

Allegro® PCB Editor

Version 15.2

Training Manual

Book 2

July 21, 2004



© 1990-2004 Cadence Design Systems, Inc. All rights reserved.
Printed in the United States of America.

Cadence Design Systems, Inc., 555 River Oaks Parkway, San Jose, CA 95134, USA

Cadence Trademarks

1st Silicon Success®	First Encounter®	QPlace®
Allegro®	FormalCheck®	Quest®
Assura™	HDL-ICE®	SeismIC™
BlackTie®	Incisive™	SignalStorm®
BuildGates®	IP Gallery™	Silicon Design Chain™
Cadence® (brand and logo)	Nano Encounter™	Silicon Ensemble®
CeltIC™	NanoRoute™	SoC Encounter™
ClockStorm®	NC-Verilog®	SourceLink® online customer support
CoBALT™	OpenBook® online documentation library	SPECCTRA®
Concept®	Orcad®	SPECCTRAQuest®
Conformal®	Orcad Capture®	Spectre®
Connections®	Orcad Layout®	TtME®
Design Foundry®	PaciflC™	UltraSim®
Diva®	Palladium™	Verifault-XL®
Dracula®	Pearl®	Verilog®
Encounter™	PowerSuite™	Virtuoso®
Fire & Ice®	PSpice®	VoltageStorm®

Other Trademarks

All other trademarks are the exclusive property of their respective owners.

Confidentiality Notice

No part of this publication may be reproduced in whole or in part by any means (including photocopying or storage in an information storage/retrieval system) or transmitted in any form or by any means without prior written permission from Cadence Design Systems, Inc. (Cadence).

Information in this document is subject to change without notice and does not represent a commitment on the part of Cadence. The information contained herein is the proprietary and confidential information of Cadence or its licensors, and is supplied subject to, and may be used only by Cadence's customer in accordance with, a written agreement between Cadence and its customer. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

RESTRICTED RIGHTS LEGEND Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013.

UNPUBLISHED This document contains unpublished confidential information and is not to be disclosed or used except as authorized by written contract with Cadence. Rights reserved under the copyright laws of the United States.

Contents

Lesson 1:	Customizing PCB Editor to Increase Productivity	1-1
	Course Directory Structure	1-2
	PCB Editor Database Tool - DBDoctor	1-3
	PCB Editor Initialization	1-6
	The PCB Editor Environment.....	1-8
	Global Environment File - 1	1-9
	Environment File Variables.....	1-13
	Default Aliases for Function and Control Keys	1-14
	Labs	1-19
	Lab 1-1: Managing Your User Environment.....	1-20
	Lab 1-2: Customizing the PCB Editor User Interface	1-29
	Answers to Search Path Quiz	1-31
Lesson 2:	Accessing Information to Enhance Productivity with PCB Editor	2-1
	CDSDoc - Cadence Online Documentation	2-2
	What is SourceLink?	2-4
	Other Web Resources	2-10
	Live and Archived Webinars	2-12
	Labs	2-13
	Lab 2-1: Starting the Online Documentation.....	2-14
	Lab 2-2: Opening Documents.....	2-16
	Lab 2-3: Using the Command Bar	2-17
	Lab 2-4: Searching Documents.....	2-18
	Lab 2-5: Reporting Problems (optional lab).....	2-21
Lesson 3:	Constraint Manager.....	3-1
	Design Process Issues	3-2
	The Constraint Manager	3-3
	Design Flow Using the Constraint Manager	3-7
	The Constraint Manager Main Window	3-8
	Creating an Electrical Constraint Set (ECSet).....	3-10
	Assigning the ECSet to Objects.....	3-12
	Labs	3-15
	Lab 3-1: Constraint Manager Tour	3-16
	Lab 3-2: Using the Constraint Manager	3-20
Lesson 4:	Advanced Constraints, Critical Routes, and Constraint Areas.....	4-1
	Constraint System Master - Standard Values	4-2
	Steps for Defining Extended Design Rules	4-3
	Labs	4-9
	Lab 4-1: Viewing Existing Rules Previously Defined in a Board Design.....	4-10
	Lab 4-2: Routing Address Lines on Specified Subclasses	4-17
	Design Constraints.....	4-22
	ECSet - The Electrical Constraints Set - Modes.....	4-23
	Nets and Xnets.....	4-27
	Constraint Areas	4-28

Labs.....	4-32
Lab 4-3: Autorouting Critical Nets with Propagation Delay Rule Definitions	4-33
Lab 4-4: Automatically Routing the Extended Design Rules.....	4-38
Lab 4-5: Defining Extended Design Rules for Areas	4-46
Lesson 5: Differential Pairs	5-1
Differential Pair Basics.....	5-2
Define a Diff Pair ECSet in PCB Editor.....	5-7
Interactively Routing Diff Pairs.....	5-11
Labs.....	5-15
Lab 5-1: Differential Pair Setup in PCB Editor	5-16
Defining Diff Pairs in Constraint Manager (Which nets are special?).....	5-23
Differential Impedance Calculator	5-27
Different Rules for Different Layers	5-28
Labs.....	5-29
Lab 5-2: Defining Diff Pairs using the Constraint Manager.....	5-30
Lab 5-3: Route Diff Pair Connections in PCB Router.....	5-34
Lesson 6: Technology Files	6-1
Reasons for Using Technology Files.....	6-2
Labs.....	6-6
Lab 6-1: Using Technology Files	6-7
Lesson 7: Automatic Placement Tools	7-1
Prerequisites for Placement	7-2
Accessing PCB Router Autoplacement.....	7-3
Interactive Placement Hints.....	7-5
Lab	7-6
Lab 7-1: Automatic Placement	7-7
Lesson 8: Testpoint Generation	8-1
How Testprep Works.....	8-4
Setting Testprep Properties.....	8-5
Setting Testprep Parameters	8-7
Generate Testpoints	8-13
General Considerations.....	8-14
The Testprep Log File	8-15
Lab	8-16
Lab 8-1: Test Preparation.....	8-17
Altering Testpoints	8-25
Fixing Testpoints, Creating a Fixture	8-26
Generating a Drill File for the Test Fixture	8-27
Lab	8-28
Lab 8-2: Interactively Working with Testpoints.....	8-29
Lesson 9: Interactive Etch Editing.....	9-1
Reviewing Basic Techniques.....	9-2
Group Routing	9-9
Tips for Interactive Routing.....	9-10
Lab	9-12
Lab 9-1: Adding and Editing Etch	9-13
Color Visibility Views.....	9-24

Displaying Disconnected Pins on VCC and GND	9-26
Labs	9-27
Lab 9-2: Changing Color Visibility for Routing	9-28
Lab 9-3: Viewing Disconnected Voltage Pins.....	9-31
Controlling Line Length	9-36
Delay Tuning	9-37
Etch Length Readout on Non-Critical Traces	9-38
Display Parasitics on a Connect Line	9-39
Z Axis Delay.....	9-40
Pin Delay	9-41
Lab	9-42
Lab 9-4: Controlling Line Length.....	9-43
High-Speed Etch Editing.....	9-51
Logical Net Editing	9-54
Lab	9-55
Lab 9-5: Rat T's.....	9-56
Lesson 10: Design Check for the PCB Router	10-1
Router and Alignment Checks.....	10-2
Accessing PCB Router - Mainstream Model	10-5
Additional Resources.....	10-9
Labs	10-11
Lab 10-1: Running the Router Precheck	10-12
Lab 10-2: Clues to the Router Precheck Lab.....	10-13
Lab 10-3: Answers to the Route PreCheck Lab	10-18
Lesson 11: Glossing.....	11-1
Preparing for Automatic Glossing.....	11-1
The Glossing Controller	11-3
Labs	11-14
Lab 11-1: Fixing Critical Nets before Glossing	11-15
Lab 11-2: Using Gloss for Sequential Cleanup	11-17
Labs	11-21
Lab 11-3: Adding and Removing Fillets	11-22
Lab 11-4: Custom Smoothing.....	11-24
Lesson 12: Copper Planes.....	12-1
Split Planes	12-2
Labs	12-4
Lab 12-1: Split Plane Using Anti-Etch Subclass.....	12-5
Lab 12-2: Split Plane Using Add Polygon	12-9
Complex Planes	12-20
Negative Plane Island Check.....	12-23
Labs	12-24
Lab 12-3: Complex Planes.....	12-25
Lab 12-4: Viewing Negative Shapes and Negative Plane Island Check.....	12-28
Appendix A: File Extensions	A-1