

EIGHT STEPS FOR ENSURING PCB DESIGN SUCCESS

DANIT ATAR AND ALEX GRANGE, MENTOR GRAPHICS



P A D S

W H I T E P A P E R

w w w . p a d s . c o m

INTRODUCTION

Printed circuit boards, PCBs, are the backbone of electronics products, providing the electrical system upon which the performance, life span, and reliability of the end product depend. When designed correctly, products with high-quality circuitry have lower field failure rates and fewer field returns. As a result, products can be produced at lower cost and higher profit. To produce high-quality PCBs on time, without the penalty of added design time and high rework costs, it is important to detect design and circuit integrity issues early in the design process.

To get products out the door quickly and reliably it makes sense to automate the process with design tools, but how do you ensure design success? What things should you look for to maximize design efficiency and product quality? Of course your tools should be intuitive, easy to use, and powerful enough to overcome complex design challenges, but what else should you look for? This paper lists eight steps you can take to ensure PCB design success.

STEP #1 - DON'T SETTLE FOR BASIC SCHEMATIC CAPTURE

Schematic capture is essential for generating the logical connectivity of the design. It must be accurate, easy to use, and integrated with layout for the design to be successful.

Simply capturing the schematic and passing it to layout isn't enough. To create high-quality designs that perform as intended, you'll need to ensure that optimal parts are used, and that simulation analyses can be performed so that designs can be manufactured without issues.

HOW PADS CAN HELP

PADS takes schematic capture productivity, quality, and efficiency to the next level with a powerful design environment that is easier to learn and use, and is more intuitive than ever before. PADS gives you everything you need to get your job done right the first time, including full schematic design, up front simulation, and easy variant definition.

- Immediately identify design rule violations with the Dynamic Graphical Rule checker, to eliminate time-consuming respins.
- Reduce time spent laying out new designs by simply dragging and dropping a symbol onto a net to automatically create connections.
- Make dozens of connections in seconds with the new multi-net connections feature.
- Save time using "My Parts" to select "Recently Used" symbols and preserve your "Favorites" symbols.
- Use automatic component-to-bus connections. Moving a component placed on a bus updates nets and connections.
- Easily remove nets or entire buses with the Cut Nets By Area feature.

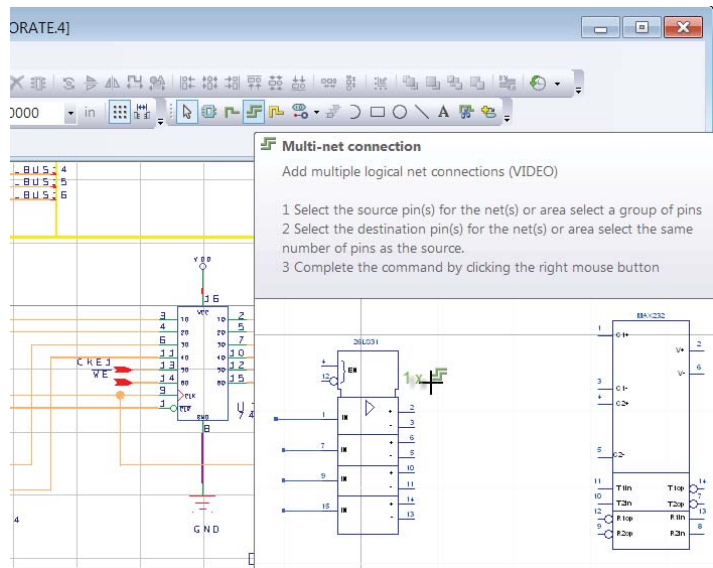


Figure 1 - With progressive tool tips, critical help is at your fingertips, making it easy to use the power in PADS schematic design. Hover over an icon for a full description and a display of the command.

STEP #2 – DON'T SKIMP ON LIBRARY MANAGEMENT

Library Management is an essential part of the design process. Easy component creation and management are absolutely necessary for quickly selecting optimal parts and placing them within the design.

PADS allows you to maintain one library for all design tasks and updates the library in real time for ease of use and accurate design development. You can access all component information from a single spreadsheet without data redundancy, multiple libraries, or time-consuming tool overhead.

HOW PADS CAN HELP

- Integrated Central Database** — For each project you work on, PADS creates storage area for schematics, constraints, and layout design data. With built-in data synchronization and cross-probing, the central library provides real-time notification of design synchronization status.
- Starter Library** — Starting a library is always a time-consuming and exhausting task. PADS provides more than 11,000 proven parts including 1700+ xDX symbols and 1000+ IPC-compliant and manufacturer-recommended PADS decals.
- PartQuest™** — With PADS you will be able to access component supplier data for a fast, accurate, and easy start to your design process with parts you select directly from online component suppliers. You will be able to access and load qualified third-party parts data and design content into PADS for an efficient work process.
- Component Management** — With PADS you can manage components directly from the schematic environment, reducing component management time by up to 60%. This integration with the schematic environment eliminates costly re-designs and quality problems that often go undetected until later stages in the design process. Your design is constantly kept in-sync and up-to-date with the component database.

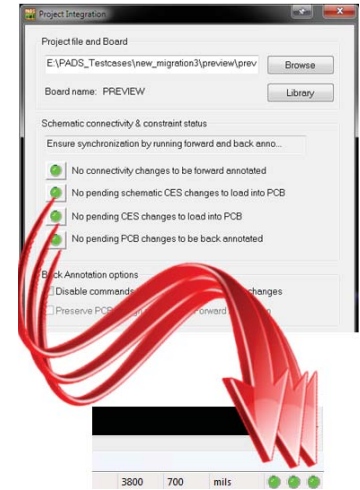


Figure 2 - PADS integrated, central database architecture ensures that you use the most up-to-date data, ensuring correct design data flow.

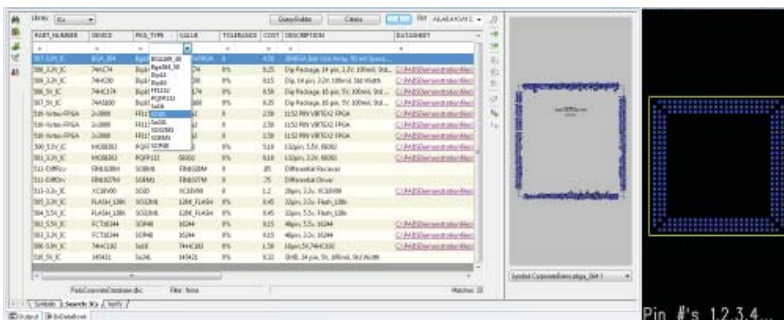


Figure 3 - Reduce the time you spend searching and manipulating part properties by up to 60% with PADS component management.

- Central Library** — Centralized storage of all library elements includes a built-in mechanism for ensuring data consistency. The central library can be man-aged in all phases of the design flow, directly from the component management system.
- Land Pattern Creation** — Footprint creation is the most time-consuming part of library management. PADS wizards create IPC-7351B compliant land patterns, reducing land pattern creation time by up to 89%.

STEP #3 – HAVE AN EFFECTIVE WAY TO MANAGE DESIGN CONSTRAINTS

With the complexity of today's critical, high-speed designs, it is very difficult to design, constrain, and manage trace routing, topology, and signal delays without a way to manage those constraints. In order to build successful products within the first iteration, constraints must be set early in the design flow so that the design will meet the required design goals. Good constraint management keeps you from using parts that are cost-prohibitive or cannot be sourced, and ultimately will ensure that your boards meet performance and manufacturing requirements.

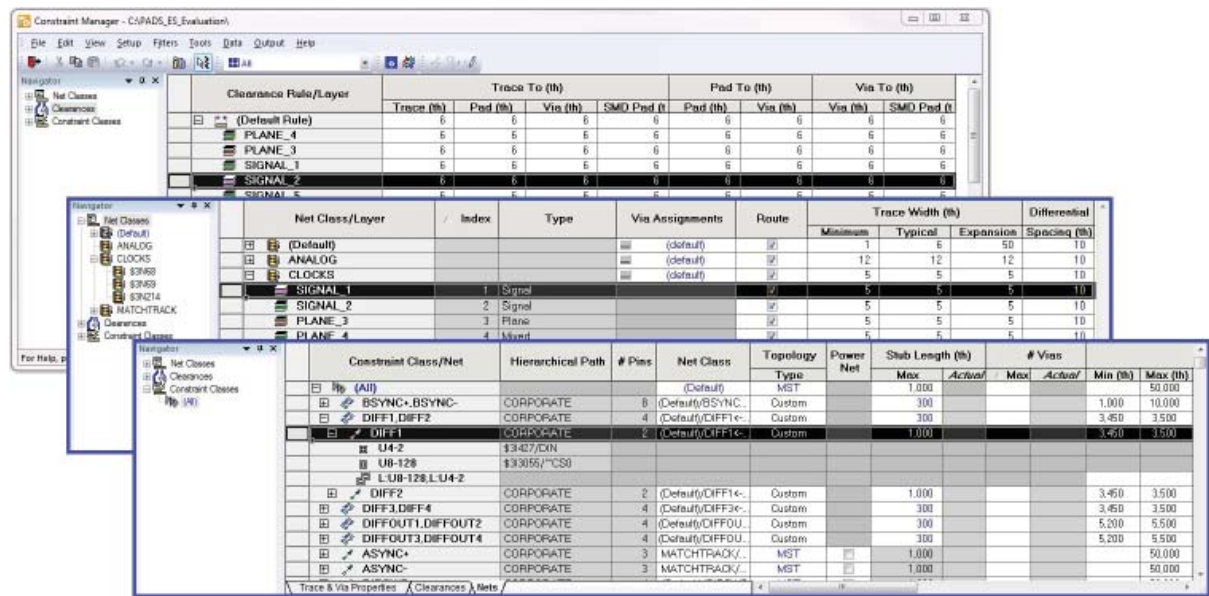


Figure 4 - The spreadsheet-based rule interface greatly simplifies all aspects of constraint management and verification within the PADS design flow.

HOW PADS CAN HELP

- PADS has a powerful and easy-to-use constraint management system for the creation, review, and verification of PCB design constraints. The spreadsheet-based interface greatly simplifies all aspects of constraint management within the design flow.
- PADS supports many constraint types, including physical manufacturing and assembly constraints such as via assignments, clearances, timing constraints, and more.

STEP #4 – ENSURE YOU HAVE THE LAYOUT CAPABILITIES YOU NEED

In recent years, designing a PCB layout has become significantly more complicated. Designs have increased in density in order to provide smaller and more portable electronics. Operating frequencies have also increased, requiring you to assess electrical characteristics that could previously be ignored to ensure a functional design. To keep up with this increased complexity, designers must have a wider range of functionality for defining advanced rule sets, creating unique RF shapes, and implementing corrective structures to improve the overall performance of the design.

During the layout process, an intelligent layout tool assists creating an efficient placement and routing strategy. Precise placement reduces violations later in the design cycle, allowing you to complete projects more quickly and with fewer errors.

Although manual routing is generally used to meet the true design intent, a good balance between interactive routing and autorouting can help meet market deadlines with improved design quality. Autorouting can also help with difficult tasks such as differential pair routing, net tuning, manufacturing optimization, and microvia and build-up technology. Using autorouters becomes much more efficient if you plan your routing strategy in advance.

Another challenge is that modern PCBs maintain thousands of nets, which can create difficulty when routing crucial areas within the design. The best method of avoiding this issue is to organize netlines into groups to create an efficient routing strategy. Creating planning groups makes it possible to mark and filter groups of nets to highlight critical nets that need to be routed.

HOW PADS CAN HELP

- PADS advanced place-and-route capabilities make it possible to create any design, regardless of density and high-speed constraints, with efficiency and ease while making sure your PCB meets your design goals.
- PADS interactive routing is highly flexible, allowing you to choose how to drive routing and how to resolve clearance conflicts based on design rules.
- PADS interactive routing handles high-speed nets, differential pairs, and matched-length groups easily, enabling you to meet all required high-speed constraints.
- Design rule checks (DRC) adhere to all constraints, ensuring that no rule is violated and reducing rework after the board is completed.
- PADS supports RF and microwave designs, via shielding for channel/co-planar waveguide design, automatic via fill for any copper shape, and support for chamfered or square corners.

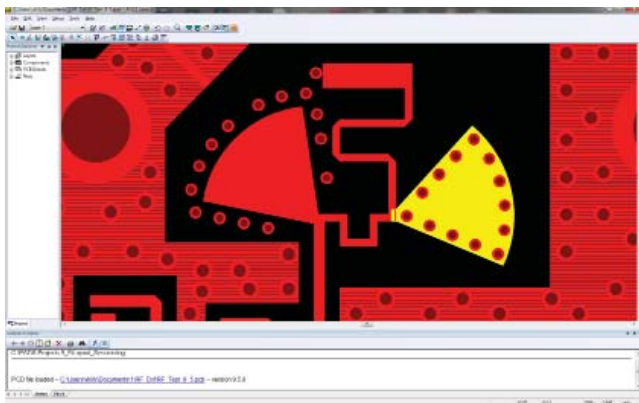


Figure 6 - RF design features include via-matrix fill for complex copper shapes.

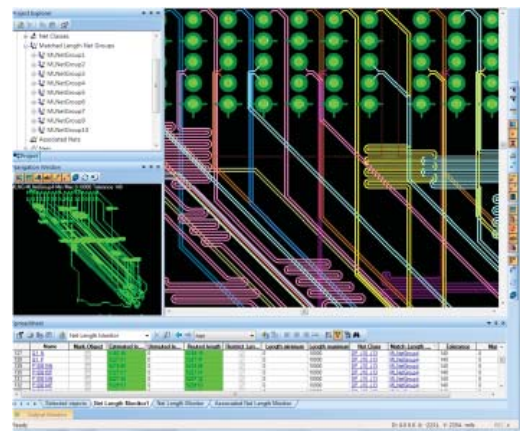


Figure 5 - Whether you prefer to route interactively, automatically, or using a combination of both, PADS gets the job done quickly and accurately.

- With physical design reuse you can quickly place repetitive circuits in multi-channel designs or reuse proven circuitry as “golden circuits.”
- Advanced packaging tools are available to improve design quality when designing with bare die. With them, you can speed the creation of manufacturing data, and the design of ball grid arrays (BGAs), chip-scale packages (CSPs), multi-chip modules (MCM), and chip-on-boards (COBs).
- PADS optimizes product quality and reduces cycle time and costs with built-in audits for design for test (DFT) and design for fabrication (DFF). Testability analysis and verification ensures 100% testability for all nets on your board.

- PADS has a simple and efficient user interface for autorouting that adheres to your constraint hierarchy. High-speed rules enable differential pair routing and unique rule assignments to generate sophisticated designs for high-speed applications.

STEP #5 – ANALYZE YOUR BOARD FOR PROBLEMS BEFORE THEY OCCUR

To meet product deadlines and improve PCB yield, it is absolutely necessary to detect, analyze, and resolve signal integrity and manufacturability issues during the design stage. At each successive step of the production process, the cost of rectifying a problem increases tenfold over the previous step. Detecting issues early will lower production costs and avoid costly re-runs through the manufacturer.

HOW PADS CAN HELP

- **Analog Simulation** – PADS includes an integrated, easy-to-use SPICE simulator for board-level analog simulation. Multiple models are supported on each symbol, allowing you to select the optimum performance components for your design. Use PADS for statistical, Monte Carlo, and Worst Case analyses to predict likely design yields.

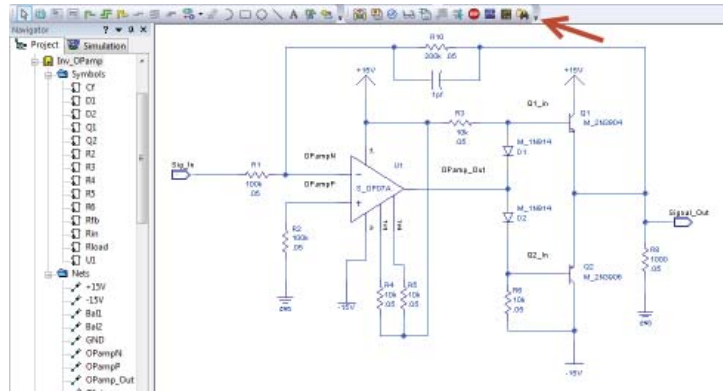


Figure 7 - PADS analog simulation is integrated directly into the PCB design flow.

- **Signal Integrity Analysis** – PADS uses powerful and easy-to-use HyperLynx® technology to ensure design goals are met without having to be an SI engineer. Avoid time-consuming configuration errors and un-routable constraints with pre-route analysis, then confirm that your design rules and constraints are met using a batch design rule check for post-layout verification.

- **Thermal Analysis** – With PADS you can run early thermal analysis of your board quickly and easily. As soon as part placement is complete, you can analyze board-level thermal problems on placed, partially routed, or fully routed PCB designs. Temperature profiles, gradients, and excess temperature maps enable you to resolve board and component overheating early in the design process.

- **Design for Manufacturability (DFM) Analysis** - Optional DFM analysis minimizes production issues, helping you reduce the number of revision spins per design, and saving time in your release schedule. With PADS, you can resolve problems such as resist slivers, unintended copper exposed by solder masks, and improper testpoint-to-testpoint spacing during layout.

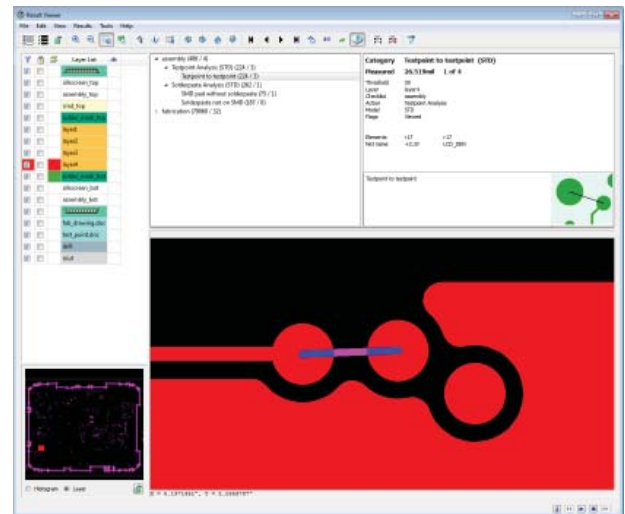


Figure 8 - PADS DFM Analysis improves fabrication yields by identifying correctable resist slivers, copper neck downs, and inferior power connections.

STEP #6 – INCREASE COMMUNICATION BETWEEN DESIGN DISCIPLINES

Collaboration between different design disciplines can have a profound effect on process efficiency. This is increasingly important throughout the entire design process in the communication between mechanical and electrical designers when incremental changes occur in either discipline. With collaboration between MCAD and ECAD, designers' violations can be detected and corrected early, whether they're on the mechanical or the electrical side. This makes the design flow easier and increases worker efficiency.

HOW PADS CAN HELP

- PADS ECAD/MCAD collaboration enables electrical and mechanical design teams to work together throughout the entire design process.
- Early detection of interference issues can reduce design costs by eliminating costly design re-spins of printed circuit designs and mechanical enclosures through virtual prototyping.
- ECAD/MCAD collaboration provides ECAD and MCAD teams with a consistent and continuous communication channel that keeps design teams synchronized as they work in their own systems' comfort zones.
- ECAD/MCAD collaboration facilitates fast, real-time collaboration on "what-if?" scenarios and evaluations in real time, thus providing everyone with immediate feedback and avoiding time-consuming rework.
- Shared functional and mechanical requirements ensure that quality, reliability, and performance are optimized within tight form factor constraints.

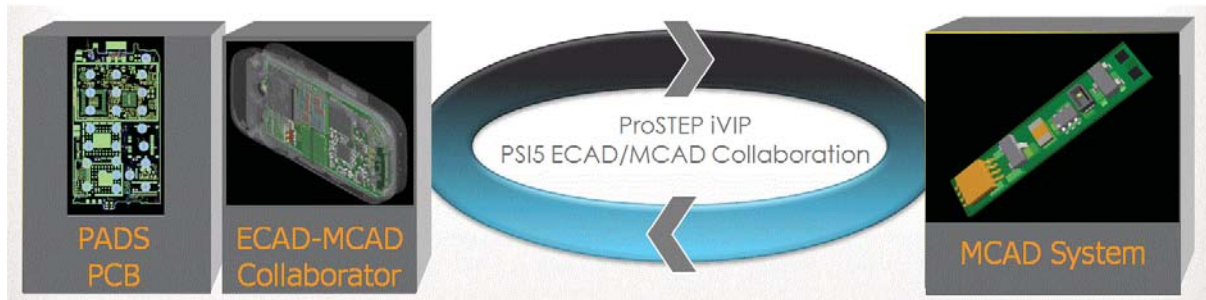


Figure 9 - ECAD-MCAD collaboration reduces design cost by preventing costly re-spins due to mismatches between PCB designs and mechanical enclosures.

STEP #7 – SAVE, COMPARE, AND STORE PCB DESIGN DATA

Proper archive management is essential to back up and manage design data, and can highly increase your productivity. Simply storing data in a vault isn't enough. You need to be able to efficiently create, index, and restore archives within the vault. It's important to be able to quickly search for content within the vault, add comments to help differentiate between different versions, and compare between schematic and layout files.

Having an efficient vault will eliminate the risk of losing design data, and will even help you to quickly start future project, by using stored schematic and layout files as the foundation of your new project.

HOW PADS CAN HELP

PADS saves all design information in a secure vault on your PC or network, and adds collaborative tools for streamlining and improving design reviews. You can create multiple backups of your project data and easily retrieve that data later for review and modifications.

View and search the vaults to see contents quickly and easily with graphical preview. Improve team collaboration with archive searching, report generation, and comparison. Add comments and information easily with intelligent red-lining that associates specific design objects and organizes comments logically by issue or topic.

STEP #8 – HAVE A GOOD TECHNICAL SUPPORT SYSTEM AND A ROAD TO THE FUTURE

The design task is complicated enough without having to search for ‘how-to’ and for support to overcome complexities in your PCB design tool. Having to search for answers online, and not getting the response you need from the product support team can be frustrating, time consuming, and even costly.

Professional, reliable, effective, efficient, and quick technical support services for your design tool makes a difference. Buying you design tool from a company that recognizes the importance of support, and gives you the quality feedback that you need, can make your life a lot easier and increase you productivity.

HOW PADS CAN HELP

- An award-winning, global customer support organization helps design teams accelerate product development and increase product quality.
- FIVE-STAR support provides frequent product enhancements and new releases as well as timely answers and highly effective solutions from specialized customer application engineers.
- Access to *SupportNet* helps you troubleshoot technical issues through detailed TechNotes and video how-to's, download the latest releases, reference documentation, and manage service requests 24x7x365.
- PADS personal automated design systems leverages Mentor Graphics' investment and strength in PCB system design technology to meet your needs, both now and in the future. PADS is scalable, and provides an end-to-end solution, ranging from high-speed routing to simulation and analysis. A variety of configurations and options are available for specific needs, as is an upgrade path to Xpedition®.

For the latest information, call us or visit: www.pads.com

©2014 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 Garden
7-35, Kita-Shinagawa 4-chome
Shinagawa-Ku, Tokyo 140-0001
Japan
Phone: +81.3.5488.3033
Fax: +81.3.5488.3004

