Printed Circuit Board Design/Prototyping Basics

by Danny R. Graves

If you want your product or project to look its best, you should get a printed circuit board designed and built.

When you approach potential investors with your revolutionary electronic product, it helps if you have a working prototype. Even if you are not looking to make a million bucks, a great-looking prototype part will impress your hobby buddies. This prototype part could have a circuit that is hand-built on perf board or other quick, cheap, and dirty methods that may work, but don't look professional. If you want your product to look the best (and sometimes work best), you should get a printed circuit board (PCB) designed and built. Even better, you should have the PCB designed such that it can be produced on a mass scale with automated machinery without major redesign. This is just in case someone wants to buy your design and make you rich. But, how do you design a PCB and get prototypes built? I will cover the basics.

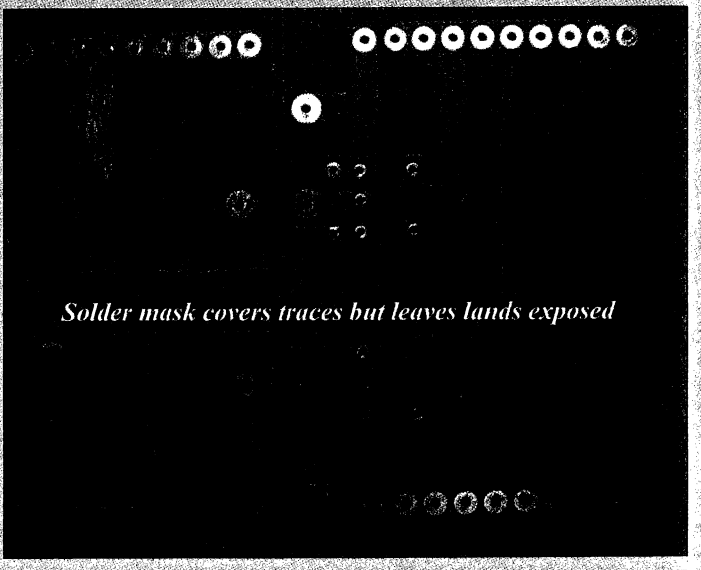

Figure 1.
An example of printed circuit board solder mask.

HOW MUCH IS PCB LAYOUT SOFTWARE AND WHAT WILL IT DO?

Some prototype printed circuit board companies will actually give you rudimentary PCB layout software. Check out **www.expresspcb.com** for an example of this. If not, several companies offer PCB layout programs for a few hundred dollars. See **www.mentala.com** for example design packages in this price range. Using this type of inexpensive software to design a circuit board can be little more than a connect-the-dots game. You simply draw the outline shape of your PCB, place the components where you want them, and connect the components together per your schematic. Although they can get the job done, inexpensive PCB layout software doesn't have some of the nice features of more elaborate design packages. Or, if they do have the features, you are limited on how complex your PCB can be.

More expensive ($5,000.00 to $50,000.00) PCB layout software allows you to enter your schematic, automatically place the components, automatically route (connect the dots), and then check your PCB layout against the schematic. These packages also have built-in design rule checkers where you can check your design against the generic rules or enter your own design rules. For instance, your contract manufacturer may not want any components closer than .060" to the edge of the PCB because the component may get impacted when the PCBs are separated. You can set up a design rule such that you get an error message when you violate this constraint. There are plenty of other design constraints that you may want to consider to make sure your PCB design is robust and can be manufactured in high volume.

IPC is an organization that publishes standards covering PCBs and PCB assemblies. Their publications detail the design standards that should generally be followed for success. You can purchase their standards from their web site (**www.ipc.org**). You may also want to contact companies such as Fuji, Panasonic, or Universal. These companies make manufacturing equipment for component placement and may offer tips to make designs more efficient to manufacture with their equipment.

PARTS OF A PCB

A PCB is made up of several components. These components are the laminate, conductive layers, lands,

holes, solder mask, and silk screen.

A PCB is made up of different layers laminated together. The laminate material is the foundation of the PCB. This can be various materials such as FR2, FR4, or CEM-1. These materials are made of such things as epoxy and fiberglass. For flexible PCBs, polyimide film laminates are used. Laminates are electrical insulator materials. The choice of substrate material greatly affects the cost of an individual PCB.

The solder mask is the material that is placed over the areas to which you do not want solder to adhere. There are a variety of different types of solder masks and processes by which to apply them. Screening the solder mask is an inexpensive but inexact way to apply the mask material. Liquid photo imageable (LPI) is a slightly more expensive, but more precise way to apply the mask. Solder mask is the stuff that is usually all over the surface of the PCB except the component pads. Figure 1 shows an example of solder mask.

A silk screen is a printing, in ink, of characters or graphics on the surface of the PCB. The manufacturing department may require a silk screen to show the location and reference designator of the components for inspection and re-work purposes. It's possible to print characters in copper, but it's often advantageous to use a non-conductive method of printing, especially in high-density designs. Figure 2 shows an example of silk screen.

Copper is what makes the electrical connections on the PCB. The strips of copper connecting components are called traces or tracks. Figure 3 shows an example of a trace. A layer of the PCB that contains traces or a continuous sheet of copper (like a ground plane) is called a conductive layer. A PCB can have one or more conductive layers. When you hear someone say they have a four-layer board, it means they have a PCB with four conductive layers. An odd number of conductive layers are generally not used because of warping effects on the PCB.

There are holes in most PCBs for mechanical mounting, mounting and soldering components, and interconnecting the conductive layers. A land is the area around a hole to which a component is soldered. Figure 3 shows an example of a land. Via holes are what connect the different conductive layers together. Vias are conductively plated on their interior surface and traces on the appropriate layers

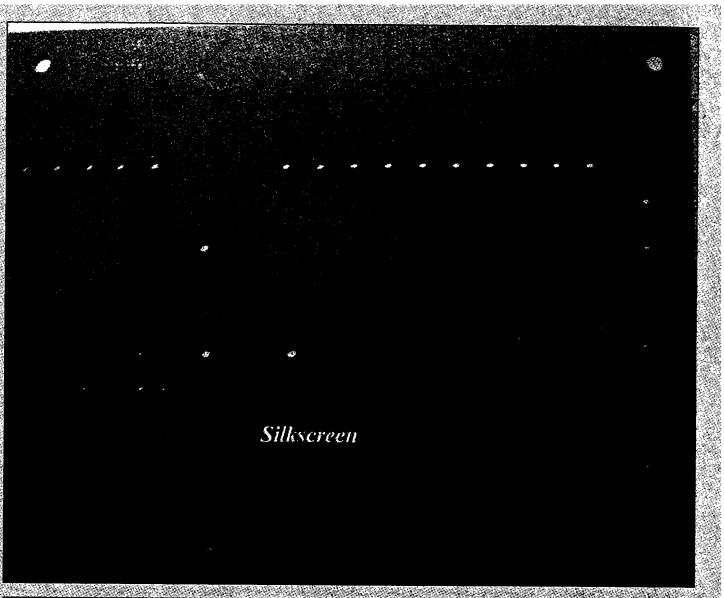

Figure 2.
An example of silk-screening on a printed circuit board.

are connected to the vias. Vias can be all the way through a PCB or they can be buried. "Buried" means that the via interconnects internal layers and cannot be seen from the exterior of the PCB. A higher density of conductors is possible with buried via holes since some layers do not have space taken up by the via. Figure 3 shows an example of a land.

DESIGN CONSIDERATIONS

Besides just making your circuit function, PCBs also dictate how easy it will be to manufacture your design. Designing it as manufacturing-friendly as possible up front will make your design easy to sell later on, as well as just making it easier to make one sample for yourself.

Even if your circuit is perfectly engineered on paper, theoretically works like a dream, and even works on a bread board, the PCB can screw up those plans. This is because the PCB has factors such as trace impedance that can affect the voltage levels at various points in your circuit. Thinner traces have higher resistance. If you are working with a few millivolts or milliamps in analog circuits, the size (length, thickness, and width) of the

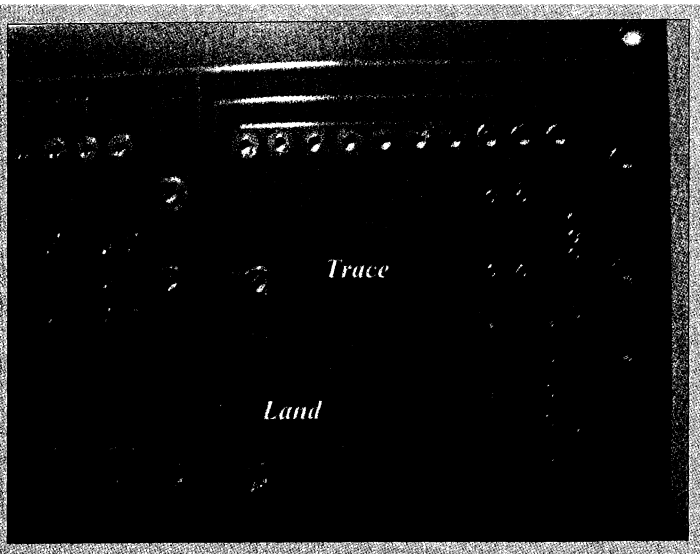

Figure 3.
An example of a land and a conductive trace on a PCB.

traces is a big deal.

You will also have to decide what type of component packages you will need to use. If they are standard component packages, you will be able to pull them from a menu on your design software, and pop them on the PCB. If they are not standard packages, you will have to draw each different type of package. You will need to determine if you are going to use all through-hole components ("old fashioned" components with wire leads), all surface mount technology (SMT) components (new fangled components without leads), or a mixture of the two types. There are other types of component packages such as ball grid array (BGA), which rest on a multitude of solder balls that get re-flowed in an oven, and chip-on-board (COB) devices, which are bare semiconductor die without a plastic package, but they are beyond the scope of easy prototyping. Heck, it is even hard to build and troubleshoot prototypes with fairly large SMT devices. But SMT sure looks high tech! Your easiest bet for prototype building and troubleshooting is to use through-hole package components. However, keep in mind that your circuit might work differently if you change to SMT, BGA, or COB later on. This is especially true if it involves radio frequencies.

PCB layout is important in circuits that operate at very high frequencies (VHF) and beyond because it can reduce or increase the amount of stray inductance and capacitance that is present. This stray inductance and capacitance makes up a significant part of the total inductance and capacitance in the circuit at these frequencies. Wide and short PCB traces will help reduce the stray inductance. It is also good practice to use a ground plane (a layer of your PCB that is entirely at ground potential) instead of having the ground connection fragmented about the various layers.

Even if your circuit does not operate at VHF and beyond, it may be sensitive to interference from transmitters operating at those type of frequencies. A 100-MHz signal can disrupt the functioning of a circuit operating at a few Hz. If your circuit ever has to go through electromagnetic compatibility (EMC) testing for applications such as automotive or aerospace, a proper layout will be critical. EMC testing subjects even low-frequency devices to varying frequency ranges to make sure they function properly and are not damaged.

SHOULD MY PCB BE INDIVIDUAL OR IN A PANEL?

For manufacturing efficiency, it is desirable to design a panel or matrix of PCBs. This is so several PCBs can be sent through the soldering or re-flow process at once. After soldering or re-flowing, the PCBs are then separated by cutting, pressing, or whatever process the PCB assembly manufacturer desires.

Panels inherently waste some PCB material because of the internal and external borders. To make sure the panel waste is minimized, the designer should work with the PCB supplier to figure out the optimum panel design. The panel design can be completed in the PCB layout software or the panel can be designed in separate CAD software and the actual PCB artwork stepped and repeated by the PCB supplier.

For cost-efficient prototypes, you will probably not want to design a panel, but just design a single PCB. Even though prototype PCBs may not require panel design, knowledge of panelization will impress anybody with which you discuss PCBs.

THE PCB IS DESIGNED ... NOW WHAT?

Once you get the PCB design finished, you will have to get the design to your PCB supplier, in the fashion that they require, in order to get your prototype. The format that PCB suppliers want is Gerber for the copper artwork and Excellon for the hole locations and sizes.

You must provide a Gerber file for each layer in your design. There will be a Gerber file for each conductor layer, the silk screen layer, the solder mask layers, etc. The PCB design software will usually give different file extensions for each layer's Gerber file.

An aperture list is a file that dictates the size of a specific entity on the artwork. For instance, the aperture list might call out a .060' round shape that is used to form a component solder pad (land). RS274X format Gerber files don't have to have a separate aperture list file since they are built into the Gerber files for each layer. However, RS274D format files do require a separate aperture list that must be provided to the PCB manufacturer with the Gerber files for each layer. There should be one comprehensive aperture file that covers all the different layer Gerber files. Most modern PCB layout software packages output RS274X format Gerber files.

There will be a couple of Excellon files. One will call out the software codes for the drill bit sizes for all the different hole sizes in your design. The other will call out the X-Y location of each hole and the corresponding drill bit code. The X-Y locations are referenced to an origin somewhere on your PCB (probably one of the corners).

All the files can be combined in a zip file to make

transfer to the PCB supplier easier. However, I have seen the zipping process corrupt some of the Gerber files. Therefore, you may want to zip, unzip, and view the files to make sure you get want you think. When you view the Gerber files, you are seeing exactly what you will get. The Gerber files can be viewed in various software programs such as GCPrevue. GCPrevue can be downloaded free from **www.graphicode.com**.

WHAT AFFECTS COST?

The PCB substrate material can be FR4, CEM-1, CEM-3, etc. The fiberglass materials such as FR4 typically cost more than the epoxy type materials such as CEM-1. The materials vary greatly with temperature characteristics and other physical properties such as stiffness. The application of your design may dictate what type of substrate material is used. Not all materials will be available from all PCB suppliers.

The thicker the copper specified, the more expensive the PCBs. Copper thickness is specified in ounces/square feet. For instance, three-ounce copper is thicker than two-ounce copper and will cost more. Your choice of copper thickness may also affect other cost-impacting aspects of your board such as a solder-mask type. The thickness of the copper also dictates how much current a given copper trace can handle. For example, a .020" wide trace can handle more current if it is on a PCB using three-ounce copper than if it is on a PCB which uses two-ounce copper.

The more layers your design uses, the higher the cost. Every layer you add may add 30% to the cost of each PCB. The more complicated your circuit and the smaller your PCB length and width, the more layers you will need. A silk screen layer gets graphics such as a part number, reference designators, or notes printed on the surface of the PCB. If you specify a silk screen, the cost will be slightly higher. This is true for both prototypes and production PCBs.

If you want an electrical test before the PCBs are shipped, there will be a substantial charge. An electrical test verifies that all the connections are present. You can specify that this test not be done and save substantial money. However, you are taking a risk. If your design is simple enough, it may be worth the risk. Some PCB suppliers will do the test even if you don't pay for it.

HOW MUCH MONEY AND HOW MUCH TIME?

If you want your PCBs in 24 hours and you are willing to pay, it can happen. For instance, if you want 10 10" x 10", FR4, single layer, two-ounce copper PCBs in 24 hours, you may pay $2,000.00. If you can wait two weeks for the same PCBs, you may only pay $800.00. With this type of order, there may also be a tooling charge of $250.00. However, if you only need a few PCBs, there are cheaper options.

There are a few PCB suppliers that offer PCBs for $30.00 to $50.00 each with no tooling charges. Check out **www.PCBexpress.com** and **www.4PCB.com** for these type prices. The lead-time on these type of offers is usually one week. The number of layers is usually limited to two and the copper is one ounce. There is also a limit on the size of the PCB. The suppliers are usually strict on the constraints on these types of bargain PCBs. Search for "PCBs" on the internet to find more of these kinds of offers. As stated before, requiring an electrical test to make sure all the connections are there will substantially increase the cost, but may be worth it if your design is complex.

CONCLUSION

Using a custom-designed printed circuit board is a professional way to present your electrical or electronic product to potential investors or just make your home project neat, easy to repair, and very cool to show your friends. With minimal effort, you can get the layout software and Gerber viewer software for free. You can also get a few prototype PCBs for under $100.00 if your design fits the constraints of certain PCB suppliers. **NV**