more expensive, so copper is used instead.

In trying to get all the wires from all of the “point As” to all of the “point Bs,” there isn’t enough room on a single layer to connect all the wires. Almost all boards therefore have at least two wiring layers. Connecting wiring between different layers is vertical structures called vias. Picturing the board lying flat on a table, a via is a wiring connection that runs vertically from the top of the board to the bottom.

Power and ground can carry a lot of current, and so most modern boards carry power and ground on sheets of conductors called, simply enough, the power plane and the ground plane.

On a typical board, the top and bottom layer (again, picture the board lying flat on a table) are used for signals, the second layer for the ground plane, and the third layer for power. This is a four-layer board and is found in most consumer electronics; it is a good compromise between cheap and functional. Cheaper boards are two layers (or even one layer), and more expensive boards are six or eight layers. (Even though a six- or eight-layer board costs more, it can hold more wiring and therefore leads to a more compact design, so there is a trade-off.)

If there are more than four layers, the internal layers lie between the power and ground plane, and this leads to an important distinction. Signals on the top and bottom layer have air on one side and the circuit board on the other. This type of routing is called a microstrip. Signals in the middle are completely surrounded by circuit-board material and lie between two planes. This type of routing is called a stripline. In general, stripline has more capacitance (the signal is surrounded by circuit-board material, which has more capacitance per unit volume than air) but is almost completely shielded from the outside world.

A typical stack-up of signals for a circuit board is shown below (Fig. 2.3).

Some of the dimensions of the circuit board are constrained by the company that fabricates the board. For example, the signal traces must have some minimum width and some minimum spacing. (Today a typical limit is 5-mil-wide traces with 5-mil spacing. 1 mil is 0.001 inches or 0.0254 mm.) The thickness of the insulator is specified so that the signal lines have a known characteristic impedance (which we will study later). Specifying the insulator and conductor thickness is called a layer stack.
How Are Circuit Boards Made?

The process starts with a sheet of fiberglass coated on both sides with copper. The thickness of copper is specified by the designer of the board, and there are a few standard thicknesses that are commonly used. Perhaps the most common thickness (and the one that will be used if you don’t specify anything) is “1-ounce copper” which is 1 ounce (weight) of copper per square foot of board. In SI units, it is a copper layer that is 35 μm thick. For digital signals, the thickness is surprisingly irrelevant due to the skin effect (more on which later).

The copper is covered in a chemical and exposed to light. The chemical is a material called photoresist that becomes soluble in acid when exposed to light. So if you place a mask in front of the light, the light-dark patterns on the board cause some of the photoresist to become soluble. (In other words, light shines through the mask and creates a shadow on the top of the board.) Specifically, the mask is dark in the same pattern as the wiring on the layer. So the photoresist becomes soluble every place where light hits it, and not in places that are in shadow. The process is repeated on the other side of the sheet. (Recall that both sides are covered in copper.)

The structure with exposed photoresist is then dipped in acid. The acid dissolves where the photoresist was exposed to light and then keeps going and dissolves the copper underneath. Where the photoresist was not exposed, the acid does not etch through and the copper remains.

The result is a two-layer board with wiring patterned on both sides. Each wire on a circuit board is called a trace.

To make a four-layer board, the company fabricates two two-layer boards and then presses them together. Specifically, a piece of fiberglass impregnated with glue (called pre-preg) is placed between the two two-layer boards, and the stack is put under pressure and heated (or laminated). Think of it as a heated sandwich with pre-preg as the meat and exposed, etched two-layer boards as the bread.

![Typical layer stack-up](image-url)
If the board has more than four layers, the process is the same except that the stack is higher and there are more pre-preg layers. It should be clear now that almost all boards have an even number of layers.

What happens next is a little quaint. The vertical structures (specifically the vias) are made by drilling holes (one hole per via). Another round of photoresist is applied and etched, and this time copper is added. Copper is added where there is no photoresist, and so all of the drilled holes become filled with copper.

This is important to understand. If the ground plane touches the part that gets drilled, the via will be shorted to the ground plane. Sometimes this is desirable; for example, this is how you get a ground connection up to a circuit component like a resistor. If the connection to ground (and/or power) is undesirable, then there has to be a hole in the ground and power planes so that, even after the drill bit makes a new hole, the power and ground plane do not touch the resulting via.

Sometimes the copper-filled hole is placed so that the lead of a circuit component can fit through it. This structure is called a plated through hole. A plated through hole is a lot like a via and can be used to route signals from top layer to bottom, connect to power or ground plane, etc. (It is also unlike a via in the sense that it is not completely full of copper.)

The result of all of this is a complete, three-dimensional wiring structure with wires and power and ground planes, ready to hold components. Some of the components lie flat on top of the board and are called surface-mount components. Others have pins that stick through the plated through holes (and are soldered on bottom) and are called through-hole components. The place where the component is located is called the component’s footprint.

Three more layers are typically also added.

First, the structure is covered in a coating (usually green) called soldermask. This coating keeps the solder from sticking to parts of the board where you don’t want it. Stated differently, it only leaves copper exposed in places where you want solder to touch it. (The green color we associate with circuit boards is actually the color of the soldermask, not the color of the fiberglass.) The soldermask has some signal-integrity consequences. It adds a layer of insulator to the top of most conductors and increases the dielectric constant (i.e., capacitance) in subtle ways.

Second, a layer of markings is added in order to make the board human-readable. This is applied to the board in a silk-screen process (yes, the same process used to put designs on T-shirts) and is called silk screen or sometimes just silk. Typical silk-screen markings include labels to identify components (called reference designators or ref des’s), markings to indicate part orientation (e.g., diodes), and sometimes the name of the company or warning labels. These markings are essential; it is the only way the factory knows which part is which and which orientation is correct. It is also the only way a repair technician knows where to look and the only way a design engineer can check the design.

Third, solder is sometimes placed where components will later be soldered. When a board has solder added, it is said to be tinned. You can recognize the solder because it looks like a (and is) shiny metal (Fig. 2.4).
How Are Circuit Boards Used to Make a Product?

Since a surface mount is usually first in the process, we will start by describing a surface mount.

The circuit board is the focal point of almost every electronic design because it contains all of the wiring and holds all of the components. So what happens next is that a worker at a factory have to put every single component on the board correctly, or else the system won’t work and your company won’t have any sales.

The way that it is done is with machines called *pick-and-place machines*. These machines are loaded with reels of components. If you designed a board with a 1N914 silicon diode, a reel of 1N914 diodes is loaded onto a slot on a machine. Every time a new board enters the machine, a vacuum head reaches over and sucks up a single diode, pulls it off the reel, goes over to exactly the right place on the board, flips the diode into the correct orientation, and then “poof” shoots it onto the board. In a cable set-top box, for example, this process is repeated for about 2000 components. Fortunately this process is a lot faster than it sounds. Also, the machine usually has several vacuum heads.

What makes the process amazing is when you stop to think what can go wrong. The part can land in the wrong place. It can be in the wrong orientation. It can be the wrong part (the wrong reel or the wrong slot). Even if everything is correct, the machine can make a mistake, and the part is not lined up correctly when it lands. Making life more difficult, these parts are often incredibly small. For example, a typical component size is “0402” meaning that it is .04 inches by .02 inches (1 mm by 0.5 mm) in size.

Once all the parts have been placed, the entire board passes through a *reflow oven* which looks like a pizza oven. The board rolls through on a conveyor and
different zones in the oven are different temperatures. The goal is to slowly heat the board to just below the point where solder melts, rapidly take it above and below the melting temperature, and then slowly cool it back to human-touchable temperature. This melts the solder and simultaneously solders all of the components onto the board at once. (Thought question: So how do you put surface-mount components on both sides of a board (Fig. 2.5)?)

The through-hole assembly processes more straightforward. The parts are inserted into the holes. Small parts, like capacitors, are inserted by machine. Complicated parts, like high-definition television connectors, are often inserted by hand. The leads are trimmed so that they only stick out a short distance below the board. Then the board is passed over a wave solder machine. The wave solder machine is basically a lake of molten solder. (In fact, the solder looks solid until you whack the side of the machine with your hand, and it undulates.) The board barely touches the solder (which is actually made to flow over a bump that rises out of the lake of solder). Every lead that sticks out sucks the solder up through capillary action, and they all get soldered (Fig. 2.6).

So what if you need to add a through-hole part to a surface-mount board? The solder wave would suck all of the soldered parts back off the board, with tragic consequences. So a board is laid inside of a selective-solder pallet which only has holes where there are through-hole parts. This leads to an important but obscure design rule – through-hole components must be kept a minimum distance from bottom-side surface mount parts so that there is room for the selective-solder-pallet hole (Fig. 2.7).
Fig. 2.6  Through-hole board. This particular board is so inexpensive that it has no traces on the top side. The bare wire JP6 is being used to jump over a trace on the bottom side. (Photograph obtained from Adobe Stock. Used with permission)

Fig. 2.7  Board containing both surface-mount and through-hole parts. Note how Capacitor C31 and chip U4 have no components. This is called a population option – this particular version of the board did not need C31 or U4, and it was built without them to save money. (Photograph obtained from Adobe Stock. Used with permission)
Backup: How Did We Get Here?

The process leads to an important question – how did the board get designed in the first place? This is the trick and in fact is more or less the subject of the rest of the book.

The entire process is important to understand. You have to understand how boards are made and how electronic devices are assembled to design them properly. You also have to understand that good signal integrity, which is the product of good design, lies in the middle of a complete ecosystem of manufacturability. In other words, your design has to have good signal integrity and be capable of being mass-produced.

One enemy of good signal integrity is the forest of details that have to be handled correctly just to get the board out of the factory. It is easy to get wrapped up in the details and lose sight of signal integrity. My own design team made that mistake numerous times.

The solution is to understand the forest of details well so that it becomes manageable and thereby free up enough time to sit down and do good design.

So what are the details?

How Are Circuit Boards Designed?

Circuit-board design starts with a designer deciding which parts to use. Actually, it starts with a designer being told to “go design something” based on her company’s needs (more on which later), but the designer starts by choosing parts. Sometimes this is easy (I want a 4.7 kΩ resistor), and sometimes it is more complicated (do I choose a capacitor with a 100°C temperature rating or 150°C temperature rating).

Once the parts are selected, the designer creates a schematic which is a diagram that shows all the parts and how they are connected. The schematic, for example, tells where the 4.7 kΩ resistor is connected into the rest of the circuit. The schematic winds up being a very important design document. It specifies the topology (interconnectedness) of the design and calls out information like what value the resistor is supposed to have. The schematic also contains reference designators, which match the markings on the circuit board. In other words, it contains information that lets you map the physical board in hand to the schematic.

Many companies have part-numbering systems so that a 4.7-kΩ surface-mount resistor might have a unique part number. If so, this information is also on the schematic.

Once the schematic is complete, if the designer is smart, then a design review is held, and several people go over the schematic to look for mistakes. Like every other step of the process, creating a schematic is error-prone, and so this step acknowledges this fact and brings in other people to double-check the design. For
example, it is very easy to get the power and ground wiring wrong, or even to forget it entirely.

From the schematic, a layout is created. This is the step that creates the physical layout. For each part in the schematic, there is a footprint, a layout of the part as it sits on the board. If a resistor is surface mount, for example, the footprint is two rectangles of solder because that is what sits on the board. If the resistor is through hole (contains wires on both sides), the footprint is two holes in the board. Since the schematic and layout are normally created in the same software package, the design tool normally has a way to map each part in the schematic to a footprint. There are subtleties in the footprint. For example, if the part is a diode, there needs to be some way to indicate the polarity of the diode. Finally, the layout is where the designer expresses information like which layer is used for what and where holes need to be cut into layers for vias.

Like the schematic, when the layout is finished, there is another design review. Note that layout is what typically creates signal-integrity issues. The layout governs how signals are routed (laid out on the board), where the planes are, where components are, the size and shape of the circuit board, etc. Mistakes in any of these can create signal-integrity issues, and that is what we will spend most of this book studying.

After layout, the circuit board is ready to order, and so the designer creates the files needed for the circuit-board manufacturer to create the circuit board. The factory will also need the files, so they will know which part goes where, and the correct orientation. These files are a major design milestone, as everything downstream of the creation of the files will use them to assemble and test the boards.

It is important to understand that the transition from layout to schematic to board is usually under a lot of time pressure. The rest of the company needs these boards (e.g., software engineers need actual units in order to write the software). This is where shortcuts get taken and signal-integrity issues are overlooked. The more familiar you are with signal-integrity issues, the more likely you are to do it right the first time, which is why (ultimately) you are reading this book.

The Board Is Dead. Now What?

Understanding the process by which boards are designed, made, and assembled into working products helps you debug boards that don’t work. When you get the first articles back from the manufacturing facility, there is an uncomfortable few days (or, alas, weeks) as the very first boards are brought up. This can be tedious or terrifying, depending on how well you did your design.

The discussion begins with a board that seems completely dead.

The first thing to keep in mind is “what is the likeliest thing to go wrong.” For example, it is easy to load a pick-and-place machine with the wrong component or for the factory to leave out an important part altogether. A quick visual check of the parts will help.
It is easy to make mistakes connecting power and ground to parts, so always check power and ground first. This is a logical extension of the absolutely true maxim to make sure it is plugged in first. The author of this book has skipped this step on more than one occasion and wasted many hours relearning this painful lesson.

After checking power supplies and voltages, be sure to check the reset circuit and the clocks. These also have the ability to kill the entire board.

Assuming the board is mostly working, the next step is to go through each sub-circuit and test them. At this point the work can be split up among multiple engineers and typically goes a lot faster.

Besides the common sense that goes into this step, there are a couple of “tricks” that will help a lot. First, be sure to hold the oscilloscope probe correctly. If you use the ground clip, you will get ringing because of the inductance of the ground connection. (This is discussed in more detail later.) If you use the ground clip, it is OK as long as you know the effect it will have on the measurement. Second, write down every issue you encounter. At some point the board may have to be redesigned, and, if so, it will be very good to have a detailed checklist of every change that needs to be made.

Finally, an observation... Every time you go through this make mental notes and then bring them up to management later. How could you have prevented the mistakes that get made? What mistakes are made often? What could you do to make it less likely for other groups to drop the ball? This sort of thinking improves the process you work under, improves your life, and starts making you promotable.

Homework

1. Consult online sources and find the layer stack for a four-layer circuit board (copper layers on top and bottom, power and ground planes in the middle) that is 63 mils thick. (This is a standard four-layer circuit board.)
2. What does 1-ounce copper mean?
3. A co-worker shows you a resistor that is a very small rectangular shape. Is it through hole or surface mount?
4. What are the steps in designing a circuit board?
5. What are the steps in manufacturing a circuit board?
6. Define the following circuit-board terms:
   (a) Trace
   (b) Plane
   (c) Via
   (d) Through hole
   (e) Surface mount
   (f) Solder mask
Signal Integrity
Applied Electromagnetics and Professional Practice
Russ, S.H.
2016, X, 181 p. 63 illus., 19 illus. in color., Hardcover
ISBN: 978-3-319-29756-9