

# SEVEN HABITS OF HIGHLY EFFICIENT PCB DESIGNERS

JOHN MCMILLAN, MENTOR A SIEMENS BUSINESS

**Mentor**<sup>®</sup>  
A Siemens Business

P C B D E S I G N

W H I T E P A P E R

[www.mentor.com](http://www.mentor.com)

## INTRODUCTION

Every step taken during the printed circuit board (PCB) design process is taken purposefully. PCBs are essentially complex puzzles where the designer takes a specified list of components and a schematic diagram, typically with multiple sets of rules and constraints, then, following industry standards, guidelines, and best practices, places and connects them precisely for fabrication and use in electronic products. This paper describes seven habits that highly efficient PCB designers take as they study, prepare, visualize, strategize, and complete PCB designs.



### 1. EFFICIENT PCB DESIGNERS PAY ATTENTION TO DETAILS

From creating the schematic symbols to polishing off the layout for fabrication, assembly, and test, even the smallest missed detail can make or break a printed circuit board design. Efficient PCB designers make it a point to understand every aspect of the design flow and, when in doubt, they check things out. Preparation is key and it starts long before the PCB layout tool is launched. From building symbols and component land patterns from scratch, to obtaining them from tool libraries or online resources, PCB designers check them out. Why? Because they can't afford for them to be incorrect no matter the source since a single incorrect symbol or land pattern alone can result in a costly board respin.



Taking time to review the design's bill of materials (BOM) and the component datasheets to verify both the components' dimensions and pin-outs is time well spent. Understanding details, such as which components support gate and pin swapping, can enable designers to optimize placement and routing during the PCB layout phase. Defining alternate package types for component and solder-side placement can save PCB real estate. Some surface-mount land patterns can even be optimized by including pre-defined breakout/fanout vias and thermal pads. An efficient designer will also identify opportunities to recommend schematic changes; for example, replacing individual resistors with a resistor network.

Other details, like defining placement boundaries within land patterns, will ensure that minimum component spacing is optimized and detectable by design rule checks (DRC). Enhancing a component silkscreen, particularly on high pin count devices with notations that identify pins, can help engineers and technicians during prototype debugging. Whether it's adding additional stitching vias to connect power or ground fills and planes or additional silkscreen notations, effective PCB designers understand all of the electrical design aspects and requirements necessary for an optimal design.

### 2. EFFICIENT PCB DESIGNERS STUDY, PLAN, STRATEGIZE, EXECUTE, AND CAN REGROUP QUICKLY

Efficient designers must become familiar with the schematic and think multiple steps ahead. Understanding a design's schematic enables the designer to expedite placement. For example, defining groups or rooms of associated components helps to expedite component placement. Understanding why and where discrete components such as decoupling capacitors and bypass resistors are needed on a board helps avoid signal and power integrity issues.

Efficient PCB designers also know the value of setting up the layout session properly and can be very resourceful. Many designers develop and save starter templates for different design starts. These often include the designer's tool operational preferences, such as hot keys that execute specific routines, color mapping, default trace widths, and layer stack-up definitions. Efficient designers know how to apply design constraints and rules to ensure design rules are followed during placement and interactive and automatic routing phases. They also develop route studies and strategies that prioritize nets in order to ensure that even the most extensive rule-driven designs are accomplished.



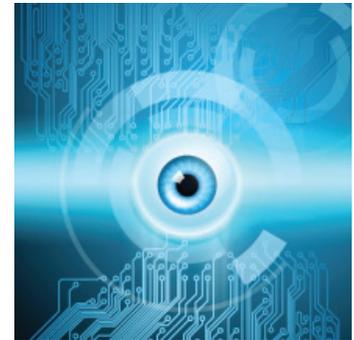
Let's face it, changes happen. There can be a variety of different reasons for design updates, such as the discovery that a component is no longer available or that a component like an FPGA required a new pin-out. It is not uncommon for a PCB designer to get an updated netlist, or perhaps several updates, during the layout phase.

The ability to re-group and adjust quickly is critical for PCB designer efficiency, especially if a design's release schedule is time-to-market critical. Efficient designers often create phased or daily backup versions of the design as part of their process enabling them to easily revert back to an earlier version if it expedites the update process. The good news is that efficient PCB designers are prepared for change and they get better, smarter, and faster with every design they complete.

### 3. EFFICIENT PCB DESIGNERS VISUALIZE

Component placement is a critical step and sets the stage for a successful PCB design layout. Component orientations, top or solder placement, and spacing that avoids shadowing and ensures optimal solderability and testing are just a few of the details that designers are aware of, plan for, and visualize.

Factors such as aligning components and break-out vias help to ensure that routing lanes are not blocked. Efficient designers can study a rat's nest of connections and begin to visualize and plan a routing strategy before the first trace is connected. They know what should be routed manually and what can autorouted. Highly efficient designers may even use specific techniques to steer routing by placing route boundaries or temporary fences and keep-out areas.



Efficient PCB designers visualize the design not only from a layout perspective but also from a manufacturing perspective. They are aware that fabricators have their own internal processes and design rule checks. Details such as knowing which components require additional placement room during layout, perhaps driven with placement boundaries for example, can minimize or totally eliminate rework time after boards are fabricated.

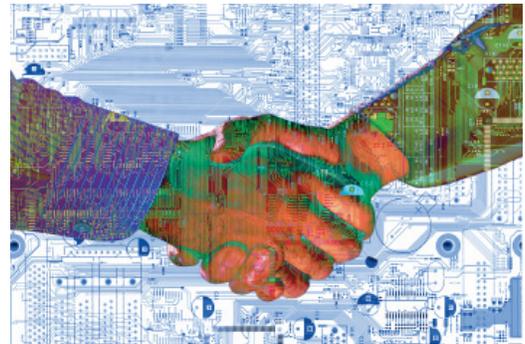
### 4. EFFICIENT PCB DESIGNERS VALUE WORKING WITH, AND CONSULTING WITH, PEERS

In the infamous words of singer Vanilla Ice's rap song; Ice Ice Baby, "stop, collaborate and listen..." Efficient PCB designers don't work in a vacuum; they understand the value of collaboration. For example, early on they work closely with mechanical design teams to ensure the proper placement of mounting holes and physical interfaces such as connectors, LEDs, and displays. And, when problems are identified, they provide valuable feedback to the mechanical teams to improve the design. They think about the end product, embrace ECAD-MCAD collaboration, and keep the end product in mind throughout the PCB layout phase.

Understanding design requirements, such as design rules and signal integrity constraints that drive connectivity, is a must. PCBs may even have specific signal integrity (SI) engineers who utilize models, run simulations, and apply strict routing and timing rules such as defining net topologies and differential pairs with minimum and maximum trace lengths, matched lengths, maximum separation, etc. Understanding and planning for how nets with T-points, like memory buses or clocks that require tuning, consume board real estate are some examples.

Highly efficient PCB designers also have relationships with PCB fabricators and understand the importance of designing for manufacturability, assembly, and testability. Designing with fabrication in mind ensures that traces are not compromised by being too close to guide pins, edges, or mounting holes.

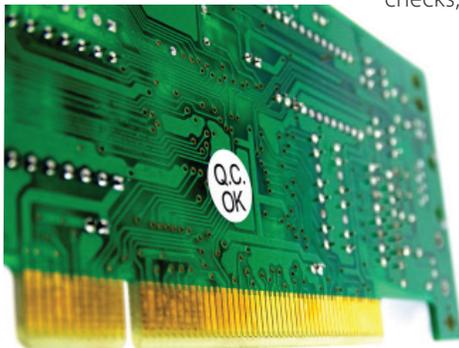
Talk to the fabricator making the PCBs and request a DFM (design for manufacture) review. Fabricators know their capabilities and process limits and should be able to advise whether adding a feature like copper thieving is advantageous. Then it's up to you as a designer to add them, which keeps you in control of the design. Understanding that even the slightest adjustments that a fabricator might make to a PCB's controlled-impedance traces can cause unexpected design performance.



Collaborating is essential to PCB design success, becoming more important depending on the size of the business and complexity of design. Collaborating can include component and model librarians, EMC, thermal, QC, and NPI engineers, and even board fabricators. That said, even the best PCB designers understand the value of peer reviews. Some designers can be so involved with their work that even simple things are overlooked, such as a missing or misplaced reference designator or noticing that nonfunctional pads on internal layers were not removed.

## 5. EFFICIENT PCB DESIGNERS STRIVE FOR PERFECTION

Do you remember when getting 95% on a test was good enough for an "A" grade? Well, that's certainly not "passing" in PCB design. In fact, even the smallest error, such as an incorrect pad size or a single trace that's too close to a mounting hole, can result in a PCB re-spin. Efficient PCB designers do everything they can to achieve design perfection. From online design rule checks (DRC) and designing for manufacturing and assembly (DFMA) checks, every effort is made to ensure that the design work is accurate.



Design reviews are critical for eliminating ambiguity between design stakeholders. Effective PCB designers often utilize specific post-design programs that combine DFM and new product introduction (NPI). These tools ensure a smooth transition to fabrication, assembly, and test from the PCB design environment. These design for fabrication (DFF) analysis tools can check fabrication files and even flag problems directly in the layout environment for immediate correction prior to sending designs out for manufacturing. They can also review bill of material (BOM) data and alternate sources of supply for assembly.

## 6. EFFICIENT PCB DESIGNERS CONTINUE TO LEARN

All PCB design steps, processes, and procedures referenced throughout this paper are derived from education, training, and hands-on experience. First and foremost, a strong foundation is absolutely essential to being a PCB designer, especially with the pace with which new components, technologies, and processes are evolving.

Continuing education is a must in the electronic product design world. This includes keeping up with the latest industry standards including IPC, ANSI, MIL, etc. as well as PCB fabrication and assembly processes. Technologies like the Internet of Things (IoT) and rigid-flex circuits are becoming commonplace in today's electronics from automotive to medical to consumer and beyond. PCB



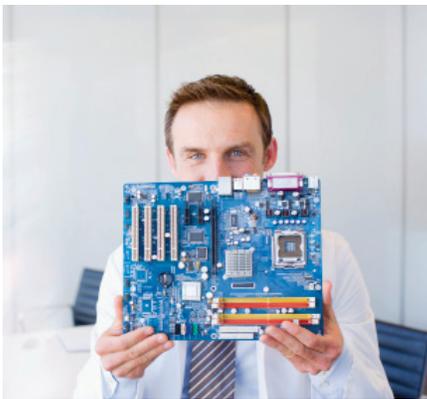
PCB Carolina 2016

designers contend with a constant stream of new and improved components, packages, and fabrication, manufacturing, pre- and post-test and assembly processes as well.

Efficient PCB designers understand that they are not an island, but are rather part of a greater community of designers with a common goal. They subscribe to publications like PCD&F and Circuits Assembly, are members of and/or regularly attend IPC chapter meetings and conferences, and may even present papers to peers at PCB conferences. Many PCB designers even seek industry credentials or certifications through training and testing like IPC's Certified Interconnect Designer (CID) program.

### 7. EFFICIENT PCB DESIGNERS DESIGNS REFLECT THEIR PASSION

In as much as canvas is the medium to an artist, PC monitors are the virtual canvas for PCB designers. In fact, giving an identical design database to ten PCB designers is likely to result in ten different finished results. Perhaps the differences are subtle or they might be visibly quite noticeable as each PCB designer has his or her own unique design style where items that aren't immutable or fixed can be tailored and unique.



For example, very rarely does a PCB designer kick-off an autorouter and is 100% percent satisfied with the finished result. Typically, PCB designers choose to hand-route physical interfaces and sections of highly constrained busses like DDR. Nets with T-points, clocks, TX and RX traces, RF circuitry, etc. are also considered for partial or complete pre-routing. Trace characteristics like corner chamfers and tuning (rule-based) are just some of the other design elements that may vary from designer to designer.

Efficient PCB designers are passionate and meticulous about their work. It's a job that requires a vast array of knowledge, skills, and attention to detail. Most importantly, they have a unique passion for electronics design. They enjoy the mental challenge, the sense of gratification, and the pride their contributions bring to the end products.

### CONCLUSION

I referred to PCB design as a complex puzzle. You start with a design envelope that provides the frame, a set of components that need to be logically and strategically placed, and a schematic diagram with guides and rules that represent how components are interconnected. But the puzzle box for electronic products doesn't have a picture on top to guide the placement of every component, trace, or pad. All is done by design, preferably efficient design.

With the aid of electronic design CAD software, PCB designers both pilot and navigate each design aspect, applying their education and experience with the goal of first-pass design success. Developing design habits that expedite design completion, improve design quality, and enhance productivity are instrumental to highly efficient PCB design.

**For the latest product information, call us or visit: [www.mentor.com](http://www.mentor.com)**

©2017 Mentor Graphics Corporation, all rights reserved. This document contains information that is proprietary to Mentor Graphics Corporation and may be duplicated in whole or in part by the original recipient for internal business purposes only, provided that this entire notice appears in all copies. In accepting this document, the recipient agrees to make every reasonable effort to prevent unauthorized use of this information. All trademarks mentioned in this document are the trademarks of their respective owners.

**Corporate Headquarters**  
**Mentor Graphics Corporation**  
8005 SW Boeckman Road  
Wilsonville, OR 97070-7777  
Phone: 503.685.7000  
Fax: 503.685.1204

**Sales and Product Information**  
Phone: 800.547.3000  
sales\_info@mentor.com

**Silicon Valley**  
**Mentor Graphics Corporation**  
46871 Bayside Parkway  
Fremont, CA 94538 USA  
Phone: 510.354.7400  
Fax: 510.354.7467

**North American Support Center**  
Phone: 800.547.4303

**Europe**  
**Mentor Graphics**  
Deutschland GmbH  
Arnulfstrasse 201  
80634 Munich  
Germany  
Phone: +49.89.57096.0  
Fax: +49.89.57096.400

**Pacific Rim**  
**Mentor Graphics (Taiwan)**  
11F, No. 120, Section 2,  
Gongdao 5th Road  
HsinChu City 300,  
Taiwan, ROC  
Phone: 886.3.513.1000  
Fax: 886.3.573.4734

**Japan**  
**Mentor Graphics Japan Co., Ltd.**  
Gotenyama Trust Tower  
7-35, Kita-Shinagawa 4-chome  
Shinagawa-Ku, Tokyo 140-0001  
Japan  
Phone: +81.3.5488.3033  
Fax: +81.3.5488.3004

**Mentor**<sup>®</sup>  
A Siemens Business

MSB 09-17 TECH16030-w