Converting MicroSim® PCBoards Designs to OrCAD Layout® Designs

Quick Start

First edition 30 November 1998

Technical Support	(503) 671-9400
Corporate offices	(503) 671-9500
OrCAD Japan K.K.	81-45-621-1911
OrCAD UK Ltd.	44-1256-381-400
Fax	(503) 671-9501
General email	info@orcad.com
Technical Support email	techsupport@orcad.com
World Wide Web	http://www.orcad.com
OrCAD Design Network (ODN)	http://www.orcad.com/odn

OrCAD®

9300 SW Nimbus Ave.
Beaverton, OR 97008 USA

Contents

Before you begin

Welcome to OrCAD

OrCAD® offers a total solution for your core design tasks: schematic- and VHDL-based design entry; FPGA and CPLD design synthesis; digital, analog, and mixed-signal simulation; and printed circuit board layout. What's more, OrCAD's products are a suite of applications built around an engineer's design flow—not just a collection of independently developed point tools. The Schematics-to-Capture translator is just one element in OrCAD's total solution design flow.

OrCAD PCBoards-to-Layout translator overview

The PCBoards-to-Layout translator converts PCB designs created in MicroSim PCBoards to OrCAD Layout designs. You can translate PCB design databases created in any version of MicroSim PCBoards and prepare them for use within OrCAD Layout.

The descriptions and explanations in this manual are written from the perspective of a PCB designer who is familiar with PCBoards but not necessarily experienced using Layout. If you have never worked with OrCAD Layout before, you should spend some time stepping through the tutorials and reviewing the *Getting Started* manual. Together these will explain the basic workings of Layout and help you become productive quickly.

You may also wish to take formal training from OrCAD to become fully proficient with Layout. Courses are offered throughout the year at various sites. Check the OrCAD Web site at www.orcad.com for the latest training schedule and registration information.

How to use this guide

This guide is designed so you can quickly find the information you need to use PSpice A/D. To help you learn and use PSpice A/D efficiently, this manual is organized by tasks, beginning with simulation basics and design entry (parts one and two), and moving on to more advanced concepts (parts three and four).

Symbols and conventions

OrCAD printed documentation uses a few special symbols and conventions.

Notation	Examples	Description
Ctrl+R	Press Ctrl+R	Means to hold down the Ctrl key while pressing R.
Alt, F, O	From the File menu, choose Open (Alt, F, O)	Means that you have two options. You can use the mouse to choose the Open command from the File menu, or you can press each of the keys in parentheses in order: first Alt, then F, then O.
Monospace font	In the Part Name text box, type PARAM.	Text that you type is shown in monospace font. In the example, you type the characters P, A, R, A, and M.
	`.MODEL MLOAD NMOS` `+ (LEVEL=1 VTO=0.7 CJ=0.02pF)`	Examples of syntax, netlist output, and source code are displayed in monospace font. The example shows an example of the syntax for the PSpice .MODEL statement.
UPPERCASE	In Capture, open CLIPPERA.DSN.	Path and filenames are shown in uppercase. In the example, you open the design file named CLIPPERA.DSN.
Italics	In Capture, save *design_name*.DSN.	Information that you are to provide is shown in italics. In the example, you save the design with a name of your choice, but it must have an extension of .DSN.

Related documentation

In addition to this guide, you can find technical product information in the online help, the online interactive tutorial, online books, OrCAD's technical web site, as well as other books. The table below describes the types of technical documentation provided with PSpice A/D.

This documentation component . . .	Provides this . . .
This guide— OrCAD PSpice A/D & Basics User's Guide	Basic information to get started in PSpice A/D. The *OrCAD PSpice A/D & Basics User's Guide* is an overview of the features available in PSpice A/D and PSpice A/D Basics.
Online help	Comprehensive information about PSpice A/D. If you can't find something in the *PSpice A/D & Basics User's Guide*, look in the online help.
	You can access help from the Help menu in PSpice A/D, by choosing the Help button in a dialog box, or by pressing F1 . Topics include:
	• Explanations and instructions for common tasks.
	• Descriptions of menu commands, dialog boxes, tools on the toolbar and tool palettes, and the status bar.
	• Netlist format samples, error messages, and glossary terms.
	• Reference information.
	• Product support information.
	You can get context-sensitive help for a error message by placing your cursor in the error message line in the session log and pressing F1 .
Online interactive tutorial	A series of self-paced interactive lessons. You can practice what you've learned by going through the tutorial's specially designed exercises that interact directly with Capture. You can start the tutorial by choosing Learning Capture from the Help menu.
Online *OrCAD PSpice A/D Reference Manual*	Reference material for PSpice A/D. Also included: detailed descriptions of the simulation controls and analysis specifications, start-up option definitions, and a list of device types in the analog and digital model libraries. User interface commands are provided to instruct you on each of the screen commands.
Online *OrCAD PSpice Library List*	A complete list of the analog and digital parts in the model and symbol libraries.
Online *OrCAD PSpice A/D User's Guide*	An online, searchable version of this guide.

Online *PSpice A/D Quick Reference Card*	Concise descriptions of the commands, shortcuts, and tools available in PSpice A/D.
ODN—OrCAD Design Network www.orcad.com/odn	An internet-based technical support solution. ODN provides a variety of options for receiving and accessing design and technical information. ODN provides:

- A Knowledge Base with thousands of answers to questions on topics ranging from schematic design entry and VHDL-based programmable logic design to printed circuit board layout methodologies.

- A Knowledge Exchange forum for you to exchange information, ideas, and dialog with OrCAD users and technical experts from around the world. A list of new postings appears each time you visit the Knowledge Exchange, for a quick update of what's new since your last visit.

- Tech Tips that deliver up-to-the-minute product information in your email box. Stay informed about the latest advances, tips, and announcements on your OrCAD product.

- Online technical support via the Tech Support Connection. Use this service to submit technical support incidents online. Create submissions, upload files, track your incidents and add comments directly into OrCAD's support database.

Translating designs

1

Overview

This chapter explains how to set up and use the PCBoards-to-Layout translator to convert MicroSim PCBoards designs into OrCAD Layout Release 9 designs.

The following sections are included:

About the translator

The PCBoards-to-Layout translator converts designs (`.PCA` files) created in MicroSim PCBoards to design databases (`.MAX` files) that can be read by OrCAD Layout 9. You can translate PCB designs created in any version of MicroSim PCBoards. Footprint and padstack libraries from PCBoards cannot be translated for use with Layout.

Translated circuit board designs contain all of the essential information in the original databases. There is no loss of accuracy in the translation process and all the basic design information is carried across. However, because of the inherent differences between the two layout editors, you may need to perform some additional editing and setup work in Layout to prepare the translated design for further use. See *Translation caveats* for descriptions of the most common situations you may need to deal with. Also, see **Chapter 2,** Differences between PCBoards and Layout for explanations of some of the major functional differences between MicroSim PCBoards and OrCAD Layout.

Setting up and running the translator

There are two ways you can start the translator: from within Layout or in stand-alone mode.

To start the translator from within Layout

1   Start Layout.

2   In LSession, from the File menu, choose Import.

3   Select PCBoards to Layout.
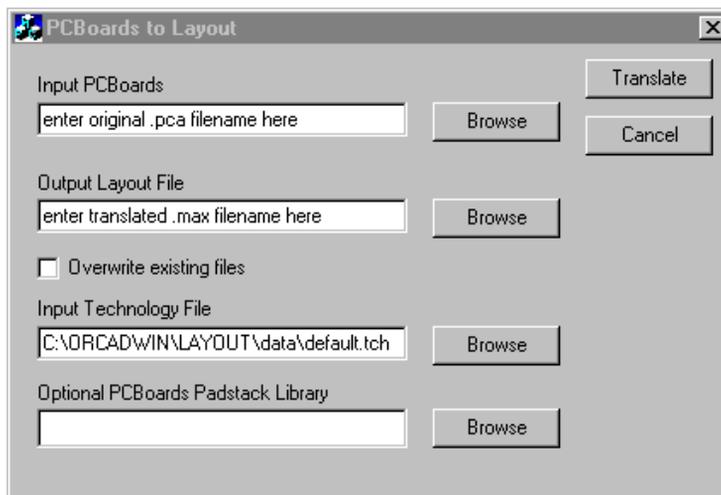
Use the Browse buttons to find files quickly and easily.

4   Enter or select the filename of the original PCBoards design (`.PCA`) in the Input PCBoards box.

5   Enter the filename and path for the translated design file in the Output Layout File box.

6   Enter or select the desired technology file in the Input Technology File box.

To select the appropriate Technology file, see the explanations of the file contents in *Appendix A: Understanding the files used with Layout* in the OrCAD *Layout User's Guide.*

7   If required, enter or select the filename and path for an alternate padstack library in the Optional PCBoards Padstack Library box.

8   Click Translate to run the translation process.

If you were using a padstack library in PCBoards other than the default library, you should enter the alternate padstack library filename and path in the Optional PCBoards Padstack Library box. This will ensure that the correct padstack definitions are translated into the new Layout database. Otherwise the default padstack library (std.PSL) will be used.

To start the translator in stand-alone mode

1   In Windows Explorer, double-click frpcb.EXE in c:\orcadwin\layout.

2   Enter the desired filenames and select the appropriate technology file, as described above.

3   Click Translate to run the translation process.

To open the translated file in Layout

1   Start Layout if it is not already loaded.

2   In LSession, from the File menu, choose Open.

3   Enter or select the filename of the translated design.

Translation caveats

Because there are fundamental differences between MicroSim PCBoards and OrCAD Layout, certain design data cannot be translated and carried over completely from one program to the other. Although every attempt has been made to assure that the PCBoards-to-Layout translator provides reliable and correct conversion of the design information, you should review and inspect the translated design layer-by-layer to be sure that the Layout database meets your final design requirements.

The following explanations will help you troubleshoot some of the more likely problem areas that may occur in translated designs.

Board Outline

In Layout, the board outline is used to define the board edge clearance as well as the outer perimeter of the board. The board edge clearance is half the width of the board outline, measured from the centerline inwards. When you translate a design from PCBoards, you must edit the board outline in Layout and make it wider in order to define an adequate board edge clearance. For example, if the board outline is 10mils wide after translation and you want a 50mils clearance, you should change the width to 100mils.

To change the width of the board outline:

1   Select Obstacle from the Tool menu.

2   Choose Select Tool.

3   Shift + Left-click the board outline.

4   Right-click and choose Properties.

5   Enter the desired width in the Width box.

6   Click OK.

7   Right-click and choose End Command.

Design rules

To understand more about Technology Files, see *Using Technology, Template & Strategy files* in this manual and *Appendix A: Understanding the files used with Layout* in the OrCAD *Layout User's Guide.*

The various design rules attached to nets for routing width, clearance, etc. are defined by the Strategy and Technology files used in Layout when opening a design file. During the translation process from PCBoards, you must specify a particular Technology file to be used. Depending on what rules were previously defined in PCBoards and how closely the Technology file corresponds to them, there could be significant differences in the new design.

Before proceeding with any further design work on the translated file, you should run a DRC check to identify any violations that may exist. You should also review and verify that all of the final routing and design rules are correct in the translated Layout database.

To investigate and correct DRC errors:

1   Select Design Rule Check from the Auto menu.

2   Check the appropriate boxes for the types of DRC rules to be applied.

3   Click OK.

4   Zoom in on a small region of the board using the Zoom In toolbar button.

5   Select Database Spreadsheets from the View menu (or click the Spreadsheets toolbar button).

6   Choose Error Markers.

7   The spreadsheet describes the types of violations that have been detected. Double-click on a particular cell to pan the screen to the location of that error.

8   Make the appropriate edits to correct the violation.

Note   *DRC markers can be removed by selecting the Remove Violations command from the Auto menu.*

To verify the routing and technology rules:

1   Select Database Spreadsheets from the View menu (or click the Spreadsheets toolbar button).

2   Choose Nets.

3   Review and verify the track width assignments and routing parameters for each net.

4   Select Database Spreadsheets from the View menu (or click the Spreadsheets toolbar button).

5   Choose Layers.

6   Review and verify the routing parameters for the layers and the overall stackup arrangement.

To load new Technology and/or Strategy files:

1   Select Load from the File menu.

2   Choose Template (*.TCH, *.TPL) from the pull-down list in the Files of type box.

3   Choose the appropriate new Technology file to be used in place of the current one in the design.

4   Click Open.

5   Repeat steps 1 - 4 for replacing the Strategy file.

Differences between PCBoards and Layout

2

Overview

This chapter explains some of the major differences between MicroSim PCBoards and OrCAD Layout. It contains the following sections:

- Starting applications with LSession on page 2-20

- Using Technology, Template & Strategy files on page 2-22

- Selecting and editing objects on page 2-24

- Understanding Obstacles on page 2-26

- Working with spreadsheets on page 2-27

- Autorouting with SmartRoute on page 2-29

Also, for more detailed assistance with these topics, you should consult the appropriate sections in the *OrCAD Layout User's Guide.*

Starting applications with LSession

When you start Layout, the first screen you see is referred to as LSession. This serves several functions. Primarily, LSession provides a gateway for launching the Layout editor, as well as the other design tools that are included with OrCAD Layout such as Library Manager, Visual CADD, SmartRoute, GerbTool, and the various database translators. LSession also provides access to the Layout Tutorials that guide you through the basic features and capabilities of Layout.

LSession is always active in its own window running in the background along with the other applications. You can return to LSession at any time and keep several applications active simultaneously.

To start a new design

1   Select New from the File menu. This launches Layout.

2   Choose the appropriate technology file (.TCH).

3   Choose the desired netlist file (.MNL).

4   Enter the name of the new design file (.MAX).

5   Click Save.

To start other applications

• Select the particular application from the Tools menu.

To start the database translators

1   Select Import or Export from the File menu.

2   Choose the particular translation you wish to execute.

3   Enter the appropriate source and destination file
    names in the subsequent dialog(s).

To start the Layout Tutorials

1   Select Learning Layout from the Help menu.

2   Click Lesson Menu.

3   Choose the particular lesson or exercise you wish to
    study.

Using Technology, Template & Strategy files

Technology, Template and Strategy files are used by Layout to define board and stackup information, placement and routing rules, and other design parameters such as grid spacing, color settings, etc. These files are configured specifically to optimize the placement and routing processes. If used in conjunction with the automated functions in Layout, they can reduce design time and improve the overall manufacturability of the circuit board.

Technology files

When you create a new design in Layout, you must specify the Technology file (`.TCH`) to be used. This file can contain virtually all aspects of a PCB design except for netlist information. There are a number of predefined Technology files supplied with Layout that cover a broad range of circuit board types. If you are unsure of which file to use, select the default to begin with; you can always load a different Technology file later. You can also create your own custom Technology files that can be saved and reused with other designs.

Template files

See the *OrCAD Layout Footprint Libraries* manual for images of the board outlines contained in the Template files supplied with Layout.

Template files (`.TPL`) are similar to Technology files but they typically contain only board outline information and basic design rules. A Template file contains the minimum amount of data required to begin a new board design. As with Technology files, Template files can be customized and saved for later reuse.

Strategy files

Strategy files (.sf) contain placement and routing rules. These are used by the autoplacement and autorouting features in Layout to produce high yield, high reliability PCB designs. The standard Strategy file is automatically loaded with each new board design; you do not have to tell Layout to load this. Each of the other Strategy files contains specific rules for particular types of board design. For instance, the file 8_SM2_H.sf has optimal autorouting rules for an eight layer board with surface mount components on both sides, and a horizontal routing bias on the top layer. Strategy files can be loaded at any time to change or update your placement or routing rules. They can also be customized and saved for later reuse.

To load a new Strategy or Technology file

1   Select Load from the File menu.

2   Choose Template (*.TCH, *.TPL) or Strategy (*.sf) from the pull-down list in the Files of type box.

3   Choose the desired file.

4   Click Open.

Selecting and editing objects

There are significant differences between MicroSim PCBoards and OrCAD Layout relating to how design objects are selected and edited.

Tool modes for selecting objects

In PCBoards, you can select any object at any time with a left-click of the mouse (provided the appropriate selection filter settings are enabled). By double-clicking on an object you can access the attribute dialog directly.

In Layout, you must first enable the selection of a particular type of object before it can be selected and edited. In other words, you must be in the appropriate editing or Tool mode in order to select a particular object and then edit its properties. For example, to select a component, you must be in the Component mode. To select a track, you must be in the Track mode. The particular mode you are in is indicated in the title bar of the editing window. You enable particular selection and editing modes using the Tool menu, or by clicking the appropriate toolbar button. By double-clicking on most objects you can access the property dialog directly. You can exit a particular mode by clicking the right mouse button (see *Right mouse button support*).

Right mouse button support

PCBoards does not support the use of the right mouse button. Clicking the right mouse button in Layout brings up context-sensitive menus that provide additional functionality related to the particular Tool that is currently enabled. Right-mouse support typically provides a means of ending a particular Tool mode (End Command), accessing the Properties dialog, or executing routine

editing functions such as Cut, Copy, Paste, Undo, etc. Becoming proficient using the right-mouse button is essential to being productive and effective in using Layout.

Understanding Obstacles

In Layout, the term "obstacle" refers to a variety of design objects that have the common characteristic of being obstructions or impediments to the ability to place parts or to route connections. The following is a list of the design objects that are considered obstacles in Layout:

- Anti-copper
- Board Outline
- Comp group keepin
- Comp group keepout
- Comp height keepin
- Comp height keepout
- Copper area
- Copper pour
- Detail
- Free track
- Insertion outline
- Place outline
- Route keepout
- Route-via keepout
- Via keepout

As this list reveals, obstacles are fundamental aspects of any design. To define or edit them, select Obstacle from the Tool menu. You can obtain additional information about the properties of obstacles in your design by examining the Obstacle spreadsheet (see *Working with spreadsheets*).

Working with spreadsheets

Another major difference between MicroSim PCBoards and OrCAD Layout is the way in which design rules, properties of objects and netlist information, etc. are manipulated. In Layout, this information is stored and edited in the form of spreadsheets. Spreadsheets provide a common format for displaying and editing related data. Becoming proficient with the use of spreadsheets will greatly enhance your productivity.

The following database spreadsheets are used in Layout:

- Components
- Packages
- Footprints
- Padstacks
- Apertures
- Layers
- Nets
- Obstacles
- Text
- Error Markets
- Drills
- Post Process
- Statistics

To access spreadsheets

1   Select Database Spreadsheets from the View menu, or click on the Spreadsheet toolbar button.

2   Choose the desired spreadsheet.

3   Make whatever edits are necessary to the spreadsheet cells.

4   The information you modify will be saved automatically when you close the spreadsheet.

Many of the spreadsheets listed here can also be prepared as ASCII format report files.

To create reports

1   Select Create Reports from the Auto menu.

2   Check the desired reports to be generated.

3   Click the appropriate buttons for viewing or saving the reports as files.

4   Click OK.

Autorouting with SmartRoute

MicroSim PCBoards includes the Cadence/CCT SPECCTRA autorouter. OrCAD Layout Plus provides the proprietary autorouter SmartRoute. Both autorouters produce high quality autorouted designs with speed and reliability. SmartRoute offers the following capabilities not found in SPECCTRA:

- true diagonal routing in opposing directions on different layers

- neural network based routing algorithms for improved routing performance and pre-route analysis of estimated routing time and completion success

- full integration with *OrCAD Layout Plus*

To become fully proficient with an autorouter, you must gain experience using it on a variety of PCB designs with different types of routing constraints. The more you work with SmartRoute, the better you will be at defining successful placement and routing strategies for future designs.

To start SmartRoute

1   Save and close the design file if it is still open in Layout.

2   From LSession, select SmartRoute from the Tools menu.

3   From SmartRoute, select Open from the File menu.

4   Choose the design file (.MAX) to be autorouted.

5   Click Open.

6   Select AutoRoute Board from the Auto menu to begin routing.

To read in a design autorouted in SmartRoute

1   Save and close the design file if it is still open in SmartRoute.

2   From LSession, select Open from the File menu.

3   Choose the design file.

4   Click Open.

For more detailed information about using SmartRoute, use the online Help feature or see the *SmartRoute User's Guide.*

Command reference

A

Overview

This appendix provides a reference for the main menu commands used in MicroSim PCBoards and lists the equivalent commands in OrCAD Layout 9.

It contains the following section:

-

Commands

Note   *Many of the Layout menu commands listed here can be accessed directly using the appropriate shortcut keys. See the OrCAD Layout User's Guide for a full description of the shortcut keys.*

Table 1

This PCBoards command...	Is similar to this Layout command...
File menu	
New	File, New
Open	File, Open
Close	File, Close
Save	File, Save
Save As	File, Save As
Job Setup	File, Print/Plot
Print	File, Print/Plot
Photoplot	Options, Gerber Settings & Post Process Settings & View, Database Spreadsheets, Apertures & File, Print/Plot
NC Drill	Auto, Create Reports, Drills
Export	LSession: File, Export, MAX to DXF
Import	LSession: File, Import, DXF to MAX
Netlist	View, Database Spreadsheets, Nets
Reports	Auto, Create Reports
View Messages	(no equivalent)

Table 1

This PCBoards command...	Is similar to this Layout command...
Edit menu	
Undo	Edit, Undo
Redo	(no equivalent)
Cut	(no equivalent)
Copy	Edit, Copy
Paste	Edit, Paste
Delete	Edit, Delete
Find	Edit, Find/Goto
Move Delta	(no equivalent)
Move by Ref Des	Edit, Select Any
Rotate	Tool, Component, Select Tool, left-click, right-click, Rotate
Flip Sides	Tool, Component, Select Tool, left-click, right-click, Opposite
Mirror	Tool, Text, Select Tool, left-click, right-click, Mirror
Disconnect	Tool, Connection, Disconnect Pin
Attributes	Tool, Component, Select Tool, left-click, right-click, Properties
Draw menu	
Repeat	most commands repeat until right-click "end command" is executed
Arc	(no equivalent)

Table 1

For more information about obstacles, see
Chapter 2 - Understanding Obstacles.

This PCBoards command...	Is similar to this Layout command...
Circle	Tool, Obstacle, New, right-click, Properties, select "Detail", right-click, Arc, left-click, drag
Line	Tool, Obstacle, New, right-click, Properties, select "Detail"
Polyline	Tool, Obstacle, New, right-click, Properties, select "Detail"
Rectangle	Tool, Obstacle, New, right-click, Properties, select "Detail", right-click, Arc, left-click hold and drag
Text	Tool, Text, Select Tool, right-click, New
More Graphics	(no equivalent)
Component	Tool, Component, Select Tool, right-click, New
Connection	Tool, Connection, Select Tool, right-click, Add
Trace	Tool, Track, Select Tool
Hole	Tool, Component, New, choose "Footprint", select library SHEET23, select desired mounting hole
Via	Tool, Via, New
Areafill	Tool, Obstacle, New, right-click, Properties, select "Copper pour" or "Copper area"

Table 1

This PCBoards command...	Is similar to this Layout command...
Void	Tool, Obstacle, New, right-click, Properties, select "Anti-copper"
Keepout	Tool, Obstacle, New, right-click, Properties, select "Route keepout"
Board Signal Keepin	Tool, Obstacle, New, right-click, Properties, select "Board outline"
View menu	
Redraw	View, Redraw
Fit	View, Zoom All (Fit)
In	View, Zoom In
Out	View, Zoom Out
Previous	View, Zoom Previous
Area	View, Zoom In or Out, left-click twice
Pan-New Center	View, Zoom Center
Highlight Net(s)	View, Database Spreadsheets, Nets, left-click, right-click, Change Color
Clear Highlights	View, Database Spreadsheets, Nets, left-click, right-click, Change Color
Toolbar	(no equivalent)
Combo Bar	(no equivalent)
Status Bar	(no equivalent)

Table 1

This PCBoards command...	Is similar to this Layout command...
Configure menu	
Styles	(no equivalent)
Snap Grid	Options, System Settings
Trace Placement Mode	Options, Route Settings
Mfg Minimum Rules	(no equivalent)
Layers	View, Database Spreadsheets, Layers & Options, Colors
Layer Display	Options, Colors
Padstacks	View, Database Spreadsheets, Padstacks
Drill Symbols	(no equivalent)
Ratsnest display	View, Database Spreadsheets, Nets
Selection Filter	Tools, various selection tools for different objects
Tools menu	
Compare to Netlist	Auto, Back Annotate
Browse Forward ECO Log	(no equivalent)
Check Component Footprints	(no equivalent)
Run Schematics	(no equivalent)
CCT: Setup	(no equivalent)
CCT: Net Rules	(no equivalent)
CCT: Edit Do File	(no equivalent)
CCT: Autoroute	Auto, Autoroute or LSession: Tools, SmartRoute

For more information about spreadsheets, see *Chapter 2 - Working with spreadsheets.*

Table 1

This PCBoards command...	Is similar to this Layout command...
CCT: Read Routes	(no equivalent)
Reannotate	Options, Components Renaming and Auto, Rename Components
DRC	Auto, Design Rule Check
Optimize Rats	Tool, Component, Select Tool, left-click, right-click, Minimize Connections
Measure	Tool, Measurement, Select Tool
Edit PADS Mapping File	(no equivalent)
Cross Probe Schematic	(no equivalent)
Options	Options, System Settings & User Preferences
Library menu	
Setup	File, Library Manager, Add or Remove
Packaging	(no equivalent)
Update Packaging	(no equivalent)
Footprint Editor	File, Library Manager
Update Footprints	(no equivalent)
Load Padstacks	(no equivalent)
Export Padstacks	View, Database Spreadsheets, Padstacks, right-click, Save to Library
Window menu	
Cascade	Window, Cascade
Tile Horizontal	Window, Tile

Table 1

This PCBoards command...	Is similar to this Layout command...
Tile Vertical	Window, Tile
Arrange Icons	(no equivalent)
Help menu	
Search for Help On	Help, Help Topics
Keyboard Shortcuts	(no equivalent)
User's Guides	(no equivalent)
Technical Support	(no equivalent)
Using Help	(no equivalent)
About PCBoards	Help, About Layout

Index