Allegro® PCB Editor

Version 15.2

Training Manual **Book 1** **July 21, 2004**

cadence

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

Cadence Trademarks

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

Other Trademarks

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

Confidentiality Notice

Contents

Lesson 1: Allegro PCB Editor User Interface 1-1
Primary Allegro PCB Editor Programs 1-2
Other Programs 1-3
Cadence SPB Tools 1-4
Course Directory Structure 1-5
Setting and Changing Your Working Directory 1-6
Allegro PCB Editor and Workspace 1-7
Toolbar 1-8
Control Panel and World View Window 1-9
Getting Help 1-10
Lab 1-11
Lab 1-1: Allegro PCB Editor Tour 1-12
Mouse Buttons 1-18
Controlling the Window Display 1-19
Navigating in the World View Window 1-21
Zoom Control Using the Middle Mouse Button 1-22
Default Aliases for Function and Control Keys 1-23
Running Commands with Strokes 1-24
Controlling the Toolbars 1-25
Drawing Parameters 1-26
Drawing Options: Display and Status 1-27
Drawing Options: Line Lock and Symbol 1-29
User Preferences 1-31
Lab 1-32
Lab 1-2: Navigating the PCB Editor User Interface 1-33

Lesson 2: Managing the PCB Editor Work Environment 2-1
Groups, Classes and Subclasses 2-2
More Groups, Classes and Subclasses 2-3
Options Window of the Control Panel 2-4
Controlling Color and Visibility 2-5
Controlling Etch Visibility 2-6
Graphics Dimming or Shadow Mode 2-7
Scripts 2-8
Lab 2-9
Lab 2-1: Script Files and Controlling Visibility and Color 2-10
The Selectable Objects List 2-16
Using the Find by Name Section 2-17
Using Find by Property 2-18
Highlighting Elements 2-19
Using the Show Element Command 2-20
Using the Display Measure Command 2-21
Labs 2-22
Lab 2-2: Highlighting and Using the Find Filter 2-23
Lab 2-3: Using the Find Filter with the Show Element Command 2-29

Lesson 3: Padstacks 3-1
Anatomy of a Padstack 3-2
Padstack Details 3-3
What is a Thermal Relief? 3-4
Flash Symbols 3-5
Lab 3-6
Lab 3-1: Creating a Flash Symbol 3-7
What Does the Padstack Designer Do? 3-10
Padstack Designer - Parameters 3-11
Padstack Designer - Layers 3-13
Defining Pad Shapes/Sizes 3-14
Adding/Deleting/Copying Layers 3-15
Saving the Padstack 3-16
Labs 3-17
Lab 3-2: Creating Padstacks for a Through-Hole Pin Device 3-18
Lab 3-3: Creating a Padstack for a Surface- Mounted Device 3-27

Lesson 4: Component Symbols 4-1
Package Symbol Wizard 4-1
Drawing Parameters Form 4-2
Drawing Origin 4-3
Moving the Drawing Origin 4-4
PCB Editor Symbol Types 4-5
Example: a 14-pin DIP Package 4-6
Adding Pins 4-7
Drawing Component Outlines 4-9
Adding Labels 4-10
Defining Area Constraints 4-11
Saving Symbol Files 4-12
Labs 4-13
Lab 4-1: Creating a DIP16 Package Using the Package Symbol Wizard 4-14
Lab 4-2: Creating a DIP14 Package Symbol 4-18
Lab 4-3: Creating an SOIC16 with the Symbol Editor (*optional lab*) 4-29

Lesson 5: Board Design Files 5-1
Creating a Board Symbol 5-2
Typical Board Outline 5-3
Drawing a Board Outline 5-4
Tooling/Mounting Holes 5-5
Chamfers 5-6
Linear Dimensioning 5-7
Defining Constraint Areas (Keepins/Keepouts) 5-8
Saving Board Symbol Files (.bsm and .dra) 5-9
Board Wizard 5-10
Labs 5-11
Lab 5-1: Creating a Board Mechanical Symbol 5-12
Lab 5-2: Creating a Board using the Board Wizard 5-30
Allegro PCB Editors - Overview 5-33
Creating a Master Design File 5-34
Defining Layer Stackup 5-35
Lab 5-36

Lab 5-3: Creating a Master Design File (.brd) 5-37

Lesson 6: Importing Logic Information into Allegro PCB Editor 6-1
Design Layout Process 6-2
Design Entry HDL-Integrated Logic Design with Physical Layout 6-3
Transfer Files (pst*.dat) 6-4
Importing Logic into PCB Editor from Design Entry HDL 6-6
Importing Logic Data 6-7
Engineering Changes—Placement 6-8
Importing Electrical Constraints 6-9
Engineering Changes—Routing 6-10
Schematic-Driven Layout 6-11
Design Entry CIS-Integrated Logic Design with Physical Layout 6-13
Design Entry CIS Interface with PCB Editor 6-14
Design Entry CIS-PCB Editor Logic Import 6-15
Third-Party Logic Import 6-16
Netlist Format 6-17
General Rules for Netlists 6-18
Device Files 6-19
Package Properties in Device Files 6-20
Loading a Third-Party Netlist 6-21
Netin Checking 6-22
Guidelines for Importing Logical Data 6-23
Labs 6-24
Lab 6-1: Design Entry HDL to PCB Editor 6-25
Lab 6-2: Design Entry CIS to PCB Editor 6-30
Lab 6-3: Importing a Third-Party Netlist 6-33

Lesson 7: Setting Design Constraints 7-1
Design Layout Process 7-2
Introduction to Design Rules 7-3
Setting Up Design Rules 7-4
Standard (default) Design Rules 7-4
Extended Design Rules 7-5
Spacing Rule Set—Step 1 7-6
Spacing Rule Set—Step 2 7-7
Spacing Rule Set—Setting Rules by Layer 7-8
Spacing Rule Set—Step 3 7-9
Labs 7-10
Lab 7-1: Standard Design Rules 7-11
Lab 7-2: Extended Design Rules—Spacing 7-13
Physical Rule Set—Step 1 7-18
Physical Rule Set—Step 2 7-19
Physical Rule Set—Etch by Layer 7-20
Physical Rule Set—Step 3 7-21
Lab 7-22
Lab 7-3: Extended Design Rules—Physical 7-23
Design Constraints 7-27
Property Assignments and Changes 7-28
DRC Marker Display 7-29
Lab 7-30

Lab 7-4: Working with Properties 7-31

Lesson 8: Component Placement 8-1
Design Layout Process 8-2
Prerequisites 8-3
Interactive Placement 8-4
Placement Grid 8-5
Strategy 8-6
Floorplanning with Rooms 8-7
Creating a Room 8-8
Assign RefDes Command 8-9
Labs 8-10
Lab 8-1: Floorplanning 8-11
Lab 8-2: Assigning Preplaced Packages 8-16
Placement-Related Properties 8-17
Placement Commands 8-18
Manual Placement Commands 8-19
Changing the Default Orientation 8-20
Lab 8-21
Lab 8-3: Manual Placement 8-22
Quickplace 8-28
Deleting Components 8-30
Labs 8-31
Lab 8-4: Using Quickplace 8-32
Lab 8-5: Removing Components from the Board 8-40

Lesson 9: Advanced Placement 9-1
Ratsnest 9-2
Automatic Swapping of Functions and Pins 9-3
Automatic Swap 9-4
Running Automatic Swap 9-5
Interactive Swap 9-6
Selecting Alternate Packages 9-7
Updating Symbols in a Design 9-9
Updating Padstacks 9-10
Modifying Padstacks 9-12
Creating a Library from a Design 9-13
Cross Placement with DE HDL 9-14
Cross Selection with DE CIS 9-15
Cross Highlighting between PCB Editor and DE CIS 9-16
Labs 9-17
Lab 9-1: Displaying Ratsnests 9-18
Lab 9-2: Swapping Components, Pins, and Functions 9-20
Lab 9-3: Advanced Placement with ALT_SYMBOL (Optional) 9-23
Lab 9-4: Using the DE HDL Schematic for Manual Placement (Optional) 9-26
Lab 9-5: Using the DE CIS Schematic for Manual Placement (Optional) 9-30

Lesson 10: Routing and Glossing 10-1
Design Layout Process 10-2
Accessing Interactive Routing Modes 10-3
Routing Grids: Fixed 10-4

Routing Grids: Variable 10-5
Finding Nets or Reviewing Rats 10-6
Adding Signal Connections 10-7
Inserting Vias 10-8
Selecting Via Types 10-9
Define Blind/Buried Via 10-10
Auto Define Blind/Buried Via 10-11
Pop-Up Menu Options 10-13
Options Form 10-14
Options Form—Smooth 10-15
Options Form—Bubble/Gridless 10-16
Labs 10-17
Lab 10-1: Defining Etch Grids 10-19
Lab 10-2: Adding and Deleting Connect Lines and Vias 10-21
Accessing the PCB Router 10-28
Autoroute Prerequisites 10-28
Preparing for Automatic Routing 10-29
The PCB Editor-PCB Router Process 10-30
Labs 10-31
Lab 10-3: Preparing for Autorouting 10-32
Lab 10-4: Using the PCB Router 10-37
Editing Existing Etch 10-40
Moving Etch with the Slide Option 10-41
Editing Vertices 10-42
Changing the Layer or Width of a Connection 10-43
Deleting Etch 10-44
Using the Cut Option 10-45
Interactive Routing Properties 10-46
Glossing the Design 10-47
Labs 10-48
Lab 10-5: Checking for Unconnected Pins 10-49
Lab 10-6: Improving Routed Connections 10-51
Lab 10-7: Replacing Etch and Using the Cut Option 10-54
Lab 10-8: Running Gloss 10-57

Lesson 11: Copper Areas and Positive or Negative Planes 11-1
Design Layout Process 11-2
Copper Area Images 11-3
Adding a Copper Area 11-4
Global Dynamic Parameters - Shape Fill 11-6
Global Dynamic Parameters - Void Controls 11-7
Global Dynamic Parameters - Clearances 11-8
Global Dynamic Parameters - Thermal Relief Connects 11-9
Adding Copper Shapes 11-10
Editing Copper Shapes 11-11
Lab 11-12
Lab 11-1: Copper Areas 11-13

Lesson 12: Preparing for Post Processing 12-1
Design Layout Process 12-2
Renaming Reference Designators 12-3

Rename Reference Designators (Main Form)....12-4
Rename Reference Designators (Setup Form)....12-6
Rename Reference Designators—Key Points....12-7
Backannotation....12-8
Backannotation Examples....12-9
Backannotation—DE HDL Export Netlist....12-10
Backannotation to DE HDL....12-11
Property Backannotation....12-12
DE CIS Integrated Logic Design/Physical Layout....12-13
PCB Editor-DE CIS Backannotation....12-14
Third-Party Backannotation Process....12-15
Third-Party Backannotation....12-16
Labs....12-17
Lab 12-1: Renaming Components....12-18
Lab 12-2:PCB Editor to DE HDL Backannotation....12-22
Lab 12-3:PCB Editor to DE CIS Backannotation....12-24
Lab 12-4:PCB Editor Backannotation to a Third-Party Schematic DE CIS Tool....12-25

Lesson 13: Preparing the Board Design for Manufacturing....13-1
Design Layout Process....13-2
Creating Silkscreens....13-3
Creating Silkscreens—Menu....13-4
Incremental Update of Silkscreens....13-5
Generating Reports....13-6
Labs....13-7
Lab 13-1: Creating Silkscreens....13-8
Lab 13-2: Creating Reports....13-11
Creating Checkplots....13-12
Generating Artwork....13-13
Artwork Parameters....13-14
The Aperture File....13-16
Film Control....13-17
Film Options....13-18
Adding a Photoplot Outline....13-20
Generating Gerber Files....13-21
Viewing Gerber Files....13-22
Labs....13-22
Lab 13-3: Creating Artwork Files....13-24
Lab 13-4: Viewing Gerber Files....13-31
Creating Fabrication Drawings....13-33
Drill Symbols and Legend Table....13-34
Drill Customization Spreadsheet....13-35
Generating an NC Drill File....13-36
Creating the Parameters File....13-37
Creating Assembly Drawings....13-39
Labs....13-40
Lab 13-5: Creating a Drill Legend....13-41
Lab 13-6: Creating Fab and Assembly Drawings....13-43
Lab 13-7: Creating an NCDRILL File....13-45

Appendix A: File Extensions....A-1