ble so that logic mistakes do not require replacing a potentially expensive IC. Smaller, more mature PLDs may still require dedicated programming hardware, which may be reason enough to avoid them if possible in favor of a small CPLD. The price difference between a PLD and a small CPLD is now slim to none.

For reasons previously discussed, signal integrity software packages may be necessary when designing high-speed digital circuits. These tools can be quite expensive, but the consequences of not using them can be even more costly in terms of wasted materials and time if a circuit malfunctions because of signal integrity problems. Before embarking on an ambitious high-speed design, make sure that signal integrity issues are either well understood or that the resources are available for proper analysis before fabricating a prototype. Certainly, not all designs require extensive signal integrity analysis. If it is known that the signal speeds and wire lengths are such that transmission line effects, crosstalk, and EMI can be addressed through conservative design practices, minimal analysis may be required. This determination generally requires someone with prior experience to review a design and make predictions based on previous work.

Risk assessment in choosing which components and technologies to employ is an important part of systems design. An otherwise elegant architecture can fall on its face if a key component or necessary development tools are unavailable. Therefore, be sure to make choices that are practical for both the application and the resources at your disposal.

19.2 PRINTED CIRCUIT BOARDS

The selection of appropriate technologies is a convenient segue into circuit construction, because the manner in which a circuit is assembled can have a great impact on the viability of the resulting prototype or product. Higher-speed circuits are more sensitive to construction techniques because of grounding and inductance issues. Most high-speed circuits can be fabricated only with multilayer PCBs, but more options are available for slower systems, especially in the prototyping phase of a project.

Circuit boards can be of either the printed circuit or manual point-to-point wiring variety. As already discussed, PCBs consist of stacked layers of copper foil that have been uniquely etched to connect arbitrary points in the circuit. The term “printed” refers to the standard technique of using photolithography to expose a chemically treated copper foil with a negative image of the desired etching. Similar to creating a photograph, the exposure process alters the photoresistive chemicals that have been applied to the foil so that the exposed or nonexposed areas are etched away when the foil is placed into a chemical acid bath. PCBs are an ideal technology, because they can be mass produced with fine control over the accuracy of each unit. Simple single- and double-sided PCBs can be manually fabricated using a variety of techniques, and the cost of having such boards professionally manufactured is low. Multilayer PCBs must be fabricated professionally because of the complexity of creating plated vias and accurately aligning multiple layers that are etched separately and then glued together. The major cost involved in designing a small PCB is often the specialized computer aided design (CAD) software necessary to create the many features that a PCB implements, including accurate traces, pads, and IC footprints. Low-end PCB CAD packages are available for several hundred dollars. High-end tools run into the tens of thousands of dollars.

Once a PCB is fabricated, it is assembled along with the various components to which it is designed to connect. Assembly may be performed manually or at a specialized assembly firm, almost all of which use automated assembly equipment. It is difficult to manually assemble all but relatively simple boards because of the fine-pitch components and the element of human error. Automated assembly equipment substantially increases reliability and improves assembly time for multiple boards, but at the expense of increase setup time to program the machines for a specific design.

Automated assembly is performed using one or two soldering techniques, depending on the types of components on a PCB. Before the widespread use of surface mount technology (SMT), the standard process was to program a pick-and-place machine to automatically insert leaded components into the correct holes in the PCB. The machine trims the leads as they are inserted so that a small length protrudes from the bottom, or solder side, of the PCB. When the PCB is fully stuffed, it is placed onto rails that carry it through a wave-soldering machine. A wave-soldering machine contains a bath of liquid solder over which the PCB is dragged. The metal component leads, in combination with the plated holes and pads of the PCB, wick molten solder up into the gaps, and the solder hardens as the PCB exits the molten solder wave. Wave soldering is a mature assembly technology that has worked well for decades.

Certain design rules are necessary for successfully wave soldering a PCB and preventing excessive solder from shorting adjacent leads. First, a PCB's top and bottom surfaces are finished with a thin layer of solder mask so that only metal intended for soldering is exposed. The molten solder will not adhere to the solder mask. Second, minimum lead spacing rules are followed that ensure that the lead gaps are too large for the solder to wick into them. Finally, rows of component leads are oriented in a single line perpendicular to the solder wave to minimize solder bridges, or shorts. If the leads of a DIP package are moved through the wave abreast, or in a parallel orientation, they are more prone to picking up an excessive quantity of solder in the many gaps formed between the leads. When the DIP is rotated 90° so that the lead rows are perpendicular to the wave, the solder can more freely travel down the narrow column and not get excessively stuck in one inter-lead gap. There are a variety of PCB layout tricks to reduce solder bridging in wave-soldered PCBs. If your PCB is expected to be professionally wave soldered, it is advisable to consult the assembly firm before designing the PCB. Manufacturing engineers at assembly firms are very knowledgeable about what works and what doesn't, because they work with this equipment on a daily basis. Each assembly firm has its own set of manufacturability guidelines that have been developed based on the tools at their disposal. Rules for one firm may not be fully in agreement with those of another. A half-hour conversation ahead of time can save days of headaches caused by myriad short circuits in an improperly designed PCB.

Wave soldering does not work for many SMT boards, because the small gaps in fine-pitch component leads act as efficient solder sponges, resulting in completely shorted boards. SMT PCBs use a solder paste reflow assembly process. The process begins by applying solder paste to the SMT pads through a special stencil. The stencil has openings in the exact locations where solder paste is desired. A pick-and-place machine then stuffs the PCB. Careful handling of the stuffed board is necessary, because adhesion of the solder paste is all that holds the parts in place. Many PCBs have components on both sides. In these situations, the bottom-side components are held in place with the solder paste's natural adhesion, which prevents them from falling off when they are upside down during the remainder of the assembly process. Once all SMT components have been loaded onto the PCB, the PCB is moved through a reflow oven with carefully controlled temperature zones to suit the solder, component, and PCB properties.

A PCB with both through-hole and SMT components usually requires a two-step process consisting of reflow followed by selective wave soldering. Selective wave soldering is accomplished with varying types of masks that are temporarily applied over the bottom-side SMT components so that they do not get shorted or stripped off the board in the solder wave. It is preferable to design a PCB without the need for a two-step process to save both time and money in assembly. An SMT PCB with only a few necessary through-hole components (e.g., connectors) may be more efficiently assembled by having a person manually solder the through-hole components after reflow.

As with wave soldering, the reflow process has its own set of manufacturing guidelines to increase the reliability of the manufacturing process. The intended assembly firm should be contacted before PCB design to fully understand its engineers' manufacturing rules. A standard rule for both wave soldering and reflow is to leave a minimum spacing, or *keep-out*, between the PCB edge and