PRINTED
CIRCUIT DESIGN

is published monthly by:
UP Media Group Inc.
2018 Powers Ferry Road, Ste. 600
Atlanta, GA 30339

Tel (678) 589-8800
Fax (678) 589-8850

Unlocking Legacy Designs

Attention to detail is the key to bringing the designs of the past into the present

by Bill Brooks

Many of you may dread the very thought of having to rework a legacy design for a product update. Legacy boards pose a challenge for the configuration managers, engineers, designers and drafters who must support them. This article will attempt to demystify the issues associated with legacy designs and suggest ways to migrate them to newer, more modern CAD platforms.

Why change?

As the old saying goes, "If it ain't broke, don't fix it!" A good PCB designer should resist the urge to "fix" a legacy design without first researching affected assemblies and getting the appropriate approvals. You may wind up fixing a problem in one product and creating one in another. If your design is subject to military (DoD) or safety agency (UL, CE) control, you must not make changes without prior approval. There are large fees associated with product certification and compliance testing that can be affected by your changes. But often return on investment (ROI) considerations can demand that old products must be changed or redesigned.

What are some of the reasons to upgrade a legacy design?

- Obsolete components—Watch out for upgraded components. Technology improvements in silicon wafer and die manufacturing have yielded improvements in older chip designs, such as faster switching signal rise times. The 74LS04 of today may not behave the same way it used to. Great articles have been written on the need to use high-speed design rules on older designs whose PCBs may have tolerated long meandering trace runs and now cannot tolerate long lines.

- Design and process improvements, and labor reduction—Customers may want new features that were not available before. Many older designs were hand-assembled and followed few guidelines for component lead spacing and clearances. The need to reduce labor costs and use automated assembly can also drive the design to change.
- Maintainability—PCB CAD software has evolved and we are often faced with the need to migrate those old designs to the new CAD platforms. Few of us have experience in reverse engineering or translating designs done in an older or obsolete software package into a newer format. How do you recover the database accurately and still have a maintainable design without creating a long costly project?

From hand tape to CAD

Some of us remember the days of hand-taped artwork. I still have my old templates and X-Acto knife. Simply put, the designer created an artwork using die cut crepe tape and clear sheets of Mylar at a scale (typically 2:1) that could be reduced to a scale of 1:1 photographically to produce the film from which the PCB was made. The orthographic film negative or positive was the common artwork format.

In the 1980s a process was developed to photographically transfer the artwork onto an orthographic erasable Mylar sheet known as Chrono-Flex. This allowed the orthographic print to become a new original artwork that could support changes by erasing old features and taping in new ones. For time-saving and archiving, I often used Chrono-Flex to clone artworks with minor differences.

Use extreme caution when handling these old artworks! The die cut tape used for pads and traces has an adhesive backing that becomes non-functional after a number of years. I hope you never have the awful experience of finding loose pad shapes and tape pieces in the bottom of the artwork envelope!

Gerber, the great equalizer

Today, instead of taped artwork, the Gerber file is the common foundation for the negatives or positives from which the board is made. If you have aging hand-taped artworks, they should be converted to Gerber format as soon as possible to preserve the integrity of the original artwork. Numerous companies specialize in the conversion of artworks and films into the Gerber format. A simple search of the Internet will yield a list of them.

When is the right time to convert that old design into the new software? For many, the right time is when there is a pending ECO or ECN against the board. If the layout is in an old software format, it gets queued for upgrade into the new software. Hand-taped artworks gets sent out to be converted to Gerber files and get imported into the PCB CAD software. Translators from one CAD platform to another are available for certain types and brands of software, and these can make the transition from old CAD to new CAD fairly effortless. Without a translator, the Gerber format is the best medium for translation.

There are a number of Gerber reading and editing packages for PC or MAC platforms. Use the Internet to locate them if you can. Many PCB design CAD packages have the ability to read in Gerber format to allow editing of the traces and pads in your own familiar editing software. But even though this gives you the ability to edit the PCB database, Gerber import will not provide you with a complete design file. The pads and traces are typically in their simplest, most difficult-to-edit form: basic graphic entities.

Although this may be a way to store the file, it is usually much more desirable to acquire a complete, editable database to use for the current revision of the board and to make it possible to make changes in the future. Some programs will let you import Gerber into the mechanical or non-functional layers. This feature permits the designer to use an actual Gerber image as a visual guide for recreating of the board. A schematic file should be made to support schematic changes and to generate a net list, though some designers will want to just create a net list and maintain that.

Typically, you load the legacy design from Gerber into your PCB editor and add the appropriate footprints to the design from your company library by superimposing them on the patterns created by the Gerber input. Often there will be differences between the Gerber representation and your library part that was created with DFM rules in mind. You will need to make adjustments to the layout as needed to incorporate these changes into the board to support DFM as much as possible, then transfer the traces from the inactive layers to the active layers.

After the net list has been loaded, the traces will contain the appropriate net list names, creating an almost exact, intelligent duplicate of the original board that can be maintained in your latest software. This gets you to the point where you can do the ECO changes to the layout and check it with design rules checking (DRC) before sign-off and release.

Upgrading the database

Moving a component without knowing the initial design requirements can have disastrous effects on certain types of boards. Not everyone keeps good documentation on the criteria that drove the locations of the parts and their spacing. Engineers and designers move on, taking with them the knowledge base that helped shape the PCB.

Electrical and mechanical engineers should be able to give the PCB designer the criteria for the design rules to be applied to the board. Careful analysis must be applied to determine the impact of any change and the costs involved in upgrading any PCB design. What are the potential impacts to tooling, automated test, RF performance, EMI, voltage spacing, current-carrying capacity, parasitic capacitance and inductance, connector locations, brackets, height clearances, logic families and material changes? Component placement, trace geometry and spacing are critical.

Pay close attention to DFM issues. Redoing a board to make it more manufacturable can be the single most effective improvement you can make to a legacy board. Be careful not to degrade the electrical performance. It is wise to have your electrical engineer perform a thoughtful review of any changes.

Specifying IPC standards for workmanship, materials, plating and processes helps to protect the quality of the PCB and sets the reject criteria for your quality assurance department. This makes it possible for inspection to accept or reject materials shipped from PCB vendors. If the boards are bad due to a manufacturing defect and that defect is not accounted for in your notes, you may still be under obligation to pay for that material, useful or not. Good drawing notes save the company money. Many old legacy designs will not have up-to-date requirements specified in the notes. Examine them for any needed changes and update the drill drawing.

Manufacturability vs. availability

A close review of the BOM vs. the availability of parts should be conducted. Scrutinize the component spacing for the axial and radial components to verify that they can be automatically inserted into the board. Check the lead vs. hole diameter clearances and look for undersized pad geometries. Check for clinch lead interferences and potential shorting conditions. Can you safely reduce the via size to a more standard size and free up more room for components? Can you consolidate drill sizes?

Here are a few more issues to consider:

- Are there tooling or test fixtures to take into consideration?
- Are there test points that cannot move? Will you have to rework the test fixture at $60 per hour?
- Are there cost-effective vendors who can stuff the boards with axial and radial or DIP packages, or will they have to hand-stuff the parts due to aging or outdated equipment? Can the parts be replaced with newer technology?
- Would that discrete logic fit into a single programmable logic chip or a smaller, more cost-effective package size?
- Would an alternate footprint or variation allow second-source parts to be used as a cost savings?

Reducing cost

A cost-reduction design cycle is a good idea, since cost is usually the reason for the legacy rework in the first place. Many old chips can be emulated or modeled in programmable logic devices (PLDs) or FPGAs. This can drastically reduce the real estate required for the same logic functions and give the engineers the option of making changes to the firmware through programming instead of layout changes.

Changing those discrete resistors and caps into 0805, 1210 or 0603 surface-mount packages can greatly reduce the cost of components, assembly and the real estate to hold them. Fine-pitch parts and BGAs are more common now, but beware that these components require special handling and newer up-to-date equipment to handle them.

Most existing chips are running at rise times that are faster than when they were manufactured, meaning the lines on the board become more critical to the signal performance. RF reflections can occur in fast rise-time transitions and can mess with clocks and switching state circuits. Be aware of the need to follow the high-speed design guidelines when upgrading boards, even if the original board was more tolerant due to slower rise times.

Things to do

- Preserve the integrity of the original artwork. Minimize handling and get photos made of your original artwork. Have it scanned into Gerber format for archiving as data.
- Check for errors in part number, revision, dimensioning, tolerances, missing pads, peeled back artwork traces, shorts open, hole sizes and tooling holes.

- Add fiducials for surface-mount designs and check for DFM issues.
- Add or upgrade notes and tolerances to the drill drawing specifying IPC standards.
- Add the company name, address, date of the board, board number, revision number and layer number to each sheet of the film so it can be easily identified at the vendor or in document control.
- Review the BOM or parts list for out-of-stock or unavailable, obsolete parts.
- Review with the EE and ME any potential changes for cost savings and circuit corrections, or possibly replacing components or complete circuits with PALs, GALs, FPGAs or other firmware options.
- Review agency approvals and restrictions.
- Look at trace and pad spacings and hole sizes, annular rings and component footprints for potential problems.
- Consider high-speed design rules and their potential effects on older boards.

Things not to do

- Don't handle the manual tape-up artworks if at all possible. Use extreme caution in handling and shipping.
- Don't change the circuit without first reviewing with the EE and the ME and checking for potential impacts to tooling, fixtures and interference with other mechanical, electrical, RF or thermal features of the entire assembly. Manufacturing review and agency review is essential to avoid more costly impacts to the product.
- Don't miss the opportunity to do a cost savings analysis when you upgrade a legacy design.

Reverse engineering

The following is an example of a typical reverse engineering process for PCBs:

- Review existing design materials with your EE, ME and manufacturing or assembly house.
- Create a schematic in your CAD tool of choice, if necessary, or manually create a net list that matches the circuit.
- Make notes about the key features that may not be changed—i.e., mounting, component locations, test pins or pads, fiducials, connectors, etc.
- Get design converted to a common format, such as Gerber.
- Check the conversion process for errors.
- Import the Gerber into your CAD tool as a guide to do the layout.
- Create necessary parts for placement and use the Gerber to get accurate placement.
- Select all the traces from the Gerber and convert them to actual traces in the design.
- Load the net list to add the nets to the traces.
- Run a design rules check to verify that the new design matches the old design and the schematic.
- Add any notes needed to upgrade the drill drawing.
- Do a prototype of the board and verify that the circuit has not changed functionality prior to release to regular production use.

Obviously, other steps may be needed depending on your CAD tools and company organization. Use of good design practices and thorough checking can make the transition from an old legacy design to an easily maintainable product a breeze.

Bill Brooks holds the IPC CID certification and has been designing PCBs since 1973. He is currently working as a printed circuit design engineer at Datron World Communications in Vista, CA. His email address is bbrooks@dtwc.com.