PCB Prototype Success Is In the Cards

Jon Titus, Senior Technical Editor

Fabricators can quickly deliver high-quality PCB prototypes. To ensure success you must play your cards right.

Many engineers routinely create printed-circuit board (PCB) prototypes to test circuits or make short production runs. Large and small companies specialize in the fabrication of small quantities of PCBs, and engineers can submit design files over the Internet and get a price quote in return. Some of these companies offer turn-around from a day to a few days.

To get started, Nilesh Parate, general manager of PCB Fab Express recommends engineers use the latest software for schematic capture, autorouting, and production of the files a PCB fabricator needs. “Engineers can use Protel, OrCAD, Eagle, or another recognized package,” said Parate. “PCB-design software creates Gerber files, but not all Gerber files are the same. If you use professional software and some free software to produce the same PCB layout, the resulting PCBs may not turn out exactly the same. The quality of the software affects the quality of the PCB. Just using the latest revision of your installed software can solve a lot of problems.” Likewise, buyers should ensure their PCB fabricator runs the latest CAD software in its facilities. (For more information about Gerber files, see, For further reading.)

Before you start to lay out a board, always consult PCB fabricators’ Web sites for “trace-and-space” design rules. Becky Martin, production manager at Avanti Circuits noted many Web sites provide a frequently-asked-questions (FAQs) section that answers basic design questions and explain design rules. “When a customer wants

a certain PCB thickness or material, a Web site FAQ or a list of capabilities usually provides that information," explained Martin. In many cases, customers can use a list of FAQs as a design checklist.
After engineers create a design, they must run a design-rules check (DRC) to ensure their layout meets their vendor's specifications. Vendors also will run a DRC and a manufacturing-rules check (MRC) to pick up any problems before they try to produce a PCB. The design rules arise from the vendor's specifications found on its Web site or in its published literature. Vendors also have rules about proximity of traces to a board's edges, hole sizes, and other criteria. According to Jim Gray, general manager at Quick Turn Circuits, engineers often forget to follow trace-widths and -spacing requirements. "The DRC gets engineers started. The MRC lets us make sure we can produce a customer's board," said Gray.

In addition to supplying PCB layout files, engineers must provide a Gerber or read-me file that includes an artwork "view" statement. According to Bill Parlin, U.S. sales director at Crimp Circuits, that information indicates how the PCB vendor will view the PCB. "We receive files with views of all kinds, some from the top side, some from the bottom, some mirrored, some mixed, and some offset. So, clearly state the view. Engineers should provide a layer-stackup sequence chart that specifies layer names and the order in which the engineers need the layers stacked in the PCB." Although manufacturers can accept files in many formats, most prefer files that adhere to the Gerber RS274x format.

"Often, engineers send us their Gerber files, but they don't give them unique names," said Jim Gray of Quick Turn Circuits. "A double-sided board has a component side, a solder side, top and bottom sides, a solder mask, and legend silk screens. But a lot of people give the files generic names, such as G01, G02, and so on, but no notes as to what those files refer to." Gray noted customers can specify colors of masks, starting or finished copper weights and material thickness, but about 50 percent of his customers don't state what they want.

Parlin reminds engineers to include complete construction, fabrication, and material specs when they request a quotation. "This information reduces the need for a manufacturer to ask questions, and it increases the accuracy of the quote by eliminating incorrect assumptions." Engineers also should realize that as they tighten tolerances, costs rise. So Parlin recommends when engineers must keep costs low, they should use the most generous tolerances that a design will allow.

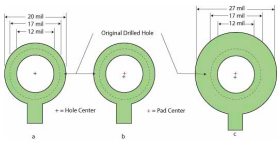

Figure 2. Circuit. Shown is the "drilling and plating" a new circuit, a large cavity, general purpose component ring. The drilled hole is the center hole. Plate the hole to make the pad. Construction of circuit.

Image courtesy Avanti Manufacturing Technologies Inc. Image by Gadget Learn to Solder. Enlargement. Photo edited by staff editors.

"We need a drill drawing, notes and plating information" said Martin at Avanti Circuits. "Typically we produce PCBs with 1 oz copper and can plate it to 2 oz. But engineers may want 3 oz copper, although they forget to tell us. Also, we need to

know what type of plating — gold, solder, lead-free tin — they want." (The "weight" of copper refers to the amount on a square foot of unetched PCB surface.)

Martin also noted that when customers need controlled-impedance traces, they should note that spec in their files or notes. The spacing between layers affects impedance, so Avanti can adjust line widths and spacing to ensure PCB traces have the correct impedance.

Top PCB Tips

A few examples can help guide engineers to success when they need prototype boards.

1. Check the hole and pad diameters of plated-through vias. If you need a 12-mil-diameter plated-through hole, for example, a PCB vendor will likely drill a 17-mil hole and the internal plating will reduce the hole to 12 mils. But drills do not always hit dead center. To ensure you still have sufficient pad area at the top and bottom of the hole, make the pad diameter 10 mils bigger than the drill size. If tight spacing prevents beefing up the pad diameter, try to use a smaller hole. In all cases, you want a large enough annular ring of copper on both sides of the board to ensure good conductivity through the via.
2. Martin at Avanti Circuits recommends making diameters of openings in a solder mask
   5 mils larger than their associated pads. The larger opening helps ensure the solder mask doesn't encroach on the electrical pad.
3. Whenever possible, set your PCB layout software to "flash" pads rather than draw them. A flashed pad exists as a single geometric entity while a drawn pad comprises many lines used to fill an area. The latter markedly increases the size of Gerber files and can cause production problems.
4. Carefully choose the type of plating you need. "Recently we had a request for a board with hot-air solder-leveled (HASL) solder," said Parlin of Crimp Circuits. "The customer planned to put BGAs on a board, but the BGA solder balls would slide off the curved surface of the HASL solder pads and cause a production problem." In this case, Parlin recommended an immersion-plated finish that would produce smooth, flat pads. "An inexpensive HASL solder coat, with its "bumpy" domed-pad topography, might result in a higher project cost because of assembly challenges."
5. As trace-and-space dimensions decrease, so does the maximum thickness of copper. "Learn about industry standards," said Parlin, "And remember PCB manufacturers often have different capabilities. For best economy, try to design a PCB without exceeding normal limits."
6. Give PCB vendors complete "contact" information. "Sometimes people send us files by email and want a quick response, but they forget to give us a phone number," noted Jim Gray at Quick Turn. "We can't give them a call to ask for more information or phone with a quote."
7. "Always include an accurate board outline on at least one layer," recommended Parlin. "Also clearly show any slots or cutouts on this layer. Vendors can use this information to create routing and scoring programs,

which saves time and money."

8. Use a minimum 5-mil line width for legends and lines used for component outlines. "We get legends with 2-mil-thick lines," said Martin of Avanti Circuits. "When we try to increase the line widths, we end up with illegible text."
9. Martin tells designers to keep traces and pads at least 20 mils from their PCB edges. Board-dimension tolerances of ±5 mils are common, and you don't want traces cut or damaged during production. Martin said Avanti will look at the PCB traces and pads and if possible try to move them slightly away from edges. But with some layouts, that is not possible.

Fabricate and Assemble

Some engineers might choose a vendor that will fabricate prototype PCBs and also assemble them. A one-stop shop can produce short runs of a design and thus reduces the buying company's costs. PCB Fab Express, for example, handles PCB fabrication and assembly of up to about 200 pieces. "The complexity of designs continues to increase," said Parate of PCB Fab Express. "Customers want more layers and smaller spacing. So, as PCB fabrication technologies advance, fabricators and customers must collaborate more rather than have customers simply send Gerber files for a quick quote."

"We don't see PCB prototype fabrication going to Asia," said Parate. "Prototype PCB vendors sell time, and as product life cycles get shorter designers want fast turn around from local companies. Engineers want to pick up the phone and talk with someone who can answer their questions. We see ourselves as innovation partners rather than just vendors."

SIDEBAR: When Do You Work with a Contract Manufacturer?

For Further Reading
For information about Gerber files, go to: http://en.wikipedia.org/wiki/Gerber_File [1]

When Do You Work with a Contract Manufacturer? by Jon Titus

Source URL (retrieved on *04/21/2015 - 8:14am*):
http://www.ecnmag.com/articles/2008/04/pcb-prototype-success-cards?qt-video_of_the_day=0

Links:
[1] http://en.wikipedia.org/wiki/Gerber_File