Getting Started in PCB Design

Mentor Graphics®

PADS®
Getting Started in PCB Design

PADS®
TABLE OF CONTENTS

- Getting Organized
- PADS Environment
- Design Flow
- Library Parts
- Schematic
- Welcome
- Netlist Creation
- PCB Design Setup
- Placement
- Design Rules
- Routing
- Design Verification
- CAM Documents
- References


This document contains information that is proprietary to Mentor Graphics Corporation. The original recipient of this information may duplicate this content in whole or in part for internal business purposes only, provided that this entire notice appears in all copies. In duplicating any part of this information, the recipient agrees to make every reasonable effort to prevent the unauthorized use and distribution of the proprietary information.

U.S. Government Restricted Rights. The SOFTWARE and documentation have been developed entirely at private expense and are commercial computer software provided with restricted rights. Use, duplication or disclosure by the U.S. Government or a U.S. Government subcontractor is subject to the restrictions set forth in the license agreement provided with the software pursuant to DFARS 227.7202-3(a) or as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software - Restricted Rights clause at FAR 52.227-19, as applicable.

Contractor/manufacturer is:
Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: www.mentor.com/supportnet
Contact Your Technical Writer: www.mentor.com/supportnet/documentation/reply_form.cfm

TRADEMARKS: The trademarks, logos and service marks (“Marks”) used herein are the property of Mentor Graphics Corporation or other third parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the respective third-party owner. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics’ trademarks may be viewed at: www.mentor.com/legal/trademarks/.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Page</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Welcome</td>
</tr>
<tr>
<td>3</td>
<td>New to PCB Design?</td>
</tr>
<tr>
<td>5</td>
<td>Getting Organized</td>
</tr>
<tr>
<td>13</td>
<td>Design Flow</td>
</tr>
<tr>
<td>17</td>
<td>PADS Environment</td>
</tr>
<tr>
<td>21</td>
<td>Library Parts</td>
</tr>
<tr>
<td>27</td>
<td>Schematic</td>
</tr>
<tr>
<td>33</td>
<td>Netlist Creation</td>
</tr>
<tr>
<td>37</td>
<td>PCB Design Setup</td>
</tr>
<tr>
<td>41</td>
<td>Placement</td>
</tr>
<tr>
<td>45</td>
<td>Design Rules</td>
</tr>
<tr>
<td>49</td>
<td>Routing</td>
</tr>
<tr>
<td>55</td>
<td>Design Verification</td>
</tr>
<tr>
<td>59</td>
<td>CAM Documents</td>
</tr>
<tr>
<td>63</td>
<td>References</td>
</tr>
</tbody>
</table>
Welcome

Welcome to Getting Started in PCB Design, a high-level introduction to the PCB design process and how PADS PCB solutions can improve your design flow.

Perhaps you're new to printed circuit board design, or maybe you're a seasoned professional. It might be that recent changes in your company have brought you new tasks and responsibilities, including those associated with the design and production of state-of-the-art electronic products. Or perhaps you might just be interested in exploring the advanced capabilities now available to help you increase your productivity and improve your product's quality.

Regardless of your motivation, in just 60 easy-to-read pages, this guide will show you how to take a design from concept to completion in a format that will prepare you to develop your own designs with confidence and accuracy. Each chapter will start with an overview of basic design concepts, and then proceed with a section on how PADS solutions can be used to streamline the process.

PADS PCB Design Solutions offer you considerable power and control over the design process. Whether your designs are simple interface boards or complex system designs, you'll find all the features and performance you'll need to get the job done.
Welcome to *Getting Started in PCB Design*, a high-level introduction to the PCB design process and how PADS® PCB solutions can improve your design flow.

Perhaps you're new to printed circuit board design, or maybe you're a seasoned professional. It might be that recent changes in your company have brought you new tasks and responsibilities, including those associated with the design and production of state-of-the-art electronic products. Or perhaps you might just be interested in exploring the advanced capabilities now available to help you increase your productivity and improve your product's quality.

Regardless of your motivation, in just 60 easy-to-read pages, this guide will show you how to take a design from concept to completion in a format that will prepare you to develop your own designs with confidence and accuracy. Each chapter will start with an overview of basic design concepts, and then proceed with a section on how PADS solutions can be used to streamline the process.

*PADS PCB Design Solutions offer you considerable power and control over the design process. Whether your designs are simple interface boards or complex system designs, you’ll find all the features and performance you’ll need to get the job done.*
Any process is easier to learn when there is an underlying structure that holds it all together. Printed circuit board design is no exception; there is a simple flow that keeps the process organized and helps to ensure design success. Once you're familiar with it, you're well on your way to creating consistent designs that can be manufactured and tested easily.

Using the many advanced design capabilities available in the PADS products, you will learn how to set up your working environment, capture your design, verify the integrity of the design, produce the physical layout, and generate all necessary manufacturing outputs. At the same time, you will be establishing a structured and repeatable design process. As we explore the design process in the upcoming chapters, we'll help you establish the knowledge and confidence you need to accomplish your design and production goals.

The next section outlines a comprehensive process for collecting and organizing your design data. If you're new to the design process, you'll find lots of useful information and ideas on how to approach a new design.

New to PCB Design?
The Design Process

Any process is easier to learn when there is an underlying structure that holds it all together. Printed circuit board design is no exception; there is a simple flow that keeps the process organized and helps to ensure design success. Once you're familiar with it, you're well on your way to creating consistent designs that can be manufactured and tested easily.

Using the many advanced design capabilities available in the PADS products, you will learn how to set up your working environment, capture your design, verify the integrity of the design, produce the physical layout, and generate all necessary manufacturing outputs. At the same time, you will be establishing a structured and repeatable design process.

As we explore the design process in the upcoming chapters, we'll help you establish the knowledge and confidence you need to accomplish your design and production goals.

The next section outlines a comprehensive process for collecting and organizing your design data. If you're new to the design process, you'll find lots of useful information and ideas on how to approach a new design.
Before beginning any project, it's a good idea to have a basic plan in place. Regardless of the complexity of your designs, you'll avoid problems and surprises with a little up-front organization. Designing a printed circuit board is a detail-oriented process that brings together information from a variety of sources. As you begin the design process, you will need to have this information available to complete each design task. Rather than interrupting your workflow and concentration each time you need another piece of information, we suggest you develop an organizational system that keeps these important details within reach at all times.

This guide offers simple suggestions to help you develop a system for organizing your design and project data. Upon completion of your first few designs, you will notice productivity gains as you develop and improve your own personalized system.

How you store your information is a matter of what works best for you. You may be working in a large organization that requires all information to exist as electronic files that can be shared with other team members. You may be an Engineer working in a research lab where you are the only person who needs access to this information. Perhaps you are a member of a small development team at a start-up company. Collecting and organizing your data is the important step; how you store it is a matter of preference.
Getting Organized

Before beginning any project, it's a good idea to have a basic plan in place. Regardless of the complexity of your designs, you'll avoid problems and surprises with a little up-front organization. Designing a printed circuit board is a detail-oriented process that brings together information from a variety of sources. As you begin the design process, you will need to have this information available to complete each design task. Rather than interrupting your workflow and concentration each time you need another piece of information, we suggest you develop an organizational system that keeps these important details within reach at all times.

How you store your information is a matter of what works best for you. You may be working in a large organization that requires all information to exist as electronic files that can be shared with other team members. You may be an Engineer working in a research lab where you are the only person who needs access to this information. Perhaps you are a member of a small development team at a start-up company. Collecting and organizing your data is the important step; how you store it is a matter of preference.

This guide offers simple suggestions to help you develop a system for organizing your design and project data. Upon completion of your first few designs, you will notice productivity gains as you develop and improve your own personalized system.
Electronic vs. Physical Storage

Time and resources typically dictate how you will organize and store your information. Many designers create a 3-ring binder for each project and divide it into sections that contain different types of information. Others create an online directory to store all project-related data. Still others use a combination of techniques. Use the method that works best for you. With high speed networks and Internet connections, much of what you need to complete your design will be available in electronic format; however, many companies impose restrictions on Internet access, and not all component information and design data will be available online.

A good starting point might be to create a project binder and keep it near your workstation. Create as much online content as possible, and keep hard copies of the most important data in the binder. This way you can have ready access to information during your design sessions and share selected data with others as needed.
Contact Data

At some point in your project you will probably need additional information or answers to project related questions. Because a contact list will save you time when you need it most and help to keep your project moving forward, create a simple form that lists all of the persons involved with the project and their contact information. You should probably include business addresses, phone numbers, and email addresses. Depending upon the size of your organization and the scope of your project, this list could include the Project Manager, Design Engineer, Mechanical Engineer, manufacturing personnel, board vendor(s), contract manufacturers, and/or customers.

Project Information

We suggest that you give each design a unique means of identification so that you can locate it easily as it moves through the design process. Individual companies typically develop a system of identification that works with their design, documentation, purchasing, and project management systems. As a minimum, your system should include the project name, board name, board description, board part number (for the unpopulated printed circuit board), assembly number (for the finished PCB assembly), and revision levels. Try to develop a system that allows for multiple versions and revisions of the boards. This will enable you to accommodate future design changes as well as alternate assembly versions of your product.
Preliminary Circuit Design
A preliminary design could be a few hand-drawn sketches, a formal schematic drawing, copies of circuit designs from component manufacturers' data sheets, or pages from application notes. If you are not the project Engineer, try to obtain as much detail about the design as feasible so that you can complete your work with as few interruptions as possible. If your organization has a standard for schematic layout and documentation, we suggest that you review the necessary requirements to make sure you have all of the information you need.

Component Data Sheets
Component data sheets provide information that you will need to complete each of your design tasks successfully. You will extract this information and place it into your design at various stages. In particular, you will need the name of the component manufacturer, part number, schematic or logic symbol, pin numbers, signal names, physical package data, and attribute data, such as voltage, tolerance, and value.

Most component data sheets are available in PDF format from manufacturers' web sites. Many multiple-page data sheets contain information that you will not need for schematic/board design, so examine each sheet carefully to determine if it contains data important to your design. A good system is to copy the PDF file onto your workstation or file server and then print and store in your binder only the pages that contain design data you need. This will make it easier to find information when you need it. Later, if you find that you need additional information, you can open the data sheets on your system to find the additional data you need.
Board Outline Drawing

To create the physical shape of a board in the design database, you need to know the physical dimensions of the board. In addition, you need to know about any cutouts, slots, mounting hole locations and the exact position of any fixed-location components, such as connectors, switches, LEDs and/or panel components; the proposed layer count; and the board thickness. Typically, this information is provided by a mechanical engineer in the form of a board outline drawing. Ideally, this drawing should be dimensioned from a mounting or tooling hole, and not from a corner of the board.

Layer Stackup

As board designs become more complex and signal integrity concerns increase, many designs incorporate multiple signal and power plane layers. A simple 4-layer design may consist of signals on the outer layers, and separate power and ground planes on the inner layers. More complex designs might contain 10, 12, or 16 or more layers arranged in complex, interleaved arrays to control signal integrity and impedance. Your designs may vary and contain a special layer stackup requirement.

If you are not the Engineer on your project, arrange a meeting early in the design phase to determine the layer stackup requirements. Also, if you have controlled impedance signals, work with your board vendor to establish how they will match their available layer materials with your manufacturing requirements. You will need all of this information to properly create your fabrication drawing(s) later in the process.
Special Footprint Requirements

Before you can transfer your schematic design to a physical board layout, every component in your design will require a footprint. Though many of the components you use in your designs will match with industry footprint standards (such as IPC-7351), other components will be unique, and you will have to construct a custom footprint for them before you can place them into your board layout.

As you gain experience and complete more designs, you will build and accumulate a library of commonly used parts. Each time you gather the data for a new design, you should check your component footprint requirements against the contents of this library. Any components that are not contained in your library will need to be constructed. Make sure you document the requirements and verify that the component data sheets that you have collected contain all the physical component dimensions that you will need to construct the footprints. Later in this guide, we'll introduce you to the Decal Wizard that you can use to quickly build many of your required footprints.

Connector Pinouts and Numbering

When you place a connector in a schematic design or a board layout, specific signals need to be connected to particular pins on those connectors. Manufacturers use different methods for numbering pins on similar connectors. Also, mating connectors may use reversed numbering schemes.
In order to prevent possible errors in pin numbering, closely examine the manufacturers' data sheets for pin numbering, and then verify the correct numbering assignments, or consult with the Project Engineer. As an additional precaution, verify that the numbering in your schematic symbols and your PCB decals are in agreement. Later, when you place your connectors into a board layout, you can use this data to label the pin numbers on the end pins of each connector to aid with pin identification during test/debug cycles or servicing.

**Design Rules and Constraints**

Design rules enable you to specify the routing requirements for your designs. The manufacturing and design processes typically dictate specific requirements, including minimum trace width, minimum spacing, recommended via sizes, and power trace widths. Your designs may also include requirements for critical signal trace routing widths as well as special rules for individual nets, including maximum length, matched length, controlled impedance, and differential routing rules.

If you are not the Engineer on your project, arrange to meet early in the process to determine the routing requirements for each type of signal used in your design. Each signal type, such as clock lines, power buses, data signal, address lines, and communications signals, can all have different and distinct requirements, so be sure to document all of the rule requirements.
Design Flow

PADS PCB Design Solutions provide a highly integrated environment in which all applications communicate information seamlessly, allowing continuous design synchronization. Changes are passed between the schematic and the PCB layout, allowing instantaneous updates in both forward and backward directions.

PADS solutions share a common interface design, enabling you to become productive in each program quickly. Data can be seamlessly passed between the design applications as well as simulation programs.

System Design
Physical Design
Analysis & Verification

- Schematic capture
- Variants and reuse
- FPGA integration

- PCB layout
- Interactive routing
- Autorouting
- Manufacturing prep

Analog simulation
Mixed A/D simulation
HDL simulation
Signal integrity analysis
Application Integration

PADS PCB Design Solutions provide a highly integrated environment in which all applications communicate information seamlessly, allowing continuous design synchronization. Changes are passed between the schematic and the PCB layout, allowing instantaneous updates in both forward and backward directions.

PADS Design Flow

System Design
- Schematic capture
- Variants and reuse
- FPGA integration

Physical Design
- PCB layout
- Interactive routing
- Autorouting
- Manufacturing prep

Analysis & Verification
- Analog simulation
- Mixed A/D simulation
- HDL simulation
- Signal integrity analysis

PAIDS solutions share a common interface design, enabling you to become productive in each program quickly. Data can be seamlessly passed between the design applications as well as simulation programs.
Design Process Overview

Taking a new design from concept to completion requires a designer to pay attention to many details. We all know that designing a printed circuit board is a continuous process of making design decisions and tradeoffs. Throughout the design process, you must weigh a number of conflicting factors and make calculated design choices in order to obtain the best possible design outputs. Knowing that your design tools have the ability to provide and manage the required content gives you a strong foundation on which to build your design. PADS PCB Design Solutions offer a fully featured front-to-back design flow that allows you to balance rules and constraints in an intelligent and predictable manner to produce quality designs that can be easily produced.

Though there are hundreds of operations you must perform to produce a final design, we can distill the process down to a few basic steps:

1. Gather the required library parts.
2. Capture the schematic.
3. Generate a netlist.
4. Construct the board outline.
5. Set up the design rules and constraints.
6. Place the components.
7. Route the board.
8. Verify the design.
9. Generate the CAM documents.

Partitioning your workflow to align with these steps will help you establish a structured approach to organizing your design tasks. It will also provide you with a number of checkpoints for reviewing your design data.
Design Process Overview

Taking a new design from concept to completion requires a designer to pay attention to many details. We all know that designing a printed circuit board is a continuous process of making design decisions and tradeoffs. Throughout the design process, you must weigh a number of conflicting factors and make calculated design choices in order to obtain the best possible design outputs.

Knowing that your design tools have the ability to provide and manage the required content gives you a strong foundation on which to build your design. PADS PCB Design Solutions offer a fully featured front-to-back design flow that allows you to balance rules and constraints in an intelligent and predictable manner to produce quality designs that can be easily produced.

Though there are hundreds of operations you must perform to produce a final design, we can distill the process down to a few basic steps:

1. Gather the required library parts
2. Capture the schematic
3. Generate a netlist
4. Construct the board outline
5. Set up the design rules & constraints
6. Place the components
7. Route the board
8. Verify the design
9. Generate the CAM documents

The Design Process

- Gather the required library parts
- Capture the schematic
- Generate a netlist
- Construct the board outline
- Set up the design rules & constraints
- Place the components
- Route the board
- Verify the design
- Generate the CAM documents

GETTING STARTED IN PCB DESIGN
PADS Environment
Welcome Screens

The PADS Welcome screens provide information and design navigation in a format that is easy to understand. From this interface you can start a new design, open an existing design, view a product tour, examine a QuickStart, take a tutorial, browse the Help system, review a Concepts Guide, or contact Technical Support.

The PADS Welcome screens are your gateway to product information and support. The Help system is your greatest resource for accurate, context-sensitive information about how to accomplish your design related tasks.
All of the PADS applications share common interface elements.
Application Links

Once you begin your design, you can easily transfer and share data between the applications. This includes cross-probing from schematic to PCB layout, ECO files, passing data to and from the autorouter, and sending data to simulation programs.

Here is an example of multiple linked applications communicating design data. A symbol is selected in the PADS Logic schematic on the left. The decal representing the part is then automatically selected in PADS Layout on the right. This feature enables you to select parts in the schematic for placement in Layout, or to select nets in the schematic to review their routing paths in Layout.

File Types and Locations

All information associated with a design is stored in the PADS database files. This means that each of the design file types (.sch for schematic and .pcb for layout) contains all of the necessary information to open and edit the files. Because no external libraries or design elements need to be included, it is easy to share or archive design databases. All design files can be output as ASCII files for editing and transfer of data to other applications.
Every design is based on the interconnection of a number of separate components into a functional product. A schematic diagram uses graphical symbolic representations of the electrical or logical functions of the components. Organizations such as ANSI and IEEE oversee various standards for the proper construction and presentation of these symbols. Typically, you will find graphical representations of the parts on the component data sheets that you collect during the organizational stage of your project. You will use these as models to construct your schematic symbols (CAE decals). In addition to the graphical symbols used in the schematic, each component must have a physical representation of the part (a PCB decal) to place onto the printed circuit board layout. The most notable standard for the construction of these footprints is the IPC-7351 standard, which recently replaced the IPC-SM-782 standard. (The reference section at the end of this guide provides more information on these standards.) The data sheets that you collect also contain the physical dimensions of the components. In addition, some may contain recommended patterns for the decals. Use the physical dimensions along with construction guidelines from your preferred standard to build the decals for your components.

The PADS libraries include thousands of common electronics components. You can use the Library Manager to browse this comprehensive collection of components. You can also use the Symbol and Decal Wizards to build new parts quickly.
Library Parts

Every design is based on the interconnection of a number of separate components into a functional product. A schematic diagram uses graphical symbolic representations of the electrical or logical functions of the components.

Organizations such as ANSI and IEEE oversee various standards for the proper construction and presentation of these symbols. Typically, you will find graphical representations of the parts on the component data sheets that you collect during the organizational stage of your project. You will use these as models to construct your schematic symbols (CAE decals).

In addition to the graphical symbols used in the schematic, each component must have a physical representation of the part (a PCB decal) to place onto the printed circuit board layout.

The most notable standard for the construction of these footprints is the IPC-7351 standard, which recently replaced the IPC-SM-782 standard. (The reference section at the end of this guide provides more information on these standards.) The data sheets that you collect also contain the physical dimensions of the components. In addition, some may contain recommended patterns for the decals. Use the physical dimensions along with construction guidelines from your preferred standard to build the decals for your components.

The PADS libraries include thousands of common electronics components. You can use the Library Manager to browse this comprehensive collection of components. You can also use the Symbol and Decal Wizards to build new parts quickly.
Electrical Properties

Electrical properties allow you to assign decals, gates, signal names, and pin numbers to a component to define its electrical behavior and connectivity in the circuit.

The Attribute Manager

You can also assign attributes such as manufacturer, part number, description, component height, value, tolerance, and cost to each component. PADS is pre-configured with many attribute types to choose from, and you can easily add additional custom attributes with the Attribute Manager.
Electrical properties allow you to assign decals, gates, signal names, and pin numbers to a component to define its electrical behavior and connectivity in the circuit. You can also assign attributes such as manufacturer, part number, description, component height, value, tolerance, and cost to each component. PADS is pre-configured with many attribute types to choose from, and you can easily add additional custom attributes with the Attribute Manager.

If your design contains parts from a variety of sources, each with differing attribute names, the Attribute Manager provides you with the capability to achieve consistency and structure by combining and renaming attribute names and values across your entire design.

To simplify the process of organizing all of this information, PADS allows you to link symbols, decals, electrical properties, and attribute data together into a common element referred to as a Part Type.
Part Types and Wizards

Each PADS library contains a collection of related Part Types, providing you with thousands of parts to choose from when creating your designs. If you can't locate the parts you need, you can use a number of tools, including the Symbol Editor, the Decal Editor, and the Part Editor, to create new symbols, decals, and part types. In addition, using the Symbol and Decal Wizards enables you to quickly build complex schematic symbols and PCB decals in minutes.

Use the Symbol Wizard to quickly build CAE decals.

PADS offers a selection of Decal Wizards to construct PCB decals quickly.
The Library Manager

The PADS Library Manager provides you with a simple, powerful tool for creating and managing your component libraries.

Learn More

For more information about how CAE decals are constructed, see the “Creating a CAE Decal” topic in the PADS Logic tutorial Creating Library Parts. For more information about how PCB decals are constructed, see the “Creating a PCB Decal” topic in the PADS Layout tutorial Creating Library Parts. To get a better understanding of Attributes, see the chapter “Working with Attributes and Labels” in the PADS Layout Concepts Guide.
The Schematic

The schematic is a graphical representation of your design that contains the connectivity, part attribute information, and design rules. From this design you can extract information in a variety of formats, such as a netlist of the connectivity, a Bill of Materials, data to send to simulators, and prints or plots of the design. Because this is an intelligent design database, it contains not just a pictorial representation of your design, but the actual interconnection of the components, as well as the component attributes such as part numbers, values, manufacturers, and tolerances.

A typical schematic viewed in PADS Logic.

The PADS Logic Schematic Editor allows you to capture your designs, including support for hierarchical blocks, quickly and efficiently. Use the PDF output option to produce intelligent PDF design files to share with engineers and vendors.
The Schematic

The schematic is a graphical representation of your design that contains the connectivity, part attribute information, and design rules. From this design you can extract information in a variety of formats, such as a netlist of the connectivity, a Bill of Materials, data to send to simulators, and prints or plots of the design. Because this is an intelligent design database, it contains not just a pictorial representation of your design, but the actual interconnection of the components, as well as the component attributes such as part numbers, values, manufacturers, and tolerances.

The PADS Logic Schematic Editor allows you to capture your designs, including support for hierarchical blocks, quickly and efficiently. Use the PDF output option to produce intelligent PDF design files to share with engineers and vendors.
Creating a Schematic Design

You create the schematic in PADS Logic by selecting a sheet size and then placing the desired component symbols on the sheet. Support for TrueType® fonts allows access to a broad selection of Engineering and Scientific notation characters as well as provides a professional typeset appearance to your schematics.

Advanced part search commands are available to help you quickly locate the desired parts in the libraries. You can use the Add Part button on the Design Toolbar to add new parts to your schematic.

PADS Logic provides a full complement of wiring tools and commands that enable you to connect individual nets or entire buses quickly and easily. You use this capability to wire the interconnection between the pins of the symbols to capture how the parts on your board must be connected.
Special symbols are available to indicate power and ground connections. Support for off-page symbols includes notation of signal names and sheet numbers for easy signal identification across multi-page schematics.

**Layer Setup**

You can set up your preferred layer stackup during the schematic design session if you want, or wait until you transfer the design to the Layout environment. PADS offers the flexibility to allow the Engineer or the Designer the ability to enter or modify these settings at any stage of the design development.

You can also specify design rules to indicate clearances, routing widths, preferred routing layers, and numerous other rules that will be observed by the interactive and automatic routers. In addition, you can set up design rules in Logic, Layout or Router so you can add or modify your rules at any convenient point in the design process. Using the ECO process, rules can be seamlessly passed between environments.
Naming Nets

You can assign names to nets and buses for identification. After wiring you can edit or change net names. Additionally, you can add or edit component attributes to define component values or part numbers. You can easily modify attributes in Logic or Layout to update your design values as your design changes. Again, these updates are accurately managed by the ECO process.

Reports

Before you begin the physical layout of your design, you can use the Report function to extract a Bill of Materials and generate a component listing for preliminary cost estimates or parts procurement.
Prints

You may also want to generate prints or plots of your schematic sheets for review. PADS supports most Windows®-compatible printers as well as a selection of popular plotters. The Preview function allows you to specify the sheets and options that you want to appear on your prints.

Learn More

For more information about adding parts to your schematic, see the PADS Logic tutorial Adding and Copying Parts. For more information about wiring your schematic, see the PADS Logic tutorial Adding Constraints. For more information about creating your prints, see the PADS Logic tutorial Printing and Plotting Schematics.
Netlist Creation
Netlists

Once you have completed and reviewed your schematic design, you can generate a netlist to transfer your design to PADS Layout and create your printed circuit board design. The netlist contains all of the connectivity and component data to represent your design on the PC board.

You can establish a direct link between applications.

You can set up a direct link between PADS Logic and PADS Layout to transfer the data, or you can save the netlist as a file and import it at a later time. If you have both applications on your workstation, you may want to use the link to transfer the netlist from the schematic to a PC board layout in one simple step.

You can save a netlist to a file for transfer to another system.

If the board is being created at a different location or by another person on a different workstation, you can output the netlist to a file and forward the file to the PCB designer.
Engineering Change Orders (ECOs)

The design database is bidirectional; information can be updated in either direction between the schematic and the board layout with Engineering Change Orders. These transfers can occur through a direct link, or you can save ECOs as files and import them later into applications when requested. Netlist comparison utilities also allow you to compare different netlists to identify updates and changes.

As your design progresses, you will probably make changes along the way. Part values, package sizes, interconnect modifications, and part substitutions all occur frequently in a typical design cycle. Maintaining and keeping this data synchronized between the schematic and the board layout is an important step in the design process. PADS provides complete forward and backward annotation capabilities allowing you to update your board design with schematic changes as well as send changes from the board design back to the schematic.

For example, if you resequence the order of your reference designators on the board, these changes can be sent back to the schematic so that you can correctly identify the parts on your schematic sheets with the updated references. You can use this same process to update attribute values, part additions/deletions and wiring changes.
You can also compare the contents of two different design files to determine any changes that have been made to either of the files. Use the Comparison settings to set up the comparison rules that you want to apply.

At some point in the life cycle of your product, you may be required to identify specific design revisions. You may have multiple schematic and/or board design files for different revisions of your design and need to identify the contents of each file. The File Comparison utility allows you to quickly and conveniently identify similarities or differences between files so that you can determine the correct course of action. This valuable resource enables you to maintain proper synchronization of your design files.

Learn More
For more information about transferring netlists, see the PADS Logic tutorial Linking to PADS Layout. For additional information about the ECO process, see the PADS Logic tutorial Updating a Schematic with Design Changes and the “ECO Process” chapter of the PADS Layout Concepts Guide.
PCB Design Setup

PADS Layout provides a number of flexible tools for constructing the board outline. Full support for slots and cutouts lets you construct complex board shapes. You can also import your board outline from mechanical design packages, such as AutoCAD, using the DXF Import utility.

The Layer Definition command enables you to specify arrangements of layer stackups that are extensive enough to satisfy your most demanding high-speed design requirements. With support for up to 250 layers, PADS Layout can accommodate the requirements of even your most demanding designs. Split-mixed power and ground planes allow you to develop complex power distribution systems to support all of your high-speed designs.

The layers in your design can be set up for specific purposes, including component mounting, routing, and power/ground planes. You can also specify preferred routing directions for each layer. This allows PADS Router to predictably place traces on each layer, evenly distributing signals while reducing crosstalk.

Use the Layers Setup to specify layer types and preferred routing direction.
PCB Design Setup

PADS Layout provides a number of flexible tools for constructing the board outline. Full support for slots and cutouts lets you construct complex board shapes. You can also import your board outline from mechanical design packages, such as AutoCAD, using the DXF Import utility.

The Layer Definition command enables you to specify arrangements of layer stackups that are extensive enough to satisfy your most demanding high-speed design requirements.

Use the Layers Setup to specify layer types and preferred routing direction.

The layers in your design can be set up for specific purposes, including component mounting, routing, and power/ground planes. You can also specify preferred routing directions for each layer. This allows PADS Router to predictably place traces on each layer, evenly distributing signals while reducing crosstalk.

With support for up to 250 layers, PADS Layout can accommodate the requirements of even your most demanding designs. Split-mixed power and ground planes allow you to develop complex power distribution systems to support all of your high-speed designs.
Options

The PADS design environment offers many design options, allowing you complete control over the setup of your design environment. You can customize settings and save them for use on future designs.

Display Colors Setup allows you to control how items on each layer appear on the screen.

Use the various Options tabs to set up your design environment and to specify your individual routing and appearance behaviors.
**Split-Mixed Planes**

Split-mixed plane capabilities allow for multi-voltage power distribution schemes, the arrangements of which you can re-specify as your design needs change. Interactive editing of plane splits allows you to adjust plane divisions at any time during the design process.

---

**Dimensioning**

Once you've finished specifying the board shape, an array of autodimensioning tools is available to dimension your board for your fabrication and assembly documentation.

---

**Learn More**

For more information about setting up your board, see the PADS Layout tutorial *Preparing a Design*. For additional information on dimensioning your board, see the PADS Layout tutorial *Adding Dimensions*. 
Placement

Estimates suggest that most designers spend more than half of their design time on component placement. This is understandable, since a well-placed board is much easier to route. Because we are familiar with the demands placed on designers during placement operations, PADS offers multiple methods for achieving high quality component placements.

After you have your board outline in place and have imported your netlist, all of the components appear in a cluster at the board origin. You can use the Disperse Components command to spread all of the components around the outside of the board perimeter in preparation for placement.
Component Placement

Estimates suggest that most designers spend more than half of their design time on component placement. This is understandable, since a well placed board is much easier to route. Because we are familiar with the demands placed on designers during placement operations, PADS offers multiple methods for achieving high quality component placements.

After you have your board outline in place and have imported your netlist, all of the components appear in a cluster at the board origin. You can use the Disperse Components command to spread all of the components around the outside of the board perimeter in preparation for placement.

PADS allows multiple methods for component placement including support for cross-probing between the schematic and the board layout. This feature allows you to select and place individual components or groups of components.
Placement with Cross-probing

You can place components in Layout or Router; where you are in the development of your design typically dictates your best choice. In either environment, you can establish a direct link with PADS Logic and cross-probe between your schematic and layout. Simply select a component in Logic, and when your cursor moves into the layout workspace, the component will be selected for placement in Layout.

Cross-probing with a Single Selection

You can also select an entire group of components in the schematic, such as an entire power supply section, and all of the parts will be selected when you move into Layout. This capability is effective for quick grouping of functional areas of a design for preliminary placement. Once the final placement is completed, you can use the cross-probing feature to highlight and review your placement decisions quickly.

Cross-probing with Multiple Selections.
You can also manually place the components one at a time, as well as in groups, using the placement tools within each application. For most designs, placement in Layout is efficient and easy. Typically, you will have an empty board on which you will build up the component placement architecture as you progress.

At times you will need to add or replace components in an existing design that includes routed traces. In this situation, the placement tools in PADS Router are your best choice. You can add new components or move existing components while preserving the routed traces, actually pushing and shoving traces and components out of your way as required.

Learn More

For more information about component placement, see the PADS Layout tutorial Moving and Placing Components, the PADS Router tutorial Placing Components, and the “Placing Parts” chapter of the PADS Router Concepts Guide.
Design Rules and Constraints

Rules and constraints are what maintain the integrity of your design. With the increasing component and routing densities of today's high-speed designs, you must manage a large number of rules and design constraints. With PADS software, you can specify separate rules for clearance and trace width for individual pin pairs, nets, groups of nets, classes, components, or the entire design if you choose. You have the flexibility to control the length of critical nets or specify matched lengths for groups of nets.

PADS PCB Design Solutions offer the capability to configure a wide selection of design rules during the design process. This allows the flexibility for the Engineer or the Designer to configure rules during capture, layout, or routing.

Default

Class

Net

Group

Pin Pair

Decal

Component

Design Rules

Hierarchy

Order of Precedence

Highest

Lowest

Levels

Accessible
to PADS

Logic
Design Rules and Constraints

Rules and constraints are what maintain the integrity of your design. With the increasing component and routing densities of today's high-speed designs, you must manage a large number of rules and design constraints. With PADS software, you can specify separate rules for clearance and trace width for individual pin pairs, nets, groups of nets, classes, components, or the entire design if you choose. You have the flexibility to control the length of critical nets or specify matched lengths for groups of nets.

PADS PCB Design Solutions offer the capability to configure a wide selection of design rules during the design process. This allows the flexibility for the Engineer or the Designer to configure rules during capture, layout, or routing.
Default Design Rules Setup

You can set up default clearance and routing rules for your design and then build a complex hierarchy of rules as you develop your routing strategy. For example, you might set wider clearances for critical nets such as clocks, or increase the trace widths for your power nets. You can specify rules for classes of nets to force certain rules for your address lines while you have different rules in effect for your data lines. These rules are applied during your interactive and autorouting sessions to assure that your design is routed with accuracy and meets your high-speed design constraints.
Net and Conditional Rules

As your design complexity increases, so can your rule assignments. In addition to the default rules, you can specify rules for individual nets or groups of nets. This gives you the flexibility to define separate rules for specific nets or net groups such as clocks or data and address buses. These rules have a higher precedence than the default rules, so assigning net rules overrides the default settings.

Conditional rules allow you to set up specific relationships between different design objects. Conditional rules can be applied to individual nets, groups of nets, classes, pin pairs, and even to individual components. You could specify that a particular class of nets must observe a clearance of 8 mils from each other, but must maintain a clearance of 25 mils from all other signals. This gives you the ability to set up exacting control over the routing of your entire design.
Routing

Whether you choose to route your board interactively, autoroute, or use a combination of both techniques, the PADS PCB Design Solutions offer you a number of different routing options. Interactive routing in PADS Layout gives you complete control over routing paths, layers, and via choices. You can select a net to route and then simply choose the desired routing path, clicking to establish each corner as you route toward your destination. You can route individual nets, or use the Bus Router to route entire buses in one operation.

In addition to the basic routing capabilities of PADS Layout, PADS Router offers advanced interactive routing features such as trace plowing, push and shove, and high-speed routing capabilities including differential pairs routing and length tuning with accordions. You can route individual nets, all nets connected to a particular component, groups of nets, or even the entire board.

The Bus Router allows simultaneous routing of entire buses. PADS Router offers multiple interactive and autorouting capabilities, allowing you complete flexibility in the routing of your design. You can easily partition your routing tasks and apply a variety of routing methods to route your board to completion.
Board Routing

Whether you choose to route your board interactively, autoroute, or use a combination of both techniques, the PADS PCB Design Solutions offer you a number of different routing options. Interactive routing in PADS Layout gives you complete control over routing paths, layers, and via choices. You can select a net to route and then simply choose the desired routing path, clicking to establish each corner as you route toward your destination. You can route individual nets, or use the Bus Router to route entire buses in one operation.

In addition to the basic routing capabilities of PADS Layout, PADS Router offers advanced interactive routing features such as trace plowing, push and shove, and high-speed routing capabilities including differential pairs routing and length tuning with accordions. You can route individual nets, all nets connected to a particular component, groups of nets, or even the entire board.

*PADS Router offers multiple interactive and autorouting capabilities, allowing you complete flexibility in the routing of your design. You can easily partition your routing tasks and apply a variety of routing methods to route your board to completion.*
High-Speed Routing with PADS Router

PADS Layout offers a selection of interactive routing methods that will allow you to accomplish many common routing tasks. However, if your designs contain high-speed signals, you will want to explore the advanced routing functionality available within PADS Router.

PADS Router offers advanced capabilities for routing high-speed traces. You can control trace length for individual nets or groups of nets. Full support for minimum/maximum matched length groups is available. Length rules that you specify will be observed by the interactive router as well as the autorouter. If you prefer to control the exact positioning of your matched length traces, you can route them interactively and monitor your progress with the Trace Length Monitor. PADS Router lets you add length to traces by interactively creating accordions.

Length tuning with accordions is accomplished with a simple click-and-drag interface that also provides a graphical Trace Length Monitor right at your cursor. There are many options available to control the amplitude and clearances between loop segments, but you can use the default settings to create accurate symmetrical accordions with a few mouse clicks.

PADS Router also allows you to specify matched lengths for groups of nets. Specify the length and tolerance for the group, and PADS Router will maintain equal lengths on all of the specified nets. You can also specify min/max lengths when you have a wider window of acceptability in your design.
Trace Length Monitor

When routing your length-controlled nets, the Trace Length Monitor gives you a visual indication of the length of routed and unrouted trace segments. This helps you to quickly determine when you have achieved the desired trace length.

**Differential Pair Accordions**

If you have differential pairs in your design, you'll appreciate the speed and accuracy you can attain by routing pairs together and even adding differential pairs accordions as you route.

*Both signals are simultaneously gathered at the proper gap*

*Trace gap is maintained throughout the routing process*

*Traces are routed together as a pair from source to the destination*
Autorouting Complex Designs

To meet your autorouting requirements, PADS Router is available to quickly and efficiently route your design. You need only to define your routing rules, select a strategy, and start PADS Router. Regardless of your design complexity, PADS Router applies an array of routing algorithms to accurately route your board to completion.

PADS Router allows you to route even your most complex designs.

Partitioning Your Routing Tasks

PADS Router offers you the ability to partition your routing tasks into highly functional individual steps. Here’s an example of how you might utilize this on a real-world design:

You begin by manually routing your critical high-speed nets such as clocks, and then protecting them. Next you can run a fanout pass on your power nets to quickly connect power and ground to your planes. Once your critical and power nets have been routed, you can fan out each of your major components individually using the fanout options. After reviewing these operations, you can route your data and address buses. Lastly, route the remaining nets to complete your design. You can also run Optimization and Centering passes as well as Test Point passes to automatically insert test points into your design.
Routing Strategies

PADS Router Options allow you complete control over the routing process. You can select the order and intensity of each routing pass, thereby allowing yourself the flexibility to customize your strategy based upon your particular design needs. You can route individual components, nets, or your entire design.

PADS Router utilizes a series of sophisticated route algorithms to maximize speed and accuracy when routing your designs. Once you familiarize yourself with the many routing pass options, you will quickly develop your own preferred strategy setups that are closely matched to your design needs. As you gain experience with PADS Router, you will find it to be a valuable design asset.

Learn More

For more information about routing your board, see the PADS Layout tutorial Creating and Editing Traces, the PADS Router tutorial Creating Traces with Interactive Router, the PADS Router tutorial Creating High-Speed Traces with Interactive Router, and the “Routing” chapter of the PADS Router Concepts Guide.
Before you send your design out for manufacture, you will want to check the accuracy of the final design. PADS offers a large selection of checks that you can initiate, including clearance checks, plane connectivity, trace widths, component body spacing, acid traps, soldermask slivers, and numerous other verification checks. PADS Layout and PADS Router each offer a wide range of checks.

The Design Verification checks in the PADS products ensure that your designs will meet your most stringent manufacturing requirements. Checking your designs prior to sending them to the fabricator saves you time and keeps you on schedule.

The Verify Design options in PADS Layout allow you to perform clearance and connectivity checks on your design to assure both that proper clearances exist between design objects, and that all signals are connected. If you have plane layers in your design, you should run a plane check to make sure that all of your power and ground signals are properly connected to the plane layers. Depending upon your design complexity, additional checks are available for high-speed signals, test points, and fabrication.

Many checks are available to verify that your design is correct.
Design Verification

Before you send your design out for manufacture, you will want to check the accuracy of the final design. PADS offers a large selection of checks that you can initiate, including clearance checks, plane connectivity, trace widths, component body spacing, acid traps, soldermask slivers, and numerous other verification checks. PADS Layout and PADS Router each offer a wide range of checks.

The Verify Design options in PADS Layout allow you to perform clearance and connectivity checks on your design to assure both that proper clearances exist between design objects, and that all signals are connected. If you have plane layers in your design, you should run a plane check to make sure that all of your power and ground signals are properly connected to the plane layers.

Depending upon your design complexity, additional checks are available for high-speed signals, test points, and fabrication.

The Design Verification checks in the PADS products ensure that your designs will meet your most stringent manufacturing requirements. Checking your designs prior to sending them to the fabricator saves you time and keeps you on schedule.
Design Verification in PADS Router

If you have completed your design using PADS Router, you may want to take advantage of the advanced design verification checks available. PADS Router can check your differential pairs, min/max routes, and many other high-speed routing rules. Setup is quick and easy with the Design Verification options.

In PADS Router, errors can be viewed on-screen and in a spreadsheet view or a report where you can examine and determine your course of action for each item. Verifying your design before it goes to the fabricator helps to reduce manufacturing problems and keeps your project on schedule.

In this example, while routing certain differential pair signals, the designer chose to override some of the clearance rules to route escape traces near a connector. A Design Verification clearance check shows that there are multiple violations in the design. By selecting one of the errors in the spreadsheet view, the system will zoom in on the exact location of the error so that you can examine the error condition. This example shows that the traces are too close together. The error messages contain hyperlinks that will open a report containing specific information related to each individual error.
Once you have examined the conditions related to the error, you can decide your best course of action. After you correct the error, run the check again to verify that the condition has been resolved.

**Troubleshooting Design Verification Errors in the Spreadsheet View**

This level of sophisticated error reporting takes the guesswork out of checking your designs, saving you time and assuring that your vendor will receive an error-free design file.

Learn More

For more information about verifying your design, see the PADS Layout tutorial *Checking for Design Rules Violations* and the PADS Router tutorial *Checking for Design Rules Violations*. 
Generate the CAM Documents

After your design has been captured, routed, and checked, it's time to create the manufacturing outputs. Generating the Computer Automated Manufacturing (CAM) files is the last step in the process. PADS PCB Design Solutions support the creation of the full set of manufacturing outputs, including prints and plots, photoplots (Gerber files), drill files, pick and place data, soldermasks, paste masks, fabrication drawings, and assembly drawings.

The advanced Computer Automated Manufacturing (CAM) file generation capabilities in PADS Layout allow you to quickly produce all of the files and documents needed to manufacture your design. CAM setups can be saved and reused on future design projects.

Selected document

Document type
Fabrication Layer

Summary of items contained in document

Preview button

Editing of a print document for the bottom layer of a design. You can generate prints or plots of your design layers and choose the individual design objects that you want to include in each of the files. You can also scale and rotate the prints to fit your available printing devices.
Generate the CAM Documents

After your design has been captured, routed, and checked, it's time to create the manufacturing outputs. Generating the Computer Automated Manufacturing (CAM) files is the last step in the process. PADS PCB Design Solutions support the creation of the full set of manufacturing outputs, including prints and plots, photoplots (Gerber files), drill files, pick and place data, soldermasks, paste masks, fabrication drawings, and assembly drawings.

Selected document

Document type

Fabrication Layer

Summary of items contained in document

Preview button

Editing of a print document for the bottom layer of a design.

You can generate prints or plots of your design layers and choose the individual design objects that you want to include in each of the files. You can also scale and rotate the prints to fit your available printing devices.

The advanced Computer Automated Manufacturing (CAM) file generation capabilities in PADS Layout allow you to quickly produce all of the files and documents needed to manufacture your design. CAM setups can be saved and reused on future design projects.
Once your manufacturing preferences are defined, you can save them to apply to future designs. The Preview feature allows you to visually inspect your outputs before generating the final files and sending them out for manufacture. When you are ready, all of the required documents can be generated when you press the Run button. This kind of process automation can save you valuable time on every design you create.
Once your manufacturing preferences are defined, you can save them to apply to future designs. The Preview feature allows you to visually inspect your outputs before generating the final files and sending them out for manufacture. When you are ready, all of the required documents can be generated when you press the Run button. This kind of process automation can save you valuable time on every design you create.

CAM Setup

Specifying the CAM Options

You can define any number of output files for printing, pen plotting, photoplotting, fabrication or assembly.

CAM DOCUMENTS

Use the Run button to generate the selected output files

Specify each item that you want to appear in the selected output file

CAM Preview

Use CAM Preview to inspect your outputs.

What’s Next?

Now that you've organized your information and taken a brief look at the design flow, you're ready to take things to the next step. We've only begun to illustrate the advanced capabilities available with PADS PCB Design Solutions; there is much more for you to discover.

Remember, this document only introduces you to the PADS design process. It is not a comprehensive user guide. Your most complete source of detailed feature information is the Help in each application. In addition, use the other resources available from the Welcome screens, including QuickStarts, Product Tours, Tutorials, Concepts Guides, and links to web content such as technical documents and PADS Support.

Congratulations. You're well on your way to designing your first PADS project with confidence and certainty.

Learn More

For more information about setting up and generating your manufacturing files, see the PADS Layout tutorial Creating Reports and CAM Files and the “CAM and CAM Plus” chapter of the PADS Layout Concepts Guide.
References
QuickStarts
PADS Logic QuickStart
PADS Layout QuickStart
PADS Router Interactive Routing QuickStart
PADS Router Autorouting QuickStart

Concepts Guides
PADS Layout Concepts Guide
PADS Router Concepts Guide

PADS Logic Tutorials
Learning the User Interface
Creating Library Parts
Adding and Copying Parts
Adding Connections
Adding Buses
Editing Design Elements
Assigning Constraints
Creating Reports
Linking to PADS Layout
Updating a Schematic with Design Changes
Printing and Plotting Schematics
Managing Multiple Sheets

PADS Layout Tutorials
Learning the User Interface
Creating Library Parts
Preparing a Design
Importing Design Data
Importing Schematic Data
Assigning Constraints
Moving and Placing Components
Creating and Editing Traces
Creating Traces with Dynamic Route
Autorouting with PADS Router
Creating Split Planes
Creating Copper and Pour Areas
Adding Dimensioning
Checking for Design Rule Violations
Updating the Schematic
Creating Reports and CAM Files
PADS Layout Advanced Tutorials

Placing Components Using Unions and Radial Move
Copying Circuits Using Design Reuse
Checking for Trace Length Violations
Linking and Embedding Objects (OLE)
Creating Assembly Options
Moving and Placing Components

PADS Router Tutorials

Learning the User Interface
Assigning Constraints
Preparing a Design
Placing Components
Creating Traces with Interactive Routing
Creating High-speed Traces with Interactive Routing
Autorouting with PADS Router
Checking for Design Rule Violations

Library Resources


Industry Resources

IPC-7351 Generic Requirements for Surface Mount Design and Land Pattern Standard (footprints)

Available from:
IPC -- Association Connecting Electronics Industries®
3000 Lakeside Drive, Suite 309 S
Bannockburn, IL
www.ipc.org

Technical Papers


Support

http://www.mentor.com/supportnet/pcb.cfm

PADS Home Page

www.mentor.com/pads
North American Support Center
1-800-547-4303

For support locations outside of the US, please refer to the following website:

http://www.mentor.com/supportnet/support_offices.html