cādence™

Allegro®
PCB Editor Tutorial

Product Version 16.0
June 2007

Contents

Preface . . . 9

Purpose of This Tutorial . . . 9
Audience . . . 10
How to Use This Tutorial . . . 10
  Understanding the Sample Design Files . . . 11
  Understanding Multimedia Demonstrations . . . 11
Tutorial Flow . . . 12
Related Information . . . 16
Syntax Conventions . . . 17

About Allegro PCB Editor . . . 19

Allegro PCB Editor Tools . . . 19
Allegro PCB Editor Initialization . . . 21
env File . . . 22
Cadence File Types . . . 23
Allegro PCB Editor Database . . . 26
Operating System Differences . . . 26
Requirements for a New Design . . . 27
  Allegro Design Entry HDL . . . 27
  Front-end Integration . . . 29
  Back-end Integration . . . 30
  Third-Party Netlist . . . 32
Allegro PCB Editor Flow . . . 36
  Menu Items and Corresponding Commands . . . 41
Sources of Information . . . 42
  SourceLink . . . 42
  Cadence Customer Response Center . . . 43
  Education Services . . . 43

1

Module 1: Getting Started with Allegro PCB Editor 45

Lesson 1-1: Creating a Project Directory 45
- Overview 45
- Procedure 48
- Summary 49
- For More Information 49
- What's Next 49

Lesson 1-2: Starting Up Allegro PCB Editor 49
- Overview 49
- Procedure 50
- Summary 51
- For More Information 51
- What's Next 52

Lesson 1-3: Setting Your Working Directory and Opening a Design 52
- Overview 52
- Procedure 52
- Summary 54
- For More Information 54
- What's Next 54

Lesson 1-4: Accessing the Help System 54
- Overview 54
- Demo 56
- Procedure 56
- Procedure 57
- Procedure 59
- Summary 60
- What's Next 60

2

Module 2: Introducing the Allegro PCB Editor User Interface. 61

Lesson 2-1: Identifying Parts of the User Interface 61
- Overview 61

Working with Foldable Windows 66
Procedure 71
Summary 73
For More Information 73
What's Next 74
Lesson 2-2: Accessing Pop-up Menus and Panning a Design 74
Overview 74
Procedure 74
Summary 76
For More Information 76
What's Next 77
Lesson 2-3: Zooming In and Out of a Design 77
Overview 77
Procedure 77
Summary 78
For More Information 79
What's Next 79
Lesson 2-4: Using Other Methods to Zoom In and Out of a Design 79
Overview 79
Procedure 79
Summary 81
For More Information 81
What's Next 82

3
Module 3: Using Allegro PCB Editor Control Functions 83
Lesson 3-1: Changing the Cursor Display 83
Overview 83
Procedure 84
Summary 85
For More Information 85
What's Next 86
Lesson 3-2: Controlling Color and Visibility 86
Overview 86
Procedures 89

- Assigning Colors to Subclasses . . . 91
- Summary . . . 97
- For More Information . . . 97
- What's Next . . . 97

Lesson 3-3: Controlling Etch Visibility . . . 97
- Overview . . . 97
- Procedure . . . 98
- Summary . . . 99
- For More Information . . . 99
- What's Next . . . 99

Lesson 3-4: Controlling Colors and Dimming Graphics . . . 100
- Overview . . . 100
- Procedures . . . 101
- Summary . . . 103
- For More Information . . . 104
- What's Next . . . 104

Lesson 3-5: Using the Control Panel to Manipulate Design Objects . . . 104
- Overview . . . 104
- Procedures . . . 106
- Summary . . . 112
- For More Information . . . 112
- What's Next . . . 112

Lesson 3-6: Highlighting Objects . . . 113
- Overview . . . 113
- Procedure . . . 113
- Summary . . . 115
- For More Information . . . 115
- What's Next . . . 115

Lesson 3-7: Listing Detailed Information About a Specified Object . . . 115
- Overview . . . 115
- Procedure . . . 116
- Summary . . . 118
- For More Information . . . 118
- What's Next . . . 118

Lesson 3-8: Measuring Distance Between Objects . . . 118
- Overview . . . 118

Procedure 119
Summary 120
For More Information 121
What's Next 121

4
Module 4: Using Allegro PCB Editor Design Editing Functions
123
Lesson 4-1: Naming a Symbol and Setting Drawing Parameters 124
Overview 124
Procedure 124
Summary 127
For More Information 127
What's Next 128
Lesson 4-2: Setting the Grid for a Design 128
Procedure 128
Summary 129
For More Information 130
What's Next 130
Lesson 4-3: Creating a Board Outline 130
Overview 130
Procedure 130
Summary 133
For More Information 133
What's Next 134
Lesson 4-4: Choosing Drawing Options 134
Overview 134
Procedure 134
Summary 139
For More Information 139
What's Next 139
Lesson 4-5: Defining the Stackup 139
Overview 139
Procedure 140
Summary 143

For More Information 143
What's Next 144
Lesson 4-6: Associating Design Objects with Classes and Subclasses 144
Overview 144
Procedure 144
Summary 145
For More Information 146
What's Next 146
Lesson 4-7: Adding Arcs to a Design 146
Overview 146
Procedure 146
Summary 147
For More Information 147
What's Next 147
Lesson 4-8: Adding Circles to a Design 148
Procedure 148
Summary 149
For More Information 149
What's Next 149
Lesson 4-9: Adding Text to a Design 149
Overview 149
Procedure 149
Summary 151
For More Information 151
What's Next 151
Lesson 4-10: Using Zcopy 151
Overview 151
Procedure 151
Summary 153
For More Information 153
What's Next 153

5
Module 5: Customizing the Environment 155
Lesson 5-1: Customizing Your View and Toolset 155

- Overview . . . 155
- Procedure . . . 156
- Summary . . . 158
- For More Information . . . 159
- What's Next . . . 159

Lesson 5-2: Defining Aliases and Function Aliases . . . 159
- Overview . . . 159
- Procedure . . . 160
- Summary . . . 161
- For More Information . . . 161
- What's Next . . . 162

Lesson 5-3: Setting Environment Variables . . . 162
- Overview . . . 162
- Procedure . . . 162
- Summary . . . 164
- For More Information . . . 164
- What's Next . . . 164

Lesson 5-4: Running Commands with Strokes . . . 164
- Overview . . . 164
- Procedure . . . 165
- Summary . . . 167
- For More Information . . . 167
- What's Next . . . 167

Lesson 5-5: Scripting . . . 167
- Overview . . . 167
- Procedure . . . 168
- Procedure . . . 170
- Procedure . . . 171
- Summary . . . 172
- For More Information . . . 172
- What's Next . . . 172

Lesson 5-6: Using Color Visibility Views . . . 172
- Overview . . . 172
- Procedure . . . 173
- Summary . . . 174

Preface

This preface discusses the following topics:

- Purpose of This Tutorial on page 9
- Audience on page 10
- How to Use This Tutorial on page 10
- Tutorial Flow on page 12
- Related Information on page 16
- Syntax Conventions on page 17

Purpose of This Tutorial

The Allegro PCB Editor tutorial is designed to be used as a common tutorial document for Allegro PCB Editor, OrCAD PCB Editor, and APD. Except where noted, any specific mention of Allegro PCB Editor, Allegro PCB Editor commands, or tutorial instructions are applicable to OrCAD PCB Editor as well.

The Allegro PCB Editor Tutorial provides lessons and a sample design file to help you learn how to work with Allegro PCB Editor and APD.

Important

> This tutorial shows you how to work with Allegro PCB Editor and APD in menu-driven editing mode, or verb/noun use model. Ensure that you work with the tutorial in menu-driven editing mode, using *Setup –Application Mode – None* (`noappmode` command) to exit from a current application mode and return to menu-driven editing mode.

The goal of this tutorial is to acquaint you with the Allegro PCB Editor and APD environment and some of its basic functions. The tutorial does not cover the process of developing a printed board design.

This tutorial is based on Release 16.0. However, when appropriate, the procedures include steps that you can use if you are running a later version. You should expect to spend approximately sixteen hours to complete the lessons in this tutorial.

The tutorial contains these modules:

- Module 1: Getting Started with Allegro PCB Editor
- Module 2: Introducing the Allegro PCB Editor User Interface
- Module 3: Using Allegro PCB Editor Control Functions
- Module 4: Using Allegro PCB Editor Design Editing Functions
- Module 5: Customizing the Environment

For details on the modules, see the Tutorial Flow on page 12.

Audience

This tutorial is intended to train new users who have design experience with other tools but are unfamiliar with Allegro PCB Editor and APD. It also serves to refresh veteran users who access the tools infrequently. To work successfully with the editor, you must have a basic knowledge of printed circuit board (PCB) design.

How to Use This Tutorial

The training is offered in these learning modes:

- Written lessons provide detailed procedures for performing basic operations.
- Sample design files offer a starting point for practicing with the tools.

The written lessons and sample designs reinforce your learning experience. Use them in the way that is most comfortable and efficient for you to learn the fundamentals of working with Allegro PCB Editor.

Understanding the Sample Design Files

The sample design files, `cds_routed.brd` and `cds_routed_DRC.brd` are basic printed circuit boards that you can load and use to begin working with Allegro PCB Editor. The tutorial uses these same design files to illustrate the procedures. You can work with the design files as you progress through the lessons.

To locate the files, see the `<`*`installation_directory`*`>\doc\algrotutorial \examples directory.`

Understanding Multimedia Demonstrations

Some tutorial lessons may be accompanied by multimedia demonstrations, which offer visual ways to grasp concepts and techniques that are described in the procedures. The demonstrations support and illustrate the procedures.

You can launch a multimedia demonstration by clicking on the hyperlink in the Demo section that precedes the procedure for each lesson, if available.

Getting the Flash Player

To view the multimedia demonstrations, when available, you need to install the appropriate Macromedia Flash Player on your system. Adobe Flash Players for different operating systems are free and available at:

`http://www.macromedia.com/shockwave/download/alternates/`

Important

The multimedia demonstrations included in this tutorial do not run on the IBM AIX operating system. However, the tutorials have been designed to be used without viewing the demonstrations. You can follow procedures in the lessons while you perform the tasks using the sample designs.

Viewing a Multimedia Demonstration

To see how to control a Flash multimedia demonstration, click on this link to view a demonstration:

Controlling a Flash Multimedia Demonstration

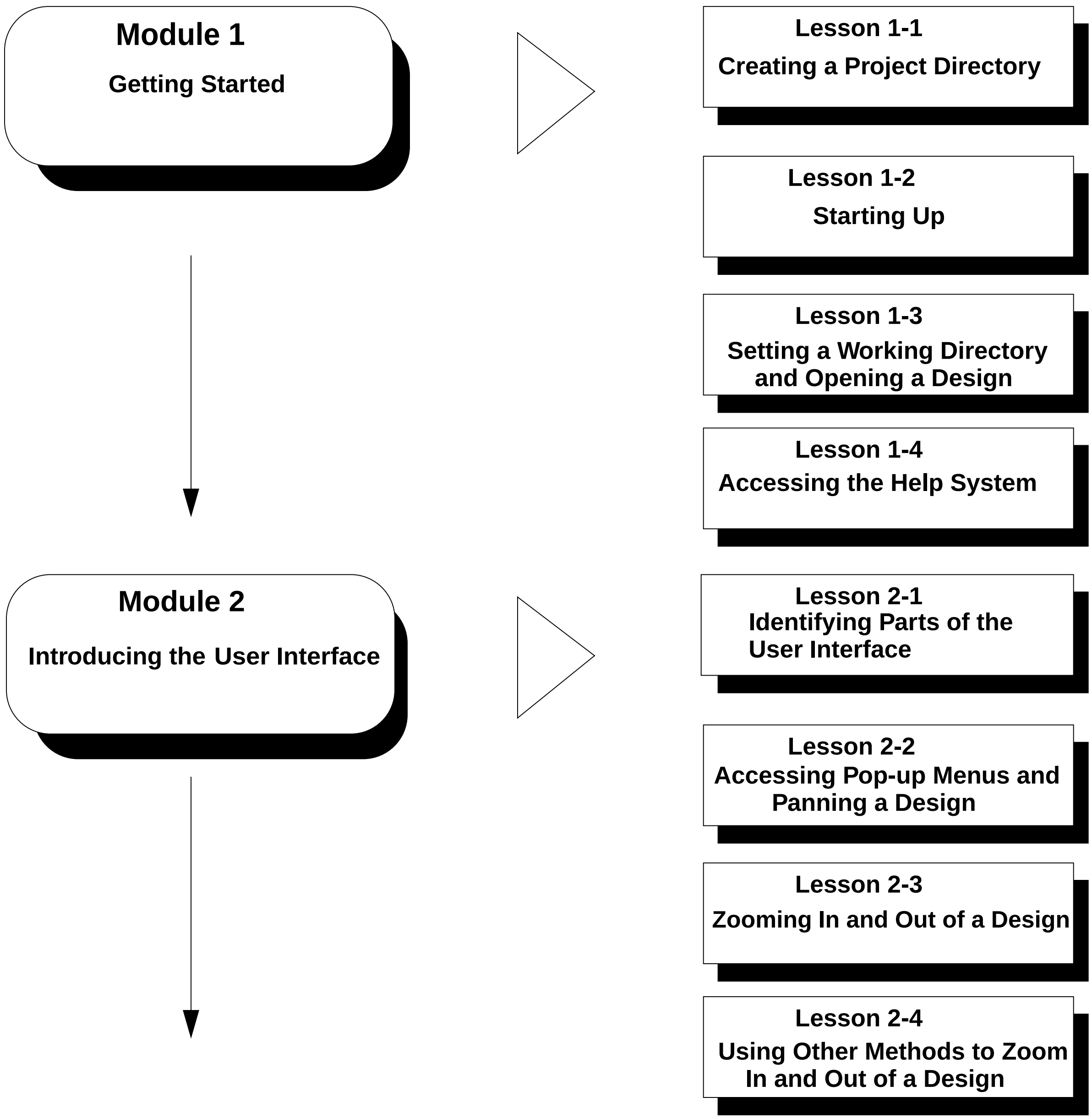
Depending on the demonstration, audio may be included. If a multimedia demonstration contains audio, the link launching shows these logos: .

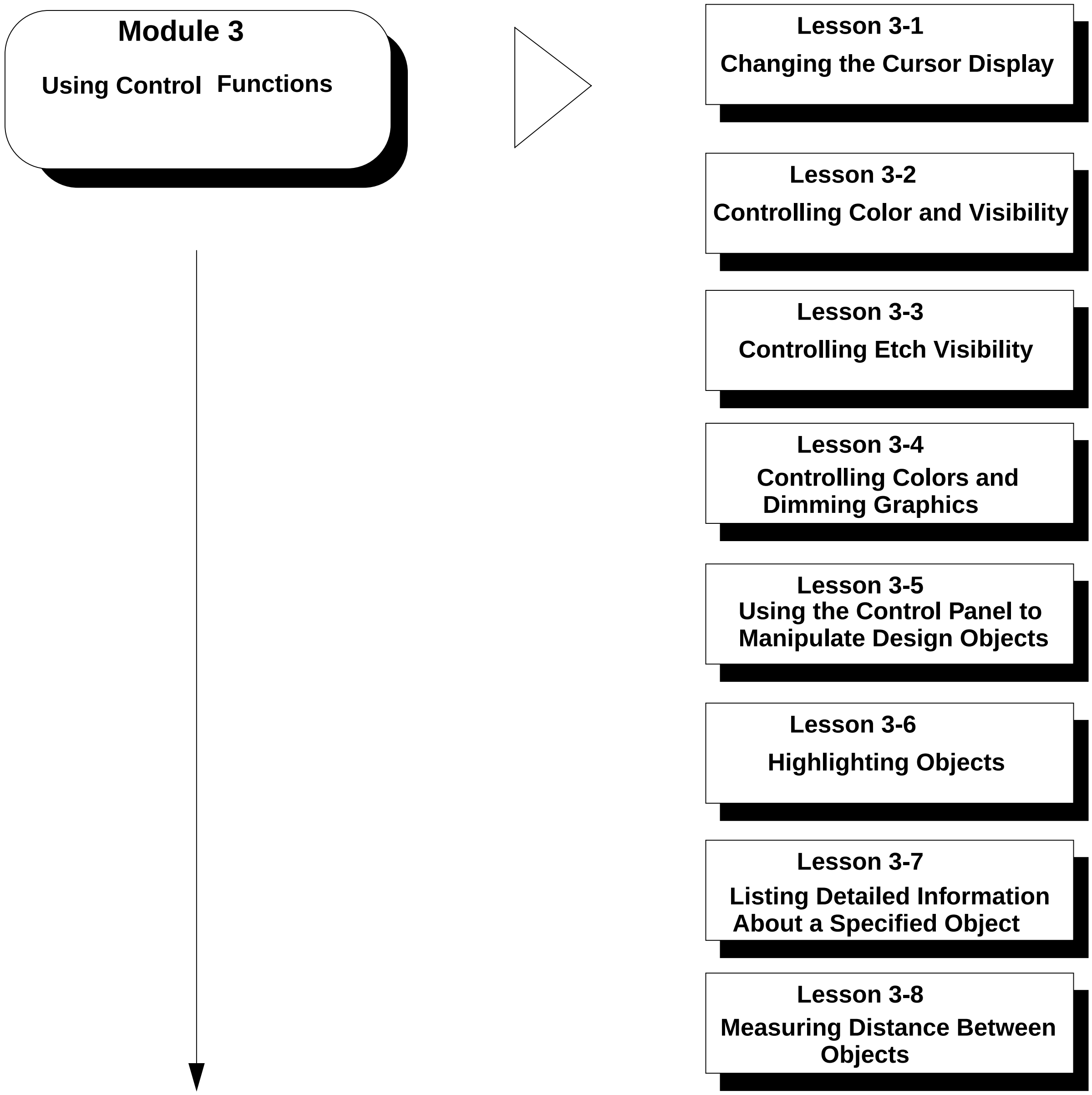
Note: To hear audio in a Windows-based system, ensure that the Volume icon in the task bar is on.

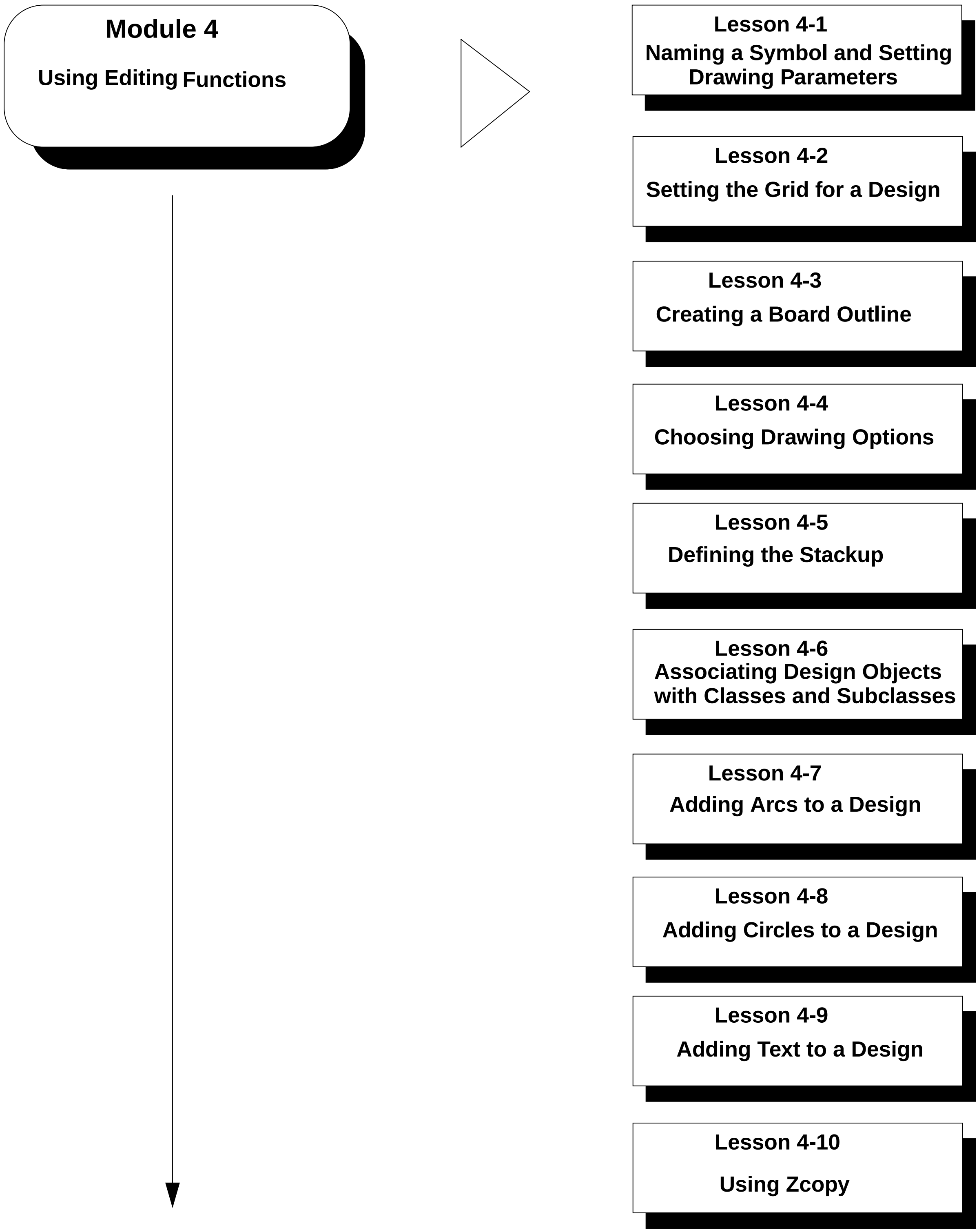
Tutorial Flow

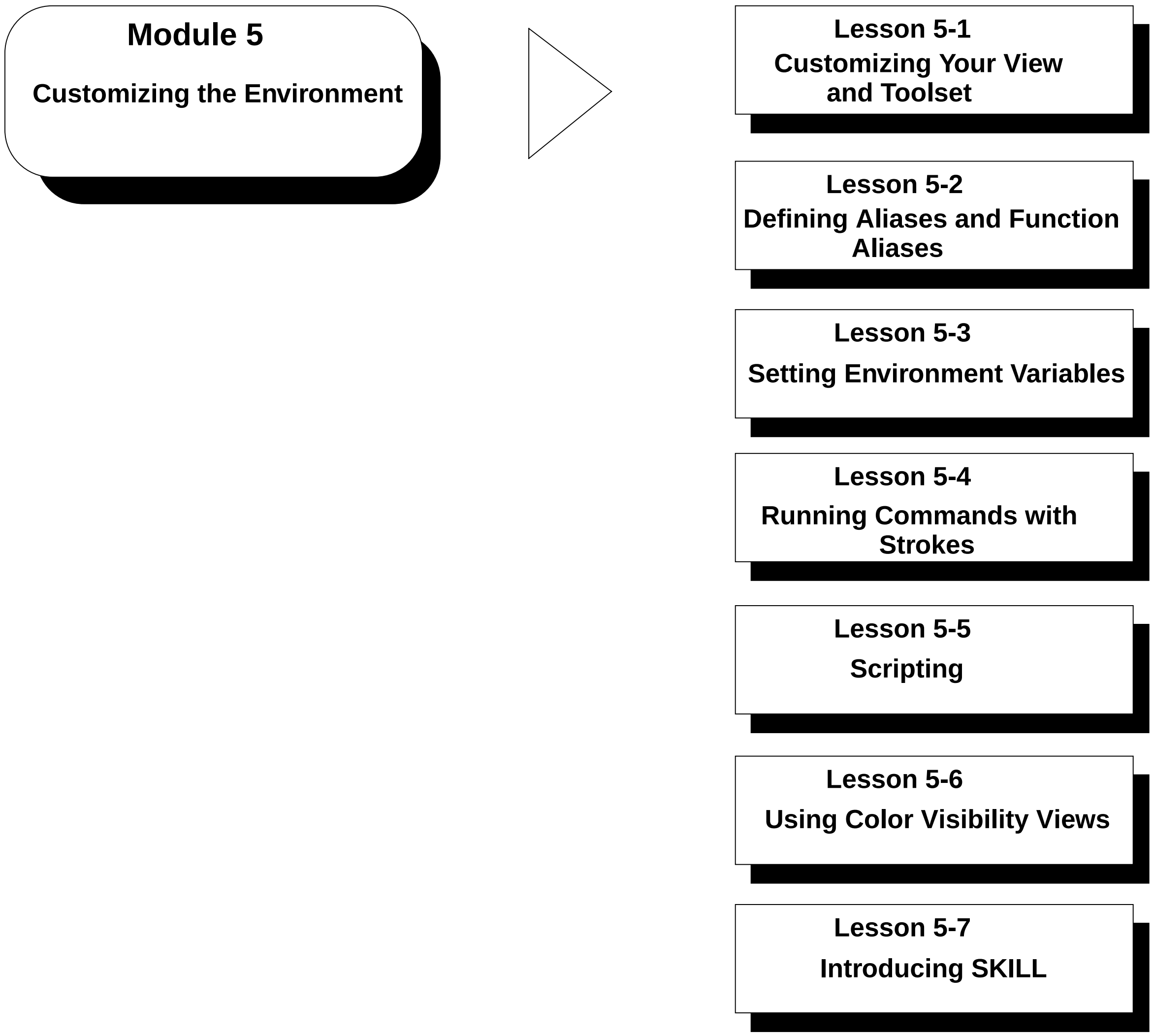
In addition to the chapter, About Allegro PCB Editor, which provides some basic information about the product, this tutorial consists of the modules and lessons shown in Figure 1-1.

Figure 1-1 Modules and Lesson




Module 3
Using Control Functions
Lesson 3-1
Changing the Cursor Display
Lesson 3-2
Controlling Color and Visibility
Lesson 3-3
Controlling Etch Visibility
Lesson 3-4
Controlling Colors and Dimming Graphics
Lesson 3-5
Using the Control Panel to Manipulate Design Objects
Lesson 3-6
Highlighting Objects
Lesson 3-7
Listing Detailed Information About a Specified Object
Lesson 3-8
Measuring Distance Between Objects


Module 4
Using Editing Functions
Lesson 4-1
Naming a Symbol and Setting Drawing Parameters
Lesson 4-2
Setting the Grid for a Design
Lesson 4-3
Creating a Board Outline
Lesson 4-4
Choosing Drawing Options
Lesson 4-5
Defining the Stackup
Lesson 4-6
Associating Design Objects with Classes and Subclasses
Lesson 4-7
Adding Arcs to a Design
Lesson 4-8
Adding Circles to a Design
Lesson 4-9
Adding Text to a Design
Lesson 4-10
Using Zcopy



Related Information

At the end of each lesson, you will find hyperlinks to related sections of the *Allegro PCB Editor User Guide, Allegro Package Designer User Guide,* and the *Allegro PCB and Package Physical Layout Command Reference*. You can also access these manuals from the Help menu.

Syntax Conventions

This list describes the syntax conventions used in this tutorial.

`literal`	Key words that you must enter literally. These keywords represent commands (functions, routines) or option names.
Courier font	Indicates command line examples.
UI	Words in this font refer to menus, labels, fields, or tabs on the user interface.
`variable`	Words in this font refer to arguments for which you must substitute a value.

About Allegro PCB Editor

This chapter provides some basic information about the Allegro PCB Editor and APD that you can review before starting the lessons in this tutorial.

Throughout this tutorial, references are made to Allegro PCB Editor and APD. However, most of the Cadence Silicon Package Board (SPB) products – APD, and Allegro PCB SI – are built on the same base. These other tools have additional functionality that is not shared with Allegro PCB Editor, but the method to interact with the design is the same.

The chapter discusses these topics:

- Allegro PCB Editor Tools on page 19
- Allegro PCB Editor Initialization on page 21
- env File on page 22
- Cadence File Types on page 23
- Allegro PCB Editor Database on page 26
- Table 2-2 on page 26
- Requirements for a New Design on page 27
- Allegro PCB Editor Flow on page 36
- Allegro PCB Editor Menus and Functions on page 39
- Sources of Information on page 42

Allegro PCB Editor Tools

Based on the licenses you have purchased and the product choices made by the installer, you may have access to these tools:

- Allegro PCB Editor – Used to create and modify PCB designs. Based on whether you are in layout mode or in symbol creation mode, the editor appears with a specific menu set.

- Constraint Manager – A spreadsheet-based product, which acts as a command center for the correct-by-design process. Constraint Manager establishes, manages, reviews, and validates physical and spacing constraints as well as electrical design rules or constraints that control interconnect signal quality. As of 16.0, Constraint Manager (specifically the Physical and Spacing worksheets) is available for all PCB Editor tiers. This powerful tool allows you to graphically create, edit, and review topology templates or electronic blueprints. It provides real-time updates of the spreadsheets, and automatically integrates the results for you.

- Padstack Designer – A graphical user interface that lets you create and visualize multi-layer padstacks. This tool eases the definition of complex padstack geometries by visualizing the padstack from the cross-section and plane views.

- Allegro PCB Router – A tool that handles high-density printed circuit boards requiring complex design rules. The Allegro PCB Router uses powerful, shape-based algorithms to efficiently use the routing area. In addition, the Allegro PCB Router integration with Allegro PCB Editor layout, Allegro PCB SI, and APD provides high-speed constraint management across the entire design flow.

- Allegro PCB SI – A tool that offers an integrated high-speed design and analysis environment for creating digital PCB systems and integrated circuit (IC) package designs. Allegro PCB SI allows you to explore and resolve electrical performance-related issues in all stages of the design cycle. By exploring and making trade-offs among timing, signal integrity, crosstalk, power delivery, and EMI, you can optimize electrical performance before committing to final design for manufacture.

- Allegro Package Designer XL (APD) – A tool that uses the SpiderRoute autorouting technology for complex, high-density interconnect IC packages. APD SpiderRoute complements the already robust IC packaging routing environment, which includes the Allegro PCB Router, by providing innovative algorithms for supporting all chip-attach technology. APD SpiderRoute provides IC package designers with true any-angle, multi-layer routing capability, concurrent routing, pre-route feasibility, and on-the-fly pin swap.

- Online documentation.

Also installed is a number of programs that you can run from an operating system prompt. These programs may display graphical user interfaces when run, or may require that you enter arguments and options from the keyboard.

Allegro PCB Editor Initialization

When you start Allegro PCB Editor, it reads these files:

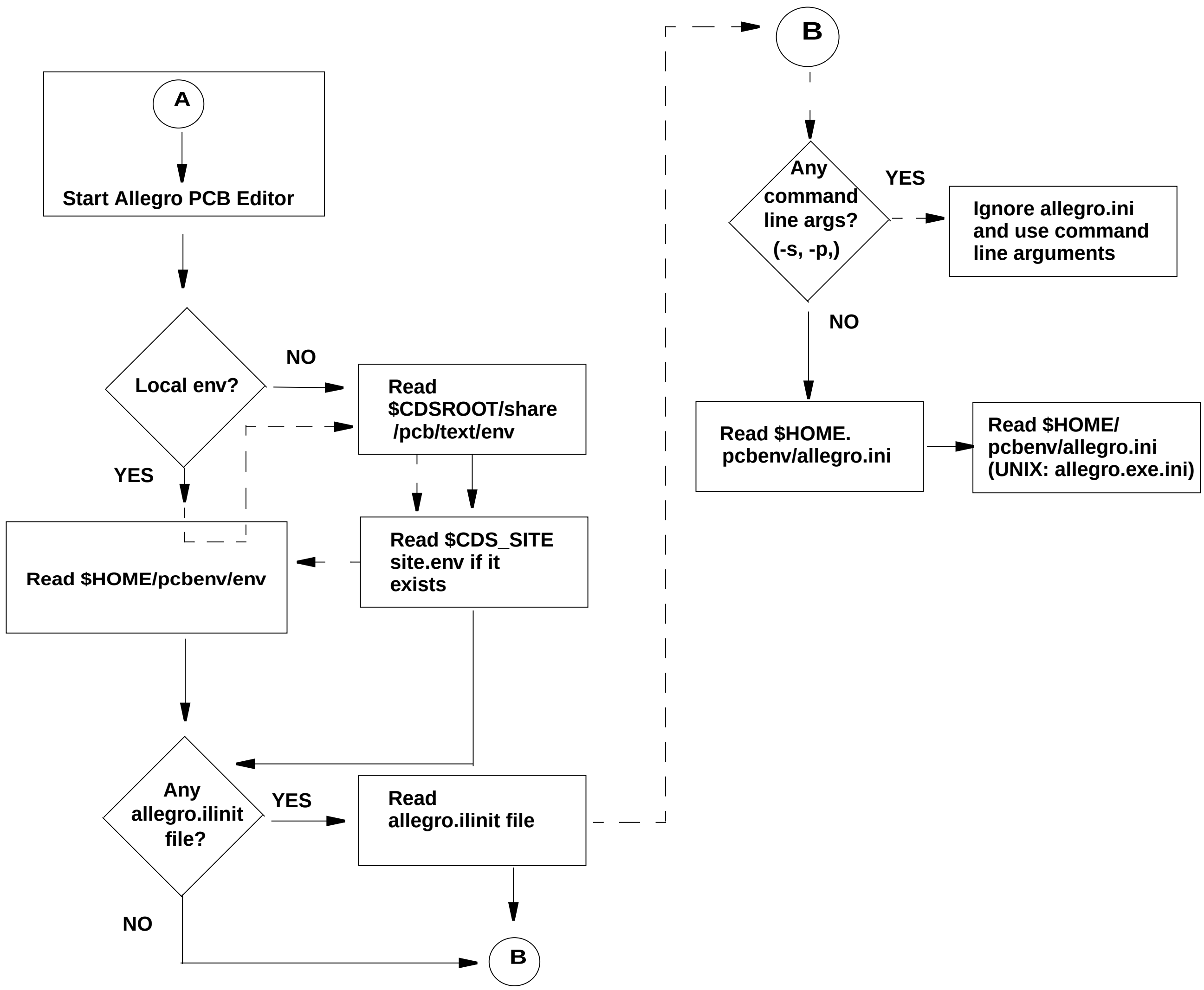
- `env`
- `allegro.ini`
- `allegro.ilinit`

The environment (`env`) file, located in the `pcbenv` directory, determines the location of libraries, menus, forms, scripts, other Allegro PCB Editor directory pathnames, and keyboard assignments (aliases). Allegro PCB Editor reads the `allegro.ini` text file, located in the same directory. This file stores various settings such as the toolbar setting window size, plotting setup, and so on. You should not edit this file, but if you delete it, the editor restores the default settings.

At startup, Allegro PCB Editor also searches for the `allegro.ilinit` file. This file contains the location of the SKILL directory and loads the SKILL commands for use. The directory search order is:

```
$CDSROOT/share/pcb/etc/skill;$ALLEGRO_SITE/skill;$HOME/
pcbenv;
```

Figure 2-1 Allegro PCB Editor Initialization



env File

When you start Allegro PCB Editor, it looks for a `$HOME/pcbenv` directory. If it does not find one, it creates a `pcbenv` directory with startup files such as `env` file, `allegro.ini`, and `allegro.geo`, at a location determined by the value of the environment variable `HOME`. The `.geo` and `.ini` files store your window and toolbar preferences.

If you have not explicitly set a `HOME` variable, the Allegro PCB Editor uses a combination of the `HOMEDRIVE` and `HOMEPATH` variables to generate the home directory (`HOMEDRIVE:\HOMEPATH`) on Windows. If the `HOMEDRIVE` and `HOMEPATH` variables do not exist, the editor uses `C:\`.

Caution

Do not edit the files in your `pcbenv` ***directory. Instead, use the User Preferences Editor dialog box to set environment variables. See Lesson 5-3: Setting Environment Variables on page 162 for information on performing this task. If your home directory is inaccessible or write-protected, you cannot save any of your preferences.***

Cadence File Types

Cadence supports the file types described in Tables 2-1 and 2-2.

Table 2-1 Allegro PCB Editor Database Objects

If the File Has This Extension...	It Is a...	And You Use This Tool...
`.brd`	PCB design database file.	Allegro PCB Editor with Layout menus
`.dra`	Drawing file. You must create this file before you create a symbol file. Later, this file is compiled into a binary symbol file.	Allegro PCB Editor – Allegro Package
`.pad`	Padstack file.	Padstack Editor
`.mcm`	Multi-chip module design file.	APD
`.osm`	Library file that stores format symbols such as a legend or a company logo.	Allegro PCB Editor – Allegro Format
`.psm`	Library file that stores package/part symbols, for example, an IC.	Allegro PCB Editor – Allegro Package

If the File Has This Extension...	It Is a...	And You Use This Tool...
`.bsm`	Library file that stores drawing or board/substrate symbols, for example, a board or design outline.	Allegro PCB Editor – Allegro Mechanical
`.fsm`	Library file that stores flash symbols such as a thermal pad for raster formats.	Allegro PCB Editor – Allegro Flash
`.ssm`	Library file that stores shape symbols such as a special shape for a padstack.	Allegro PCB Editor – Allegro Shape
`.mdd`	Library file that stores module definitions.	Allegro PCB Editor – with Layout menus
`.dsn`	A file created by translating design information from the layout system. It contains PCB boundary data, layer definitions, padstack definitions, component data, netlist, preroutes, and design rules.	Allegro PCB Router

Allegro PCB Editor supports the reports, input files, and output files described in Table 2-2.

Table 2-2 Reports, Input Files, and Output Files

If the File Has This Extension...	It Is...	Function/Option
`.rou`	An ASCII file in Excellon Format.	Generates output for an NC router based on the parameters you set in the NC Parameters Dialog Box using the `ncdrill param` command.
`.tap`	An output text file that contains NC drill data.	Created when the design is ready for manufacturing.

If the File Has This Extension...	It Is...	Function/Option
`.txt`	A text file, for example, `art_param.txt`, that describes machine-related parameters or `art_aper.txt` that lists the size and shape of each aperture according to aperture wheel.	ASCII text files created during the various processes. The example files are created when the design is ready for manufacturing.
`.scr`	A script or macro file used to play back recorded tasks.	Created during script creation.
`.il`	A SKILL script.	Created with SKILL commands to provide automatic functions.
`.log`	A log file that contains data on processes.	Created during the specific process.
`.art`	An artwork file for selected film records.	Created when the design is ready for manufacturing.
`.dat`	A data file, such as the import logic files: `pstnet.dat`, `pstxprt.dat`, and `pstchips.dat`, or the export logic files: `compview.dat`, `netview.dat`, and `funcview.dat`.	Created during the various processes. The example files are created by the front-end tools and by Allegro PCB Editor.
`.jrl`	A journal file which contains a record of events — menu picks, keyboard activity, and so on.	Recorded for each session in the editor.
`.do`	An Allegro PCB Router script file containing rules and Allegro PCB Router commands.	
`.did`	An Allegro PCB Router output file that contains design rules such as clearance, wiring, timing, cross-talk, and so on.	Generated when you run an automatic routing command on a design.

If the File Has This Extension...	It Is...	Function/Option
`.ses`	An Allegro PCB Router output file.	Provides routing and optional placement information to the Allegro PCB Editor.

Allegro PCB Editor Database

The Allegro PCB Editor database is binary; the format changes with each major release, for example, from 14.x to 15.x. The current database can be read by later releases, but not by earlier releases. You can use the `uprev` command to convert a database for use by a later release.

See the `uprev` command in the *Allegro PCB and Package Physical Layout Command Reference* for additional information. You can use the `extracta` command to obtain textual information from the database. See the *Completing the Design* user guide in your documentation set for additional information.

Operating System Differences

The differences between using the Allegro PCB Editor on Windows or on UNIX are:

- Use of slashes in pathnames.

  UNIX uses forward slashes in pathnames. Windows uses back slashes in pathnames.

- Allegro PCB Editor startup is different on UNIX and Windows. See Lesson 1-2: Starting Up Allegro PCB Editor on page 49.

Note: Allegro PCB Editor does not support embedded spaces within a file name on either UNIX or Windows. Spaces are supported in directory paths only on Windows.

The Allegro PCB Editor tutorial databases (`.brd` files) can be opened in either operating system.

Requirements for a New Design

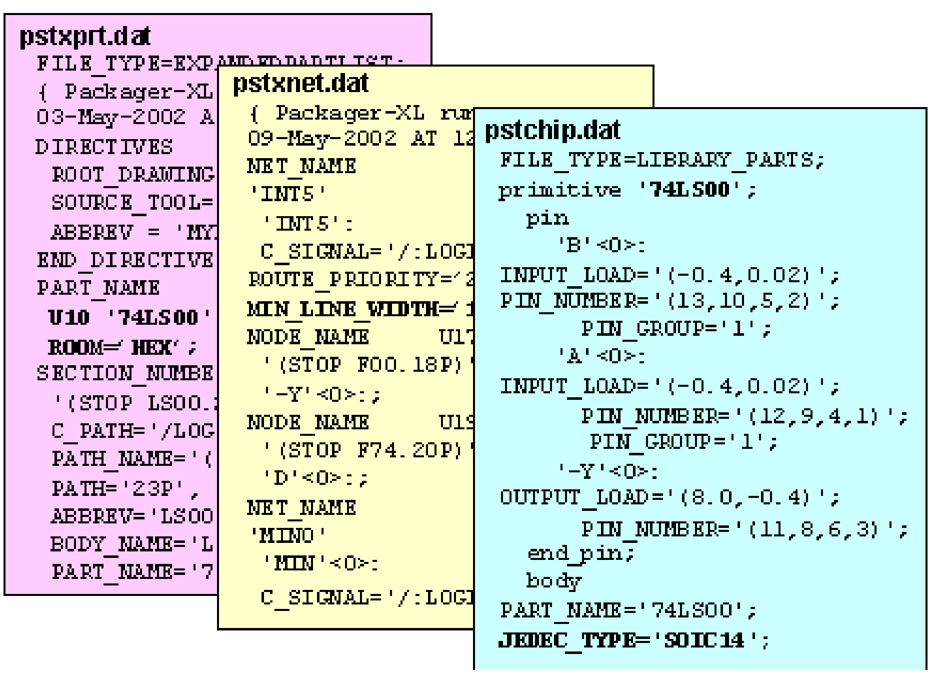
You can create a design in the Allegro PCB Editor by importing logic from:

- Allegro Design Entry HDL schematic or netlist
- Allegro Design Entry CIS schematic or netlist
- OrCAD Capture CIS schematic or netlist
- Third-party netlist

Allegro Design Entry HDL

After the schematic is created in Allegro Design Entry HDL XL, the Packager-XL utility combines the logic devices with physical packages, assigning a reference designator and physical pin numbers to each symbol in the schematic. The packaged parts and their connections are written into transfer files (Figure 2-2 and Table 2-3). These files transfer information from the schematic to an Allegro PCB Editor design.

Figure 2-2 Transfer Files (pst*.dat) for Traditional Flow

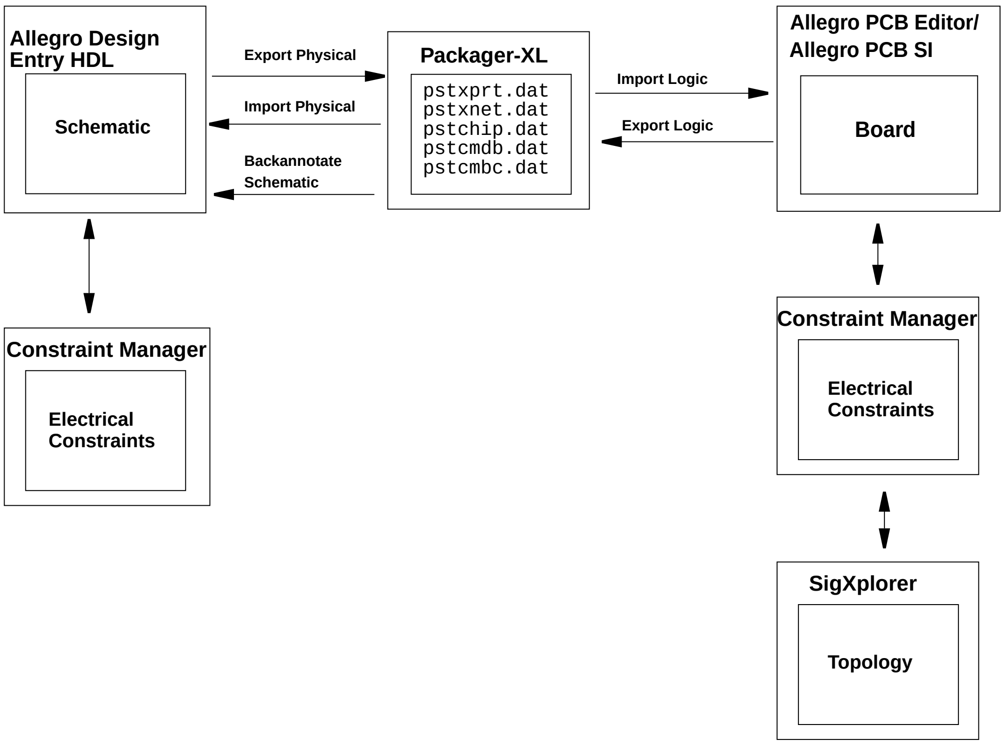


Note: If you are using the traditional flow, which means that you are not using the Constraint Manager with Allegro Design Entry HDL XL, Allegro PCB Editor reads `pstxprt.dat, pstxnet.dat, and pstchip.dat` netlist (output) files from Allegro Design Entry HDL. In the Constraint Manager-enabled flow, Allegro PCB Editor reads `pstxprt.dat, pstxnet.dat, pstchip.dat`, `pstcmdb.dat`, and `pstcmbc.dat` files. Based on information contained in these files, Allegro PCB Editor produces or updates an Allegro PCB Editor layout file.

Table 2-3 Descriptions of Transfer File for Traditional Flow

File	Description
`pstxprt.dat`	An expanded parts list file that lists each physical package (created by the Packager-XL) in the schematic with its reference designator and device type. For packages comprised of multiple logic gates, this file identifies which gate is placed in which section of the physical package. This file may also contain some properties attached to parts in the schematic, such as ROOM='IF', VALUE='4.7K'.
`pstxnet.dat`	An expanded netlist file that uses keywords (`net_name, node_name`) to specify the reference designators and pin numbers associated with each net in the schematic. This file may also contain some properties attached to nets in the schematic, such as ROUTE_PRIORITY, ECL, and so on.
`pstchip.dat`	A device definition file (`chips`) that contains electrical characteristics (for example, pin direction and loading), logical-to-physical pin mapping, and voltage requirements. It defines the number of gates in a device, including gate and pin swapping information. This file also contains the name of the package symbol that represents this device type in the physical layout (such as JEDEC_TYPE='DIP14_3', ALT_SYMBOLS='(T:SOIC14)').

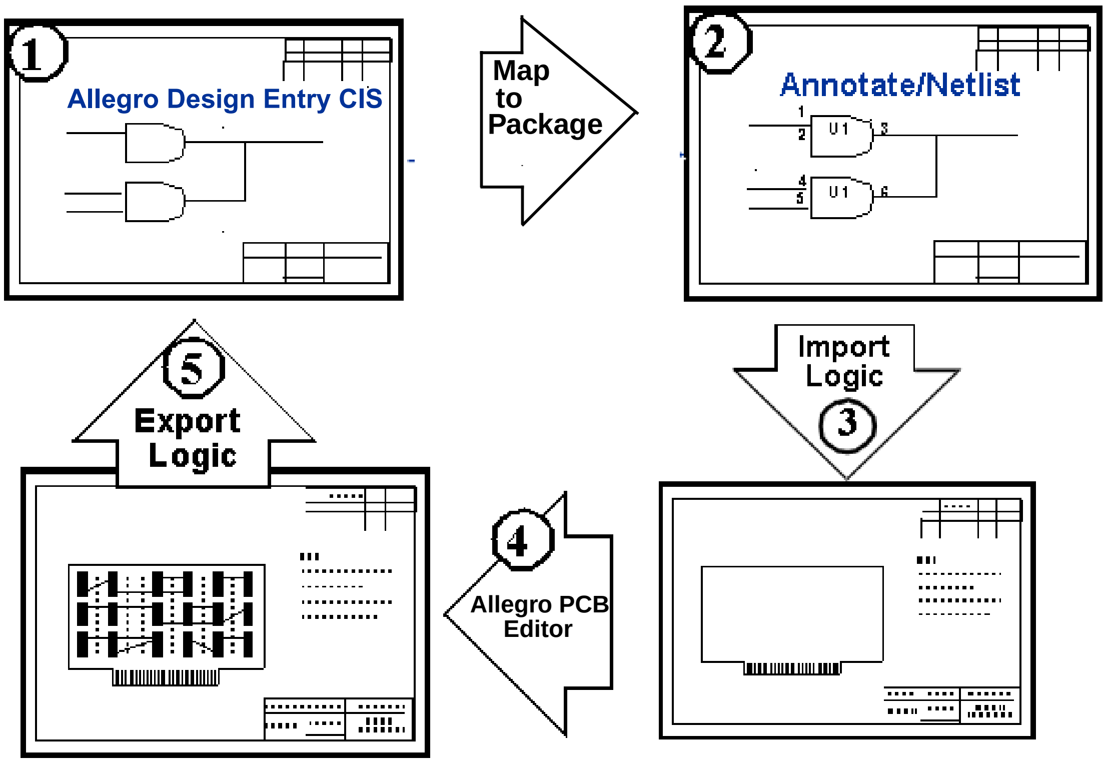
Figure 2-3 Allegro Design Entry HDL XL-Integrated Logic Design with Physical Layout



Front-end Integration

Figure 2-4 shows the front-to-back integration between Allegro Design Entry CIS (or OrCAD Capture CIS) and the Allegro PCB Editor tools.

Figure 2-4 Allegro Design Entry CIS-Integrated Logic Design with Physical Layout



Allegro Design Entry CIS: It is not required that the Allegro Design Entry CIS schematic resides in the same directory as the Allegro PCB Editor design. However, it is recommended that you keep the two together.

Annotate: The Annotate program converts the logic devices into physical packages, assigning a reference designator and physical pin numbers to each symbol in the schematic.

Netlister: The Netlister creates the transfer files used by the Allegro PCB Editor. By default, these files are: `pstxnet.dat`, `pstxprt.dat`, and `pstchips.dat`.

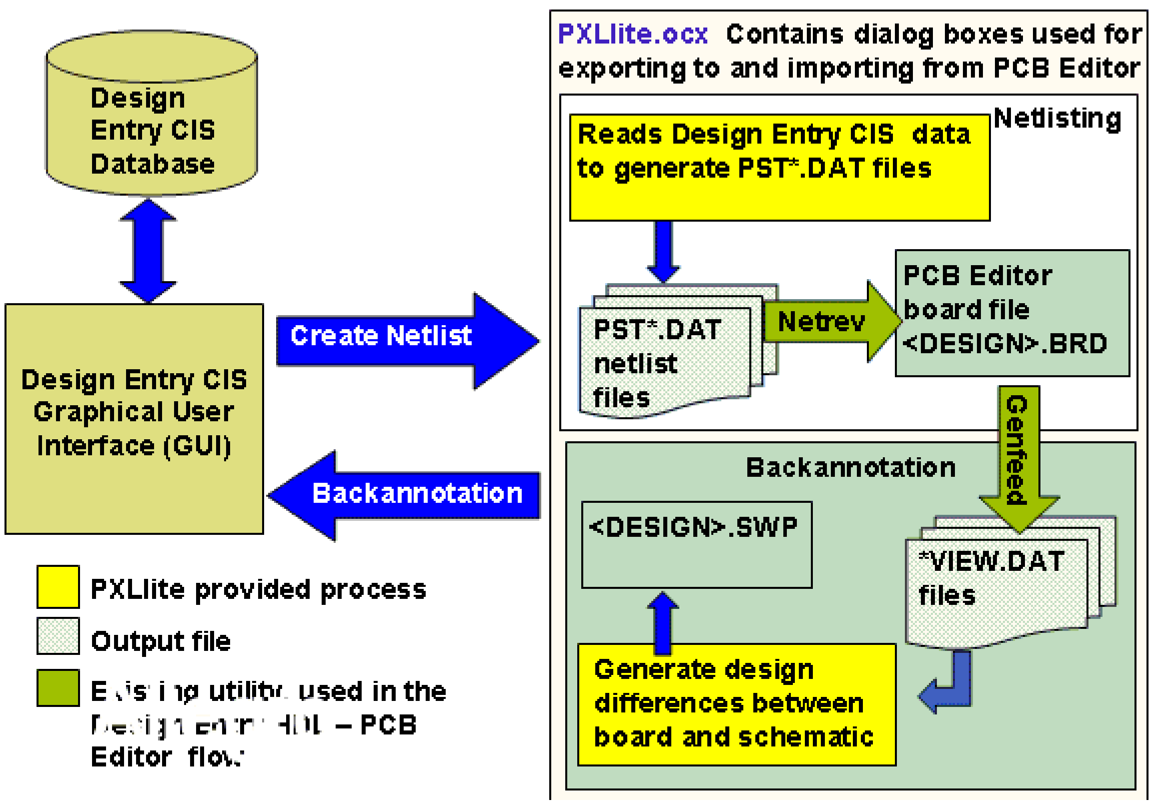
Back-end Integration

Import Logic: After you import logic, the design contains connection information.

Allegro PCB Editor: This tool places pin and gate swaps for optimum routing results, routes, and generates manufacturing output.

Export Logic: This program generates backannotation files that Allegro Design Entry CIS uses to update the schematic.

Figure 2-5 Allegro Design Entry CIS Interface with Allegro PCB Editor



The Netlister (PXLlite) reads the Allegro Design Entry CIS database and creates the same format `.pst` files as the Packager-XL routine. Therefore, Allegro PCB Editor can use the same program to read in either an Allegro Design Entry CIS schematic or an Allegro Design Entry HDL schematic.

Allegro PCB Editor performs backannotation. Then, Allegro Design Entry CIS reads these files and updates the schematic to reflect any changes made to the design by the Allegro PCB Editor, such as pin and gate swapping, reference designator changing, and so on.

Third-Party Netlist

If you have not used Allegro Design Entry HDL or Allegro Design Entry CIS to generate the schematic, you must use a netlist and device files.

The netlist contains the part and connectivity data. Device files are library files that describe the parts in the netlist (one device file per device type). Allegro PCB Editor reads the netlist into a design and produces a log file that lists any errors found in the netlist or device files.

You can also generate a backannotation file to return data back to the third-party system.

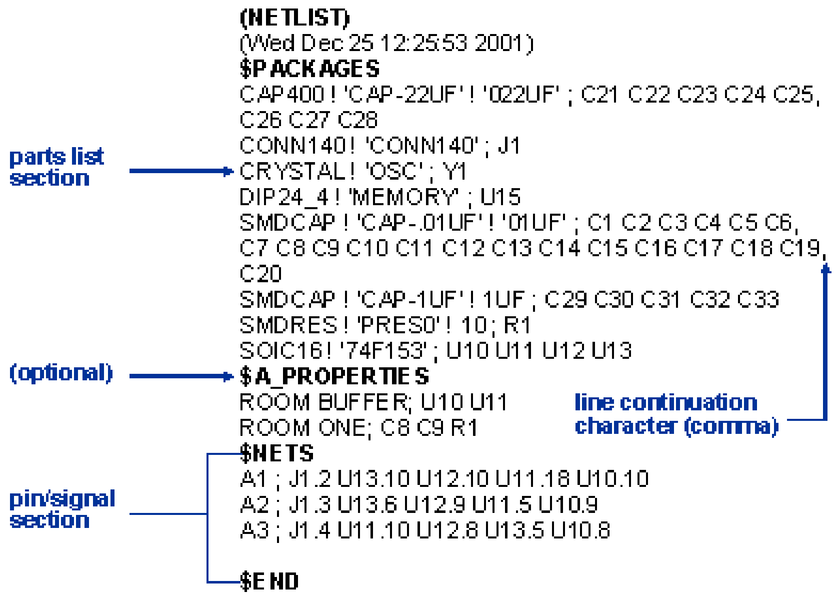
When it reads the netlist, the `devpath` environment variable is used to locate the device files required.

Netlist

The netlist contains two main sections (Figure 2-6): PACKAGES and NETS. The PACKAGES section contains a parts list; the section begins with $PACKAGES. You must identify each reference designator in the design in this section.

The NETS section contains all the nets in the design and the pin connections for those nets. This section begins with $NETS.

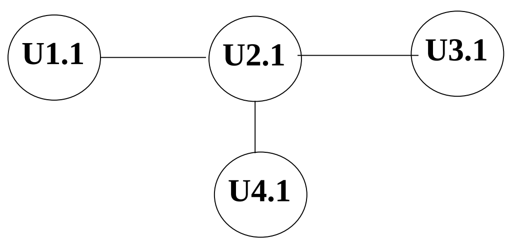
Figure 2-6 Netlist Example



Be sure to:

- Use the $A_PROPERTIES section when adding properties to the netlist.
- Include the line, $A_PROPERTIES, after you define all the parts in the $PACKAGES section when adding component or part-level properties to the netlist.
- Include the line, $A_PROPERTIES, after you define all the nets in the $NETS section when adding net or signal-level properties to the netlist.
- Use the $SCHEDULE section to define specific pin order connection. Include this section after the $NETS section. Figure 2-7 shows an example of a schedule describing a T connection.

Figure 2-7 Example of T Connection Schedule



$NETS
CLK; U1.1 U2.1 U4.1 U3.1
DATA1; U1.3 U5.5 J2.1
$SCHEDULE
CLK; U1.1 U2.1 U3.1; U2.1 U4.1

Table 2-4 shows the maximum field width and allowable characters for each data field in an Allegro PCB Editor netlist.

Table 2-4 General Rules for Netlist

Field Name	Length	Acceptable Characters
`package name`	31	A to z, 0 to 9, dash (-) and underscore (_)
`device type`	30	All except ! and '
`function designator`	30	All except ! and '
`reference designator`	30	All except ! and '
`pin number`	30	All except ! and '
`pin name`	30	All except ! and '
`net name`	31	All except ! and '
`property value`	79	All except ! and '
`tolerance`	79	All except ! and '
`user part number`	79	All except ! and '

Data fields are not case-sensitive in netlists. Other rules to remember when creating a netlist include:

- Do not exceed 78 characters on a line in a data record. Extend records by adding a comma after the last instance in a line. The comma acts as a continuation mark.
- Include comments in parentheses; they are ignored by the `netin` process. Do not include comments within a data field.

Device File

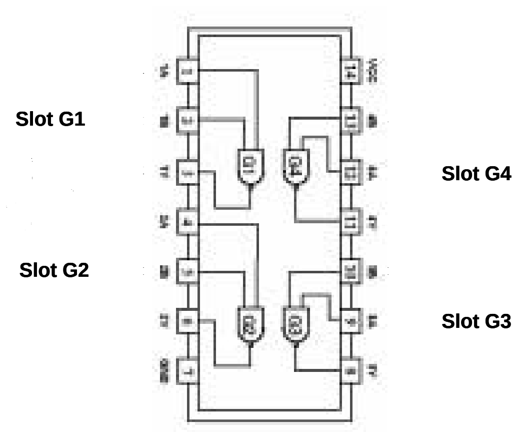
A device file (Figure 2-8) must exist for each different part type used in the netlist. The device file name must be the part type as it appears in the netlist with the extension`.txt`. Allegro PCB Editor determines the path used for locating the device files with the environment variable `devpath`, defined in the `env` file.

You can create a device file in symbol mode using the *File – Create Device* (`create device`) menu command.

Figure 2-8 Device File Example

filename = 7400.txt

```
(Netlist contains device type 7400)
PACKAGE DIP14_3
CLASS IC
PINCOUNT 14
PINORDER    7400 A B -Y
PINUSE      7400 IN  IN OUT
PINSWAP     7400  A  B
FUNCTION G1 7400  1   2   3
FUNCTION G2 7400  4   5   6
FUNCTION G3 7400  9  10 8
FUNCTION G4 7400 12  13 11
POWER    +5V; 14
GROUND   GND;  7
END
```



You must use device files if you import third-party netlist data into the Allegro PCB Editor. The Allegro Design Entry HDL and Allegro Design Entry CIS schematic tools provide electrical component descriptions and connectivity data. Third-party netlists do not contain electrical component descriptions, and therefore require the use of device files. Similar to symbol files, which provide physical component descriptions, device files provide electrical descriptions. Where physical descriptions include pin spacing, body size, and padstack information, electrical descriptions define input and output pins, power pins, and gate assignments.

When creating device files:

- Use lowercase letters with a `.txt` extension for device file names. However, note that the contents of device files are not case-sensitive.
- Use parentheses to enclose comments.

- Include these mandatory lines in the device file:

  PACKAGE

  PINCOUNT

For additional information, see the *Transferring Logic Design Data* user guide in your documentation set.

Allegro PCB Editor Flow

The Allegro PCB Editor integrated suite of software tools for systems design helps you perform the major tasks of PCB and Single Chip Microprocessor (SCM)/ Multi-Chip Module (MCM) design, including:

- Logic design import

  Create a printed circuit board design based on data from a Allegro Design Entry HDL or Allegro Design Entry CIS schematic, or a netlist from another Computer Aided Engineering (CAE) system. Then, backannotate from the design to the schematic. Update Allegro PCB Editor and APD designs by performing engineering change orders (ECOs).

- Physical layout

  Place design elements and route them, either manually or automatically with the Allegro PCB Router.

- Design analysis

  Perform design analysis with SigNoise and EMControl.

- Manufacturing output

  Generate silk screens and penplots, and create artwork and drill files.

Figure 2-9 shows the functional relationship between Allegro PCB Editor and other Cadence or Electronic Design Automation (EDA) tools for logic design, physical layout activities, and design analysis.

Figure 2-9 Functional Relationship Among System Design Tools

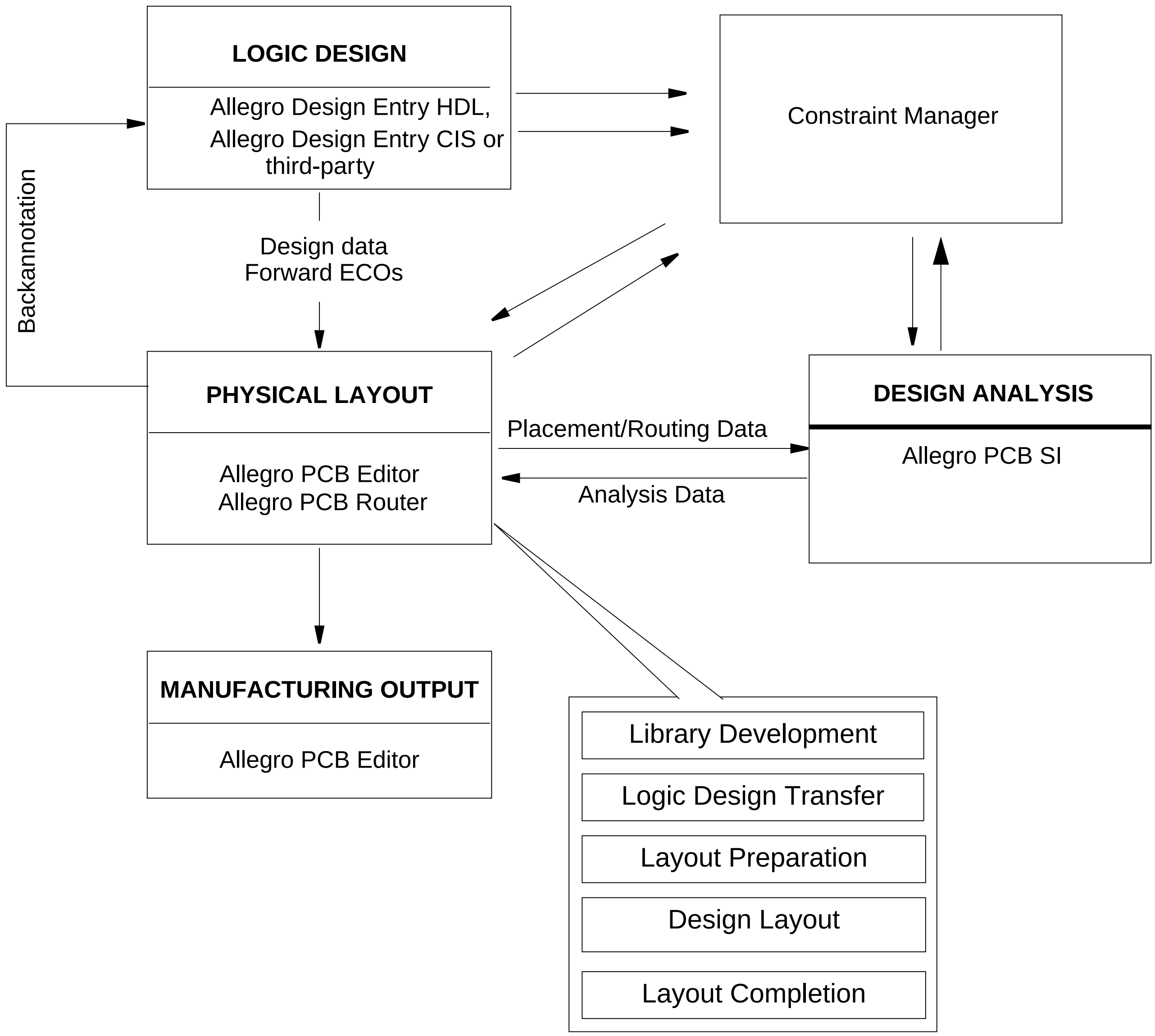
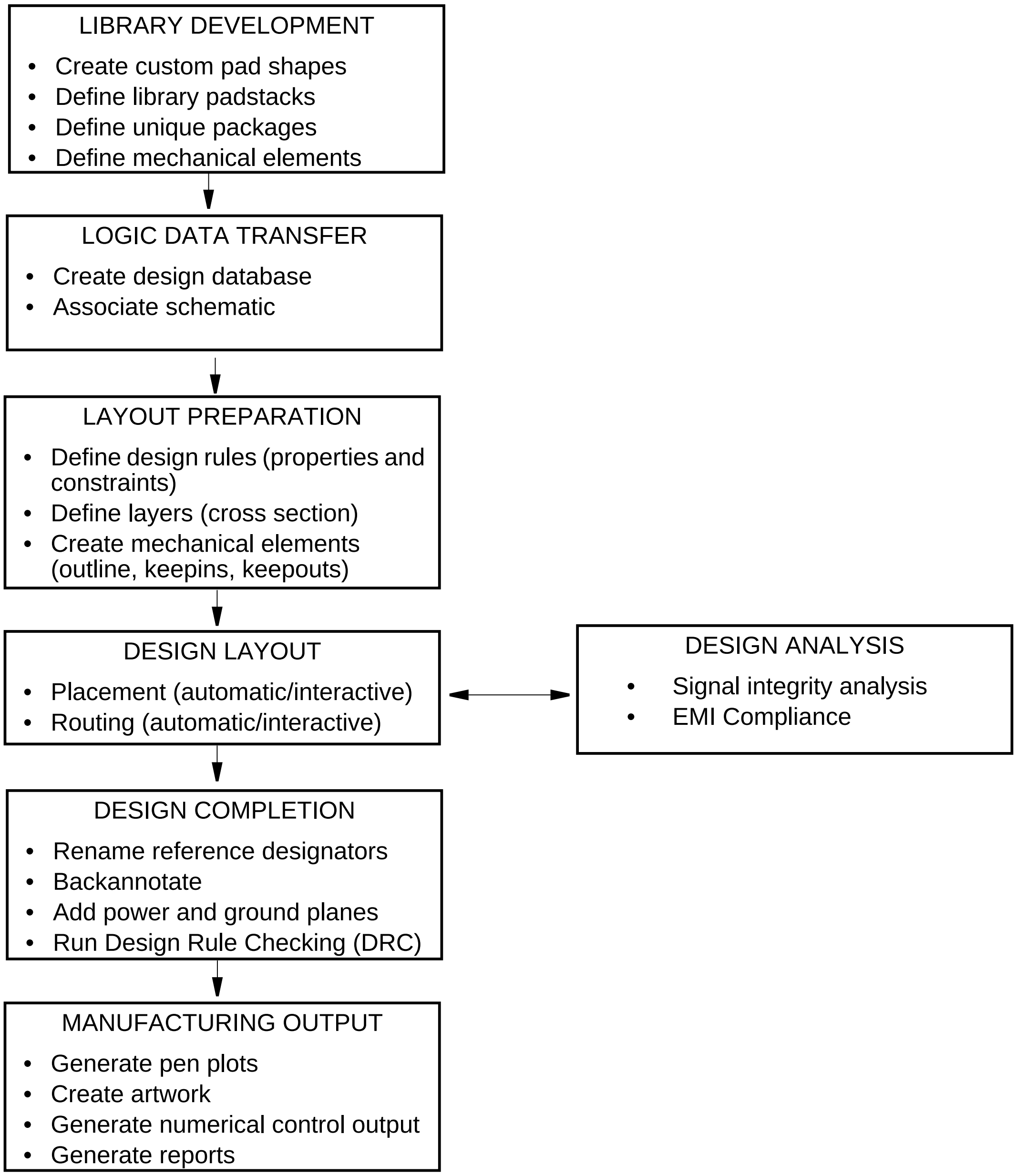


Figure 2-9 defines the typical PCB design flow process using Allegro PCB Editor.

Figure 2-10 PCB Design Flow Using Cadence Tools


LIBRARY DEVELOPMENT
• Create custom pad shapes
• Define library padstacks
• Define unique packages
• Define mechanical elements
LOGIC DATA TRANSFER
• Create design database
• Associate schematic
LAYOUT PREPARATION
• Define design rules (properties and constraints)
• Define layers (cross section)
• Create mechanical elements (outline, keepins, keepouts)
DESIGN LAYOUT
• Placement (automatic/interactive)
• Routing (automatic/interactive)
DESIGN ANALYSIS
• Signal integrity analysis
• EMI Compliance
DESIGN COMPLETION
• Rename reference designators
• Backannotate
• Add power and ground planes
• Run Design Rule Checking (DRC)
MANUFACTURING OUTPUT
• Generate pen plots
• Create artwork
• Generate numerical control output
• Generate reports

Allegro PCB Editor Menus and Functions

Allegro PCB Editor menu bar (shown below) groups tasks within each menu.

For example, if you choose the *Route* menu, you can choose items under the menu that correspond to the routing tasks you want to perform. Table 2-5 lists all the menus common to the tools and products and summarizes the tasks for each menu. Table 2-6 shows the *Layout* menu, used only in symbol creation, and the *Analyze* menu, used only by Allegro PCB Editor XL.

Table 2-5 Common Menus

Menu Name	Functions
File	Lets you open, save, and close existing files, create new files, import information such as logic, Gerber artwork files, DFX data, IDF data, and so on, export a variety of data, and run scripts.
Edit	Lets you manipulate objects in your design, such as moving, copying, rotating, and deleting objects.
View	Lets you zoom in and out of a design, create, change, or restore a color visibility view, and customize your work environment.
Add	Lets you add lines, circles, rectangles, filled rectangles (frectangles), arcs, and text to your design.
Display	Lets you display and control colors and visibility of classes and subclasess (for more information, see Lesson 4-6: Associating Design Objects with Classes and Subclasses on page 144), highlight and dehighlight elements, calculate capacitance between any two conductor elements, view properties, and display ratsnest lines in your design or remove them from your design.
Setup	Lets you set up drawing parameters, grids, subclasses, and layers; define vias, constraint sets, properties, and areas; define user preferences (variables), and open the Constraint Manager.

Menu Name	Functions
Shape	Lets you perform a variety of shape tasks including adding multi-sided shapes, rectangles, or circles to your design, creating non-copper polygons or rectangles within a copper area, creating circles within etch shapes that are recognized as unfilled during penplotting and photoplotting, choosing pins or vias to create an unfilled clearance hold for static shapes, and converting groups of lines and arcs into shapes.
Logic	Lets you handle all electrical changes, scheduling nets, and changing nets.
Place	Lets you set up automatic placement controls and define automatic placement grids for placing components, symbols, and modules in a design. A module is a user-defined grouping of components and related etch and pins.
Route	Lets you route manually or automatically.
Manufacture	Lets you specify parameters for adding drafting items to the layout, set parameters for the NC drilling program, add test points to the design, and create a Bill of Materials (BOM). **Note:** *Manufacture - Dimension/Draft* commands in the layout mode are available under the *Dimension* menu item in symbol mode.
Tools	Lets you create modules, modify both design and library padstacks, specify parameters for silkscreening, create reports, check the database, and update the Design Rule Checking (DRC) markers.
Help	Lets you access Allegro PCB Editor help system, user documentation, web resources, and information about the International Cadence Usergroup (ICU). See Lesson 1-4: Accessing the Help System on page 54.

Table 2-6 Special Menus

Menu Name	Function	Used In...
Layout	Lets you add pins, connections, reference designators, part numbers, and so on. **Note:** *Manufacture - Dimension/Draft* commands in layout mode are available under the *Dimension* menu in symbol mode.	Allegro PCB Editor – symbol mode
Analyze	Lets you manage setup and simulation; specify the device and interconnect libraries used by the simulator during signal analysis; assign models to devices, pins, and bondwires; and remove model assignments.	Allegro PCB Editor XL

Menu Items and Corresponding Commands

Allegro PCB Editor menu items have corresponding commands that you can enter at the command line. For example, choosing *Route – Connect* from the menu bar operates in the same way as if you typed the `add connect` command in the console window.

When you choose a menu item from the menu bar, the name of the corresponding command appears in the Status window (lower right corner). For more information, see Lesson 2-1: Identifying Parts of the User Interface on page 61.

The commands also appear in the journal (`.jrl`) file. The journal file is a session transcript of all the commands executed, and messages generated by Allegro PCB Editor.

Verb/Noun Command Structure

Allegro PCB Editor graphical user interface (GUI) adheres to most Microsoft Windows™ standards for pull-down menus, accelerator keys, mouse use, icons, and so on. Allegro PCB Editor differs from most Windows applications, however, in that it follows the verb/noun structure where you select the *command–then–object* method of command execution. In Allegro PCB Editor:

1. First choose a command.
2. Then choose the specified object.

For example, to delete an object, choose *Edit – Delete* from the menu bar or type `delete` at the console window prompt, then choose the object that you want to delete.

Important

This tutorial shows you how to work with Allegro PCB Editor and APD in menu-driven editing mode, or verb/noun use model. Ensure that you work with the tutorial in menu-driven editing mode, using *Setup –Application Mode – None* (`noappmode` command) to exit from a current application mode and return to menu-driven editing mode.

Sources of Information

Additionally, you can obtain information from the following:

- SourceLink
- Cadence Customer Response Center
- Education Services
- International Cadence Usergroup (ICU)
- Cadence Channel Partner

SourceLink

SourceLink is a Cadence web site that provides technical information. You need to register so that you can access SourceLink.

Access to SourceLink is limited to customers with a current Cadence Maintenance agreement. For customers without access to Sourcelink, contact your Cadence Channel partner.

Using SourceLink, you can:

- Get information on current and upcoming releases.
- Read technical application notes.
- Download SKILL code written by application engineers and other customers.
- Create Service Requests directly with the Customer Response Center.
- Check the status of Service Requests and Cadence Change Management System (CCMS) Change Requests (CCRs).

You can access SourceLink with your Web browser at sourcelink.cadence.com or by using the Allegro PCB Editor Help menu. See Lesson 1-4: Accessing the Help System on page 54.

Cadence Customer Response Center

Technical support is available for customers who have a maintenance agreement with Cadence. If you need to report a problem in the software or documentation, submit a request from your SourceLink account, or contact your Cadence Channel partner.

Education Services

Cadence offers many education services for customers including traditional classes and web-based training, and will customize training for specific needs. Visit this web site, www.cadence.com/education, for a description of classes and their schedules. OrCAD customers can contact their local Cadence Channel Partner for training services offered.

1

Module 1: Getting Started with Allegro PCB Editor

This module comprises these lessons:

- Lesson 1-1: Creating a Project Directory on page 45
- Lesson 1-2: Starting Up Allegro PCB Editor on page 49
- Lesson 1-3: Setting Your Working Directory and Opening a Design
- Lesson 1-4: Accessing the Help System on page 54

Completion Time

It should take approximately 2 hours to complete the written lessons in this module.

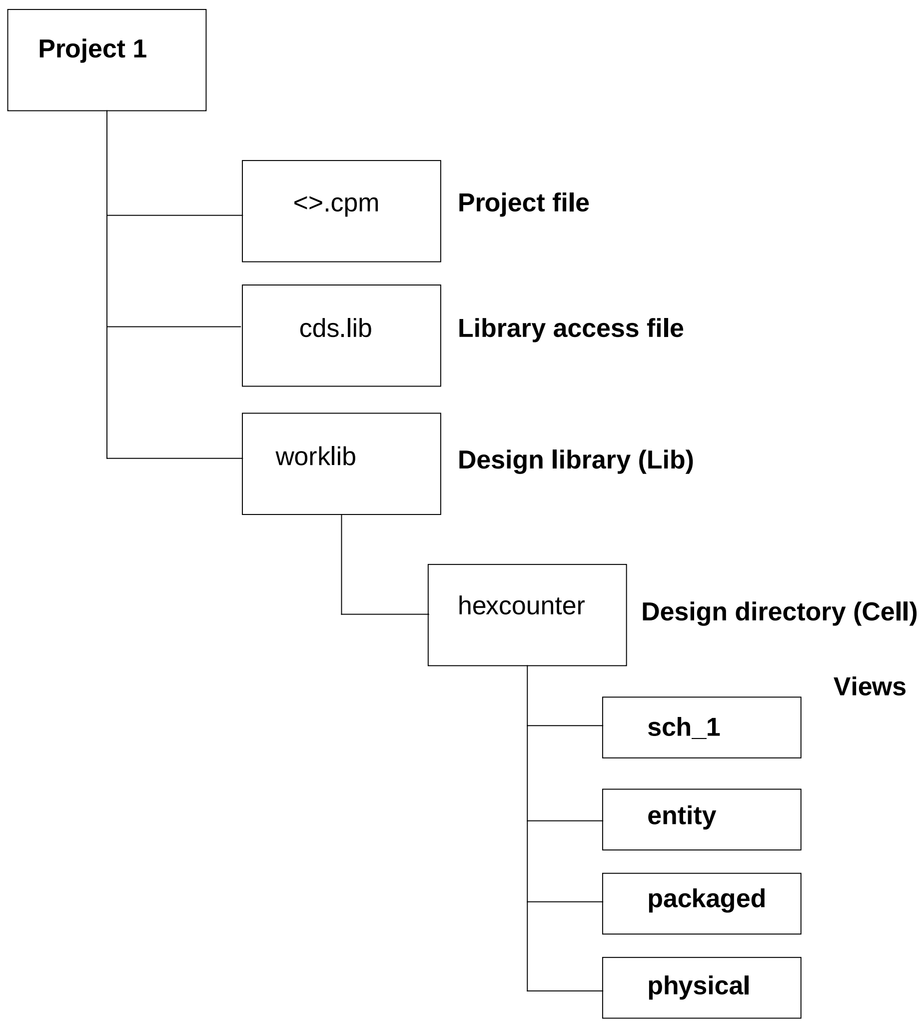
Lesson 1-1: Creating a Project Directory

Overview

You can set up various acceptable directory structures to accommodate the Allegro PCB Editor projects. For example, if you are using Allegro HDL Project Manager, the interface to the Cadence board design solution and library management, the tool automatically creates the directory structure shown in the project directory example below.

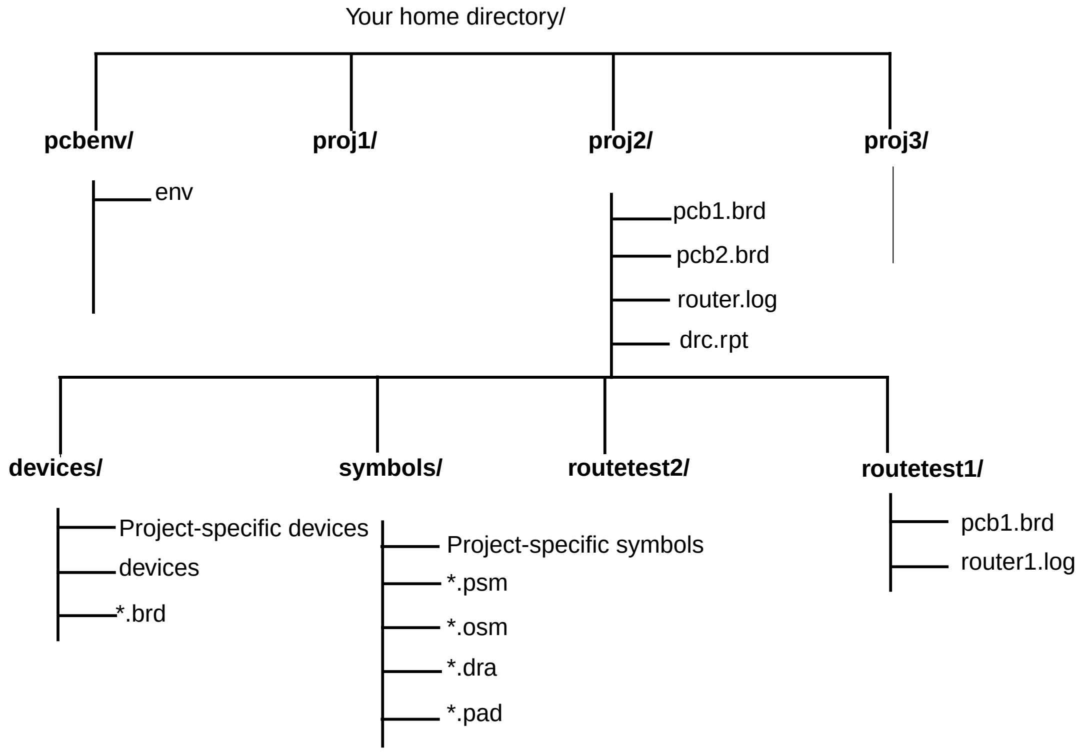
Note: Allegro HDL Project Manager is not available in the OrCAD product line.

Allegro HDL Directory Structure



You can also create a directory organized by project type. The example below shows a simplified view where you store design information in your home directory. This example shows several project directories (for example, `proj1` and `proj2`) and subdirectories under each project.

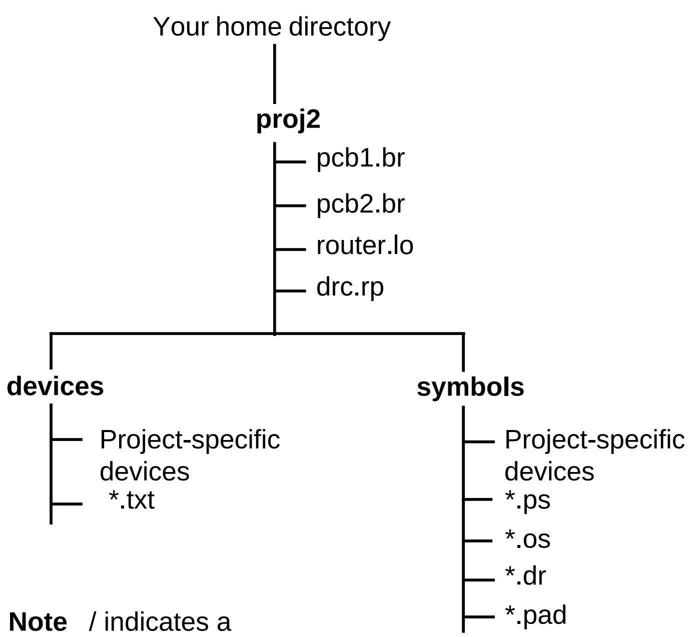
Allegro Design Entry CIS Directory Structure



Note: / indicates a directory

The symbols and devices directories beneath a project directory contain symbols and devices that are unique to that project. These subdirectories parallel the structure of the library directories supplied by Allegro PCB Editor in `<install_dir>/share/pcb/pcb_lib`, where *`install_dir`* is the directory in which Allegro PCB Editor is installed. A project can also contain other subdirectories, such as temporary directories for routing tests that let you run batch routes without replacing log or design files. By default, Allegro PCB Editor searches for symbols using this structure.

Allegro Third Party Directory Structure



In this lesson, you will learn how to create a project directory. For specific commands, see the Help available for your operating system.

Procedure

1. Create a directory called `proj2`.

   Use this location to save board files, log files, and reports when you work on your project.

2. Copy the board file, `cds_routed.brd` and `cds_routed_DRC.brd` from `<`*`installation_directory`*`>\doc\algrotutorial\examples` to the `proj2` directory.

   You will use this directory when working on the lessons in this tutorial.

3. Create a `devices` subdirectory in the `proj2` directory.

   Use this location to store device files when you work on your project.

4. Create a `symbols` subdirectory in the `proj2` directory.

   Use this location to store symbols when you work on your project.

Summary

You now know that there are different methods for setting up a project directory. You also know how to set up a sample project directory.

You have learned:

- **Library directory pathname:** `<`*`install_dir`*`>/share/pcb/pcb_lib`

For More Information

See “Introduction to Project Manager” in the *Allegro Project Manager User Guide*.

What’s Next

Go to Lesson 1-2: Starting Up Allegro PCB Editor to learn how to start up the Allegro PCB Editor.

Lesson 1-2: Starting Up Allegro PCB Editor

Overview

If you purchased more than one type of Allegro PCB Editor tool, when you invoke Allegro PCB Editor, you must choose which license to use from the product suite. For additional information on product choices, see Allegro PCB Editor Tools on page 19.

In this lesson, you will learn how to start up Allegro PCB Editor on Windows or UNIX.

Procedure

1. Start Allegro PCB Editor in one of the following ways, depending on whether you are working on Windows or UNIX:

a. On Windows, use one of the following methods to start the editor:

❍ Click the Windows *Start* button (bottom left of your screen) and choose *Programs – Cadence Design Systems* ** - ** where ** - ** represents the name you assigned to Allegro PCB Editor application.

❍ Double click the Allegro PCB Editor icon.

❍ Double click a *.brd* file.

If you choose this method, be sure that you have associated the Allegro PCB Editor executable with the *.brd* file type.

❍ Open the Allegro Project Manager, your project, and then click *Layout*.

b. On UNIX, use one of the following methods:

❍ At the shell prompt, type:

```
allegro &
```

❍ At the shell prompt, type:

```
allegro
```

❍ Open the Allegro Project Manager, your project, and then click *Layout*.

The *About* <*product name*> splash screen briefly appears.

If you type `allegro &`, it means that Allegro PCB Editor is running in the background. When you use this startup method, you may not see all messages display in the window.

The first time you launch Allegro PCB Editor, the Cadence Product Choices dialog box appears. If you do not enable the *Use As Default* option, the Cadence Product Choices dialog box appears each time you use Allegro PCB Editor.

If you enable the *Use As Default* option, the Cadence Product Choices dialog box no longer appears when you start up. However, you can still change the license. See step 10 of the Procedure on page 71 in Lesson 2-1: Identifying Parts of the User Interface for additional information.

2. Choose *Allegro PCB Design XL*, check the *Use As Default* box, and click *OK*.

   This sets the Allegro PCB Design XL version as your default. This version is used for the exercises in this tutorial.

Important

If you are not licensed for this version, choose the version for which you have a license. Tasks that specifically require Allegro PCB Design XL features will be noted.

Allegro PCB Editor with the layout command menu set appears.

Note: If you are running a release later than Release 15.0, the Allegro PCB Editor has additional features. See Lesson 2-1: Identifying Parts of the User Interface on page 61

Summary

You now know how to start up the Allegro PCB Editor.

You have learned the following:

- **New command:** `allegro &`
- **New window and dialog box:** Allegro PCB Editor Cadence Product Choices dialog box

For More Information

See the *Getting Started with Physical Design* user guide in your documentation set.

What’s Next

Go to Lesson 1-3: Setting Your Working Directory and Opening a Design to learn how to set up your working directory and open a design.

Lesson 1-3: Setting Your Working Directory and Opening a Design

Overview

The first time you start Allegro PCB Editor, the current directory is set to a location specified during the software installation. This directory name appears in the title bar of Allegro PCB Editor. All files that are created or saved from within Allegro PCB Editor are saved to the current directory by default. When you open or save files, you can change the current directory to a directory where you want to save your work.

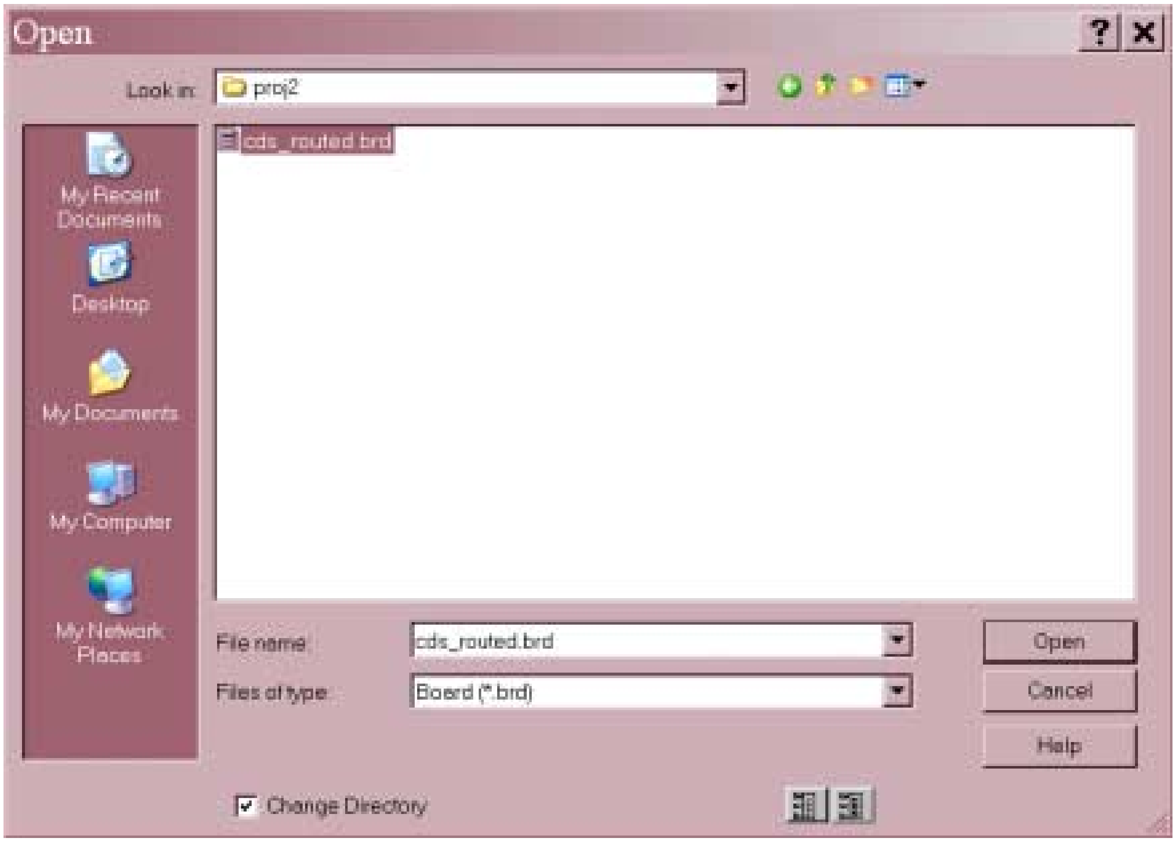
Procedure

1. From the menu bar, choose *File – Open*.

   An Open file browser window appears.

   In Module 2, you will learn to use console window commands and toolbar icons as an alternative to menu commands.

2. Using the directory structure you established in Lesson 1-1: Creating a Project Directory on page 45, navigate to the *proj2* directory.



3. Verify that the *Change Directory* box is checked.

   This option sets your working directory to *proj2*.

   Note: The two buttons below the *Help* button are preview buttons available only in releases later than Release 15.0. When pressed, the left button provides a text preview and the right button provides a graphics preview of the selected design.

4. Choose the `cds_routed.brd` file and click *Open*.

   The `cds_routed.brd` file appears in Allegro PCB Editor.

   Note: You can also open a file by double clicking it if you have associated Allegro PCB Editor with the `.brd` file type.

Summary

You now know how to set a working directory and open a board design.

You have learned the following:

- **New term:** working directory
- **New menu bar command:** *File – Open*

For More Information

See the *Getting Started with Physical Design* user guide in your documentation set.

What's Next

Go to Lesson 1-4: Accessing the Help System to learn how to get help for Allegro PCB Editor.

Lesson 1-4: Accessing the Help System

Overview

The topic-based, application-specific, context-sensitive Cadence® Help online documentation system enables you to navigate, search, and display the contents of multiple document libraries. Cadence Help supports application types that let you run Flash-based multimedia in Web browsers or media viewers and PDF files in Adobe Acrobat.

You can get help in using Allegro PCB Editor with the following methods:

- Help Menu on the Allegro PCB Editor menu bar – When you select *Help – Documentation*, a Help page interface displays all product documentation in tabbed categories so that you can get the information you are looking for quickly. You can also access web resources, including SourceLink and the web site for

Cadence Education Services. For additional information, see Sources of Information on page 42.

- ❑ Documentation tab lists user guide and reference information for key concepts and comprehensive point-of-need information. The Help page opens to this tab by default.
- ❑ Release Info tab lists release-specific information such as What's New, migration documentation, system requirements, and so on.
- ❑ Best Practices tab lists Cadence-recommended practices for key product features and tools.
- ❑ Tutorials tab lists self-paced training lessons in a step-by-step format that teach you how to use the product.
- ❑ Demos tab lists flash-based multimedia videos so that you can watch an example of how to use certain features or processes. Products with many demonstrations may have sub-categories from which to choose on the left-hand side of the Demos tab.

  Note: To view multimedia demonstrations, you need a compatible Flash player. For more information about Flash players that you can download without cost, see

  http://www.macromedia/shockwave/.

- Command Browser – A browser that lists all the editor commands and lets you run the command or obtain help on the command.
- Command console window help – Part of Allegro PCB Editor that lets you enter help and the command name at the command prompt.
- F1 help – Help available when you highlight a menu item and press the F1 function key.

  Note: In versions later than Release 15.1, you can also press the F1 key during an active command to get help.

Demo

You can take a guided multimedia tour that demonstrates Cadence Help and how it displays the user documentation that accompanies your installation of Allegro platform products.

Introducing Cadence Help.

This demonstration runs for approximately 6 minutes.

Procedure

Using the Help Menu

With Cadence Help, you can:

- Open the library of all SPB documents and search these documents.
- Display the *Table of Contents* for the current document.
- Display the *Index* (if available) for the current document.
- View the previous or next chapter in a book.
- View, print, or save to disk the PDF version of the document.
- Search for words and phrases in the library or in a subset of documents.
- Get help on using the Help system.

1. From the Allegro PCB Editor menu bar, choose *Help* to display the menu options.
2. From the menu items, choose *Documentation.* The Documentation tab appears that lists user guide and reference information for key concepts and comprehensive point-of-need information. The Help page opens to this tab by default.
3. Choose the *Release Info* tab and choose *What's New* to display the current product release notes.

   You can review, search, or print the document.

4. Choose the *Allegro Platform Migration Guide* to display information about changes from release to release of software.

   You can review, search, or print the document.

5. From the Allegro PCB Editor menu bar, choose *Help – Web Resources – SourceLink* to open SourceLink, the online Customer Support Center web site.

   For additional information about SourceLink, see SourceLink on page 42.

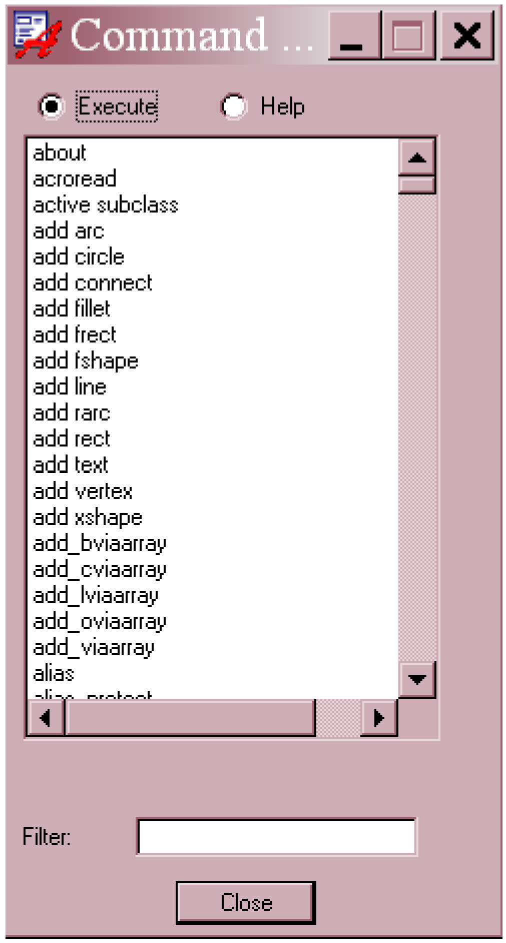
Procedure

Using the Command Browser to Access Help

1. At the console window prompt located at the bottom of Allegro PCB Editor (Command>), type `helpcmd` to display the Command Browser.

   Note: If you are running a release later than Release 15.0, you can also choose *Tools – Utilities – Keyboard Commands* from the menu bar. For additional information, see Lesson 2-1: Identifying Parts of the User Interface on page 61.

   If you click *Execute* (default) in the Command Browser, and then choose a command, Allegro PCB Editor activates the command. If you click *Help,* and then choose a command, Allegro PCB Editor displays help for the command. If you inadvertently execute a command, click the right mouse button in Allegro PCB Editor, and choose *Cancel* from the pop-up menu (in menu-driven editing mode, or verb-noun use model , in which you choose a command, then the design element).



2. In the Command Browser, click *Help* and then click on a command name, for example, `add arc`, to display documentation for the `add arc` command.

3. To limit the display of command selections:

   a. Type a command name or enter a partial string with wildcards, for example, type `?ol*` in the *Filter* text box.

      ? is the wildcard for any single character and ***** for multiple characters.

   b. Press the `Tab` key.

      The result is the following display of commands:
      `color priority`
      `color192`

```
colorview create
colorview load
colorview restore
polar
```

4. Click *Close* to dismiss the Command Browser.

Procedure

Using Other Methods to Access Help

You can access help for a particular command or function using various methods including the console window, F1 function key, and dialog boxes. In this procedure, you will learn how to access help for the *File – Export – Logic* (`feedback` command) menu option. Information on the menu bar and the console window commands is available in “Lesson 2-1: Identifying Parts of the User Interface” on page 61.

1. In the Allegro PCB Editor, type the following at the console window prompt:

   Command> `help feedback`

   The help documentation for the `feedback` command appears in the web browser.

2. Close the web browser.

3. From the Allegro PCB Editor menu bar, simultaneously, highlight *File – Export – Logic* and press the F1 function key.

   The help documentation for the `feedback` command appears again.

4. Close Cadence Help.

5. From the Allegro PCB Editor menu bar, choose *File – Export – Logic*.

   The Export Logic dialog box appears.

6. Click *Help* in the dialog box to display the help documentation for the `feedback` command.

7. Close the Export Logic dialog box.

8. Close the web browser.

Summary

You now know how to access help using the Help menu, Command Browser, console window, F1 function key, and dialog boxes.

- **New menu commands:** *Help – Documentation, Help – Web Resources – SourceLink*, *Tools – Utilities – Keyboard Commands, File – Export – Logic*
- **New console commands:** `helpcmd, help feedback, add arc`
- **New function key commands**: `F1`
- **New window and dialog box:** Cadence Help Viewer, Export Logic dialog box
- **New documents:** *Allegro PCB Design Editor: What's New, Migration Guide for Allegro Platform Products*, *Allegro PCB and Package Physical Layout Command Reference*

What's Next

Go to Module 2: Introducing the Allegro PCB Editor User Interface to learn about the various components of the Allegro PCB Editor user interface.

2

Module 2: Introducing the Allegro PCB Editor User Interface

This module comprises these lessons:

- Lesson 2-1: Identifying Parts of the User Interface on page 61
- Lesson 2-2: Accessing Pop-up Menus and Panning a Design on page 74
- Lesson 2-3: Zooming In and Out of a Design on page 77
- Lesson 2-4: Using Other Methods to Zoom In and Out of a Design on page 79

Completion Time

It should take approximately 90 minutes to complete the written lessons in this module.

Lesson 2-1: Identifying Parts of the User Interface

Overview

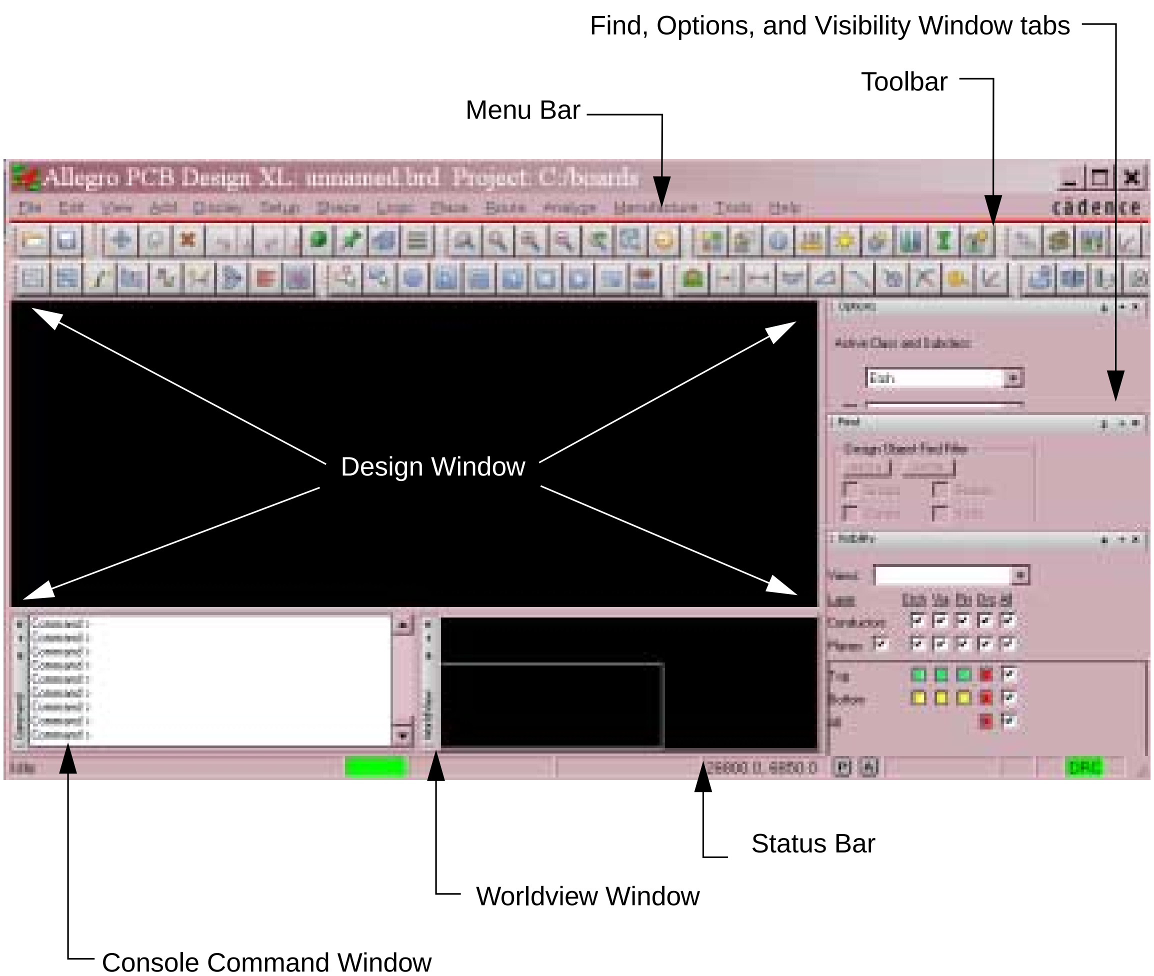
Allegro PCB Editor appears when you start up the software. The modes (Layout, Symbol) available to you depend on the task you are performing and the Allegro PCB Editor product you are running.

The task-oriented user interface has the following components:

- *Design Window*
- *Menu Bar*
- *Toolbar*

- *Control Panel* with these foldable window panes
  - *Options*
  - *Find*
  - *Visibility*
- *Command* foldable window pane
- *Worldview* foldable window pane
- *Status* bar

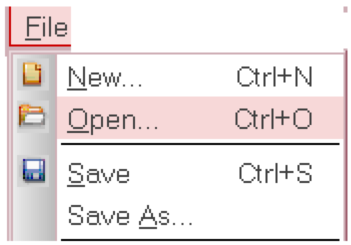
The following example shows Allegro PCB Editor in layout mode.



The following list describes the components of Allegro PCB Editor:

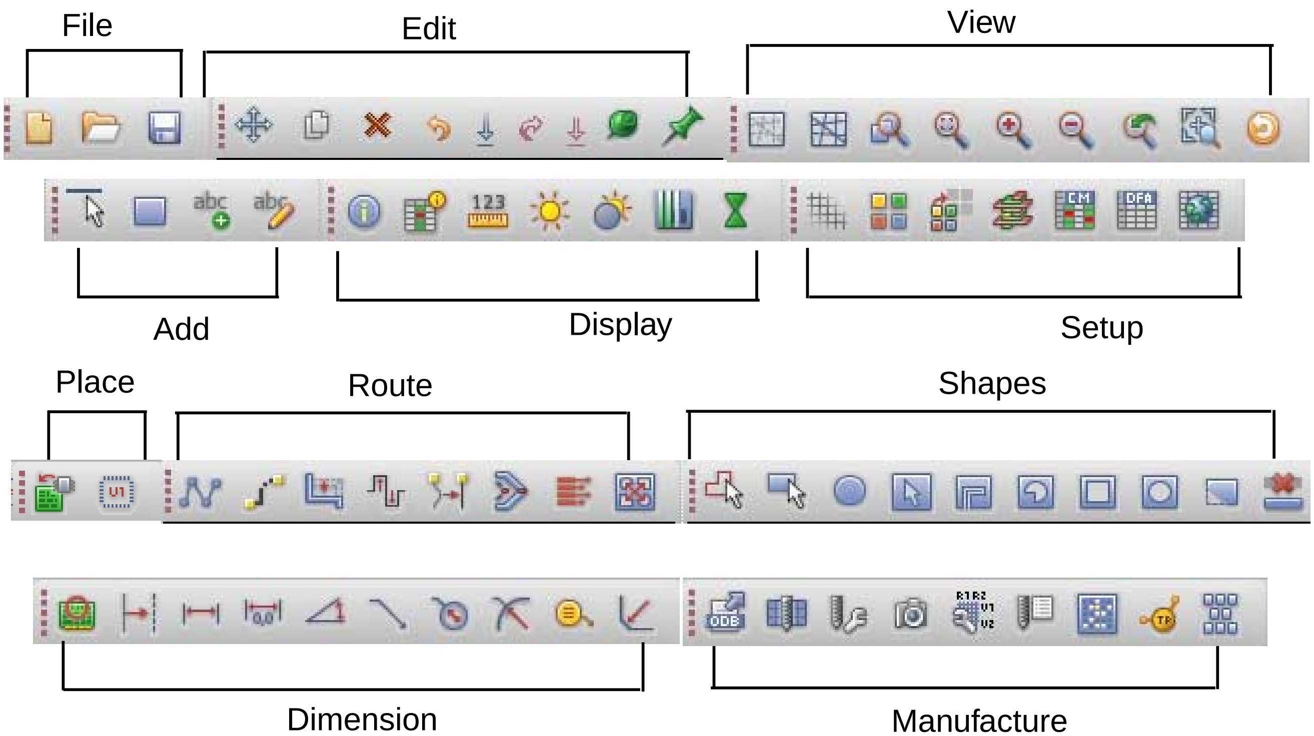
- **Menu bar** – Located below the title bar is the menu set. Allegro PCB Editor has two menu sets: Layout mode, used for general design work, and Symbol mode, used for the creation and modification of symbols. These menu sets differ, based on the product in which you are working. The pull-down menus contain all the commands required to create and modify a design.

  You can also use the accelerator key combinations to execute some commands. The key combinations appear in the pull-down menus, to the right of the command. For example, to open a file, choose *File – Open* from the menu bar or press `Ctrl+O`.



  As an alternative to using the menu items, you can use console commands. You can see the corresponding console command displayed in the Status bar when you choose a menu item. For example, when you choose the menu item *File – Open*, the corresponding console command, `open`, appears in the Status bar. See the descriptions of Command console window and Status bar in the following sections.

- **Icon toolbar** – Located below the menu bar, the toolbar contains icons that give you a quick way to access common Allegro PCB Editor commands. Dock or undock any toolbar by left-clicking on the small circles, or grippers, next to it and moving it. To learn a toolbar icon's function, position the cursor over the icon without depressing the mouse button and view its description in the tool tip that appears. Icons can be customized to suit specific needs.The labels indicate groups of icons (toolset) that correspond to functions you can perform using the menu or submenu names. For example, the first icon in the File toolset corresponds to the *File – Open* menu item.



- **Design window** – Located below the icon toolbars, the Design window is the graphical display area where you do most of your design work. You can change the default background color of the Design window. See **Lesson 3-4: Controlling Colors and Dimming Graphics** on page 100.

- **Command console window** – Located at the bottom left portion of the Allegro PCB Editor, this window has two functions. It displays messages, and prompts you when you choose menu items to perform tasks. In this window, you can also type Allegro PCB Editor console commands at the prompt as an alternative to using the menus. Using the pin icon, you can "pin" this window so it remains visible while unpinned windows remain as tabs bordering the design window.

- **Status bar** – Located to the right of the Command console window, the Status bar contains the current command being executed. In this case, the word *Idle* appears because no command is currently active. It also shows the current x, y coordinates of the cross hairs. These coordinates change as you move the mouse.

  Note: If you are running a release later than Release 15.0, you can access the *P* and *A* buttons in the Status bar.

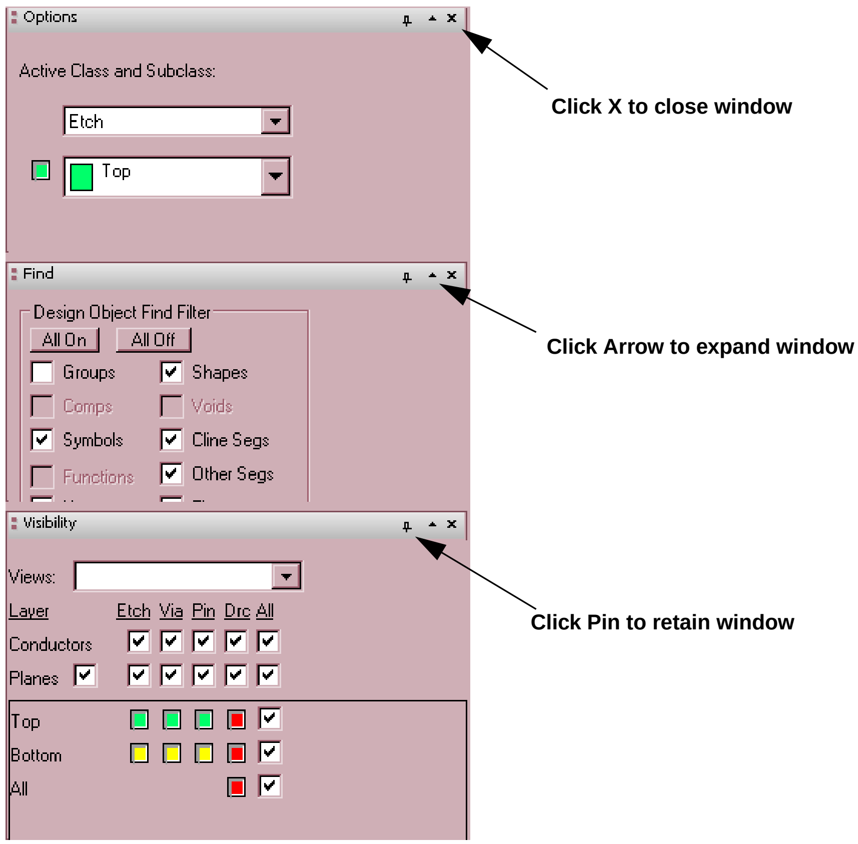
  The *P* button allows you to display a dialog box. When you click this button, and you are in an interactive command, for example, `add connect`, the Pick dialog box appears and remains displayed until you dismiss it. If the status is *Idle,* and you click the *P* button, the Zoom Center dialog box appears and remains displayed until you dismiss it. You can enter specific or incremental values in these dialog boxes.

  The *A* button allows toggling of the x, y read-out from absolute mode to relative mode. When you are in absolute mode, the x, y coordinates location is from the origin of the board. When you are in relative mode, the origin is always from the last pick and the button is labelled *R*. Allegro PCB Editor always starts designs in absolute mode.

  The Status bar has a *Stop* button, which you can use to interrupt the current command. The *Stop* button is presently unavailable because there is no command currently active.

  In the Status bar, a green box means that Allegro PCB Editor is ready for your command. If the box is yellow, it means that the system is working—but you can interrupt the system by clicking the *Stop* button, pressing `Ctrl-C`, or pressing the `Esc` key. If the box is red, the system is working and you are unable to interrupt it. You must wait until the box turns either yellow or green again.

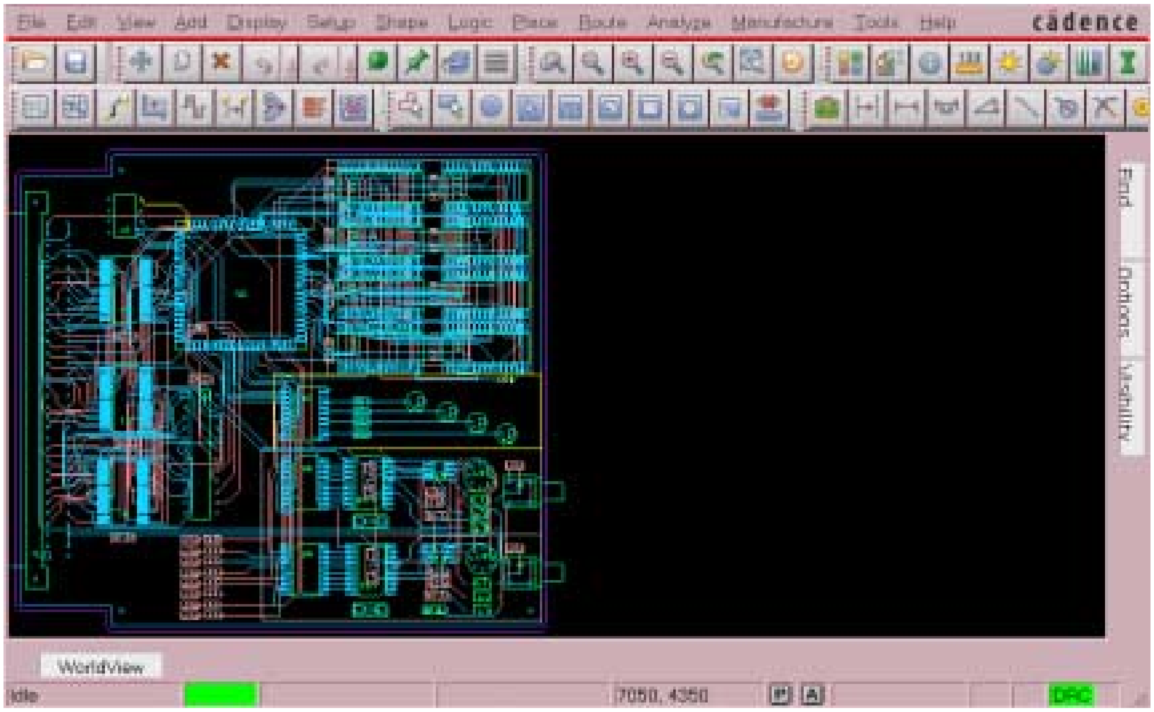
- **Control Panel** – The area to the right of the Design window contains the Control Panel, which features foldable *Options, Find*, and *Visibility* window panes that may be quickly resized, stacked, or relocated to maximize the working design area. Using the pin icon, you can “pin” a window so it remains visible, as below.



Working with Foldable Windows

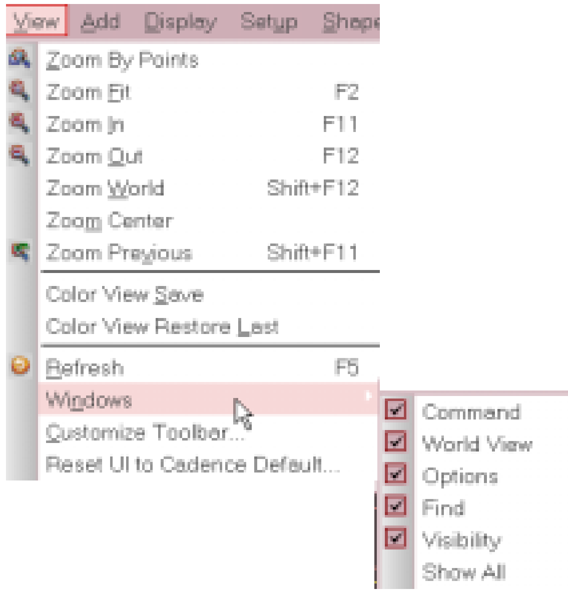
The foldable windows are particularly useful on a single monitor setup because they provide more work space, while giving the designer the option of seeing the window-pane information by simply hovering over the tabs bordering the design window. Passing the cursor over any of them quickly unfolds the tab for viewing or editing, then retracts it.

Clicking the pin icon when a window is visible causes it to revert to a tab bordering the design window as shown below.



Dock or undock the window by left-clicking to choose it and moving it anywhere within or outside the design window. In a dual-monitor system, undocking windows are useful as they can be moved to the second monitor, maximizing the work space.

You control the visibility of these windows by clicking an arrow to expand a docked window pane, clicking the X to hide it, or by using the *View – Windows* menu choices to hide or display it.



The Control Panel has three foldable window panes:

- The *Options* window pane contains parameters used to control the current interactive command.

- The *Find* window pane, sometimes referred to as the Find Filter, lets you select the objects that will be affected by the active command. You can use this window pane when selecting items with the mouse or when selecting items using the *Find By Name* box.

  Basic building blocks for use in board design, also listed in the *Find* window pane, are described below.

Design Object	Description
Groups	One or more objects linked together so that you can easily perform commands on them.
Comps	The combination of a symbol and logical description of a part.
Symbols	The physical description of a part, such as pins, part outline, and so on.

Design Object	Description
Functions	A logical unit of an electronic part such as an integrated circuit, also referred to as a gate.
Nets	The signal name associated with a component pin.
Pins	Numbered electrical connection points (pads) on a symbol or component. Non-electrical pins on mechanical symbols or components do not have pin numbers.
Vias	The physical mechanism to traverse layers when connecting a net.
Connect LInes, (Clines)	A conductor trace associated with a net name. It begins and ends on a pin, via, or Tpoint.
Lines	A graphical line.
Shapes	A closed polygon. This shape may be used to represent internal power planes, keepout areas, keepin areas, and so on.
Voids	Non-copper polygon or circle within an etch layer shape.
Cline Segs	A portion of a cline. The segment is from one vertex (bend) point to the next vertex point (*Route – Connect* command).
Other Segs	Non-cline such as an arc, circle, and line (*Add* menu).
Figures	Pre-defined shapes that can be assigned to objects such as drill symbols. Found in padstack parameters.
DRC errors	Markers placed in the design to indicate errors after design rule checking (DRC) takes place.

Design Object	Description
Text	Annotation for design objects.
Ratsnests	Lines that show logical connections between two pins, clines, or vias in a design drawing. Elements connected by the same ratsnest line are part of the same net. The ratsnest shows the circuit logic and for Emitter Coupled Logic (ECL) circuits, the order in which pins are to be connected.
Rat Ts	Database objects used to insert a branch in nets' schedules at some point other than at a component pin. A rat T has a physical location that is often an approximate location for a 'T' or a via in the net's physical interconnect.

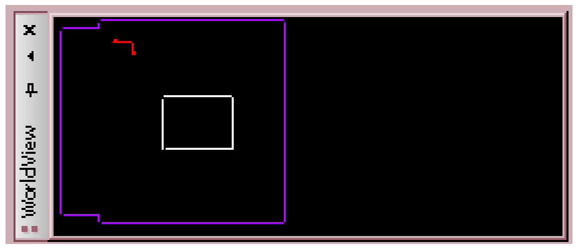
- The *Visibility* window pane lets you control the visibility of conductor objects in your design such as etch, pins, vias and so on.

For information on using the Control Panel, see Lesson 3-3: Controlling Etch Visibility on page 97 and **Lesson 3-5: Using the Control Panel to Manipulate Design Objects** on page 104**.** For information on customizing the Control Panel, see Lesson 5-1: Customizing Your View and Toolset on page 155.

The World View window pane shows the board outline and the portion of the board where you are currently zoomed in. It gives you quick and convenient access to the panning and zooming commands. For additional information on zooming and panning, see Lesson 2-2: Accessing Pop-up Menus and Panning a Design on page 74.

Current Display Window

Board Outline



Pop-up Menu (right click)

In this lesson, you will perform tasks to become familiar with the Allegro PCB Editor user interface.

Procedure

1. If it is not already displayed in Allegro PCB Editor, open `cds_routed.brd`.

   Note: If you are running a release later than Release 15.0 and you previously opened this board file, you can also choose *File – Recent Designs* from the menu bar and choose the board file.

2. Maximize the editor to full screen mode, if it is not already maximized.

3. Referring to the information in the section, Overview on page 61, identify the following parts of the editor:

   - ❑ Menu bar and menu items
   - ❑ Icon toolbar
   - ❑ Design window
   - ❑ Command console window (and command line)
   - ❑ Status bar with its traffic light and coordinate readouts
   - ❑ Control Panel: *Options*, *Find*, and *Visibility* window panes
   - ❑ World View window pane

4. View the menu options. Choose the *File* menu and note the available menu items. Slowly pass your cursor over the menus (*Edit, View, Add* and so on) from left to right. Note the various menu items available under each menu. For additional information on these menus, see Allegro PCB Editor Menus and Functions on page 39.

5. Click the left mouse button in the Design window to close your latest pull-down menu.

6. Slowly drag (do not click) your cursor across the toolbar from left to right and read the tool tips that appear.

   You can customize icon displays to suit specific needs. For additional information, see **Lesson 5-1: Customizing Your View and Toolset** on page 155.

7. When you come to the *Zoom Fit (F2)* icon, click it.

   The entire `cds_routed.brd` design is framed in the Design window.

8. Choose menu items to run commands, while viewing the corresponding console command name in the Status bar. Click the right mouse button in the Design window and click *Cancel* on the pop-up menu to inactivate a command.

9. Find commands that cause the status to be red. Then find commands that cause the status to be yellow.

10. Use one of these commands to change editors:

    ❑ From the menu bar, choose *File – Change Editor.*

    ❑ At the console window prompt, type `toolswap`.

    The Cadence Product Choices dialog box appears.

11. Choose one of the options and click *OK*.

    The editor changes.

12. Use one of these commands to change back to the previous editor:

    ❑ From the menu bar, choose *File – Change Editor.*

    ❑ At the console window prompt, type `toolswap`.

13. Use one of these commands to exit Allegro PCB Editor:

- ❑ From the menu bar, choose *File – Exit.*
- ❑ At the console window prompt, type `exit`.

Summary

You now can identify the different parts of Allegro PCB Editor user interface and are familiar with Allegro PCB Editor menus and menu items.

You have learned the following:

- **New terms:** absolute mode, relative mode
- **New menu commands:** *Route – Connect, File – Open, View – Customize – Toolbar, File – Change Editor, File – Exit*
- **New console commands:** `toolswap, add connect, exit`
- **New toolbar icon:** 
- **New function key command:** `F2`
- **Parts of the user interface:** Layout mode, Symbol mode, menu bar, icon toolbar, Design window, Command console window, Status bar, Control Panel, *Options* window pane, *Find* window pane (Find Filter), *Visibility* window pane, World View window pane, Placement toolbar, Route toolbar, Analysis toolbar, Manufacturing toolbar

For More Information

See

- the *Getting Started with Physical Design* user guide in your documentation set.
- Pick dialog box in the *Allegro PCB and Package Physical Layout Command Reference*.
- `toolswap` command in the *Allegro PCB and Package Physical Layout Command Reference*.

- `add connect` command in the *Allegro PCB and Package Physical Layout Command Reference*.
- `exit` command in the *Allegro PCB and Package Physical Layout Command Reference*.

What's Next

Go to Lesson 2-2: Accessing Pop-up Menus and Panning a Design to learn how to access pop-up menus and pan a design.

Lesson 2-2: Accessing Pop-up Menus and Panning a Design

Overview

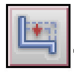
Allegro PCB Editor supports panning, or roaming a design, which is the action of moving across a design in the editor window. You can pan a design using a mouse device or arrow keys on the keyboard.

Allegro PCB Editor supports all mouse devices that have at least two buttons and are supported by your workstation. For additional information on mouse devices and panning, see the *Getting Started with Physical Design* user guide in your documentation set.

In this lesson, you will learn how to access pop-up menus and pan a design.

Procedure

1. Start Allegro PCB Editor. If necessary, see Lesson 1-2: Starting Up Allegro PCB Editor on page 49.
2. Open `cds_routed.brd`.
3. Use one of these commands to activate the `slide` command:
   - ❑ From the menu bar, using the left mouse button, choose *Route – Slide.*

- ❑ At the console window prompt, type `slide.`
- ❑ Click .
- ❑ Press SF3 `(Shift + F3)`.

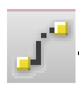
Notice that the `slide` command is listed in the Status bar.

4. Move your cursor into the editor window and click the right mouse button.

A pop-up menu appears. The contents of the pop-up menu varies with the menu item or command you are using. For example, this pop-up menu differs from the pop-up menu displayed when you choose *Route – Connect* (`add connect` command).

5. Choose *Cancel* from the pop-up menu to exit the *Route – Slide* function. You can also press the F9 function key.

6. Use one of these commands to activate the `add connect` command:

- ❑ From the menu bar, choose *Route – Connect.*
- ❑ At the console window prompt, type `add connect.`
- ❑ Click .
- ❑ Press F3.

7. Move your cursor into the editor window, click the right mouse button, and examine the pop-up menu.

8. Choose *Cancel* from the pop-up menu to exit the *Route – Connect* command, or press the F4 function key.

9. Place the cursor in the editor window. Press and hold the middle mouse button down and slide the mouse to the left, right, up, and down.

If you have a two-button mouse, you can press and hold the `Shift` key while you hold the right mouse button down and slide the mouse. Note that this functionality works in menu-driven editing mode. Ensure that you are working with the tutorial using *Setup –Application Mode – None* (`noappmode` command).

Notice how the design shifts in the direction of your cursor movement. This is *panning*. Also notice how the view changes in the World View window, located at the bottom right of the editor.

10. Use the arrow keys on your keyboard to pan the design.

11. To control the amount of movement when panning using the arrow keys:

a. Choose *Setup – User Preferences* (`enved` command).

b. Click *Roam* in the *Categories* section.

c. Set a value for the `roaminc` environment variable and click *OK*.

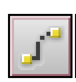
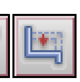
The default value is 96.

See Lesson 5-3: Setting Environment Variables on page 162 for additional information.

Summary

You now know how to access pop-up menus when you are in command mode, and pan a design.

You have learned the following:

- **New term**: panning
- **New menu commands:** *Route – Slide, Setup – User Preferences*
- **New console commands:** `slide, add connect, enved`
- **New toolbar icons:**  
- **New function key commands:** `F4, SF6`
- **New environment variable:** `roaminc`

For More Information

See the *Getting Started with Physical Design* user guide in your documentation set.

What’s Next

Go to Lesson 2-3: Zooming In and Out of a Design.

Lesson 2-3: Zooming In and Out of a Design

Overview

Allegro PCB Editor supports zooming in and out of a design. You can zoom using a mouse device, menu items, console commands, icons, function keys, or the World View window.

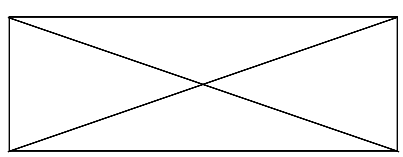
In this lesson, you will learn how to zoom in and out of a design using the middle mouse button on your mouse device.

Note: The steps in this procedure involve a three-button mouse. If you have a mouse with programmable keys, for example, a two-button wheel mouse, the mouse behavior may be different based on your settings. See the *Getting Started with Physical Design* user guide in your documentation set.

Procedure

1. If it is not already displayed in the editor window, open `cds_routed.brd`.
2. Place the cursor in the bottom right portion of the Design window. Press but do *not* hold the middle mouse button in the window.
3. Move your cursor toward the top left portion of the window.

   Notice as you move your cursor that a rectangle is drawn. This represents what will be the new display area.



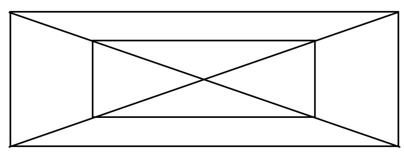
4. Select again with the middle mouse button or the left mouse button.

   The area that was contained within the white rectangle now becomes your new display area.

5. Select again with the middle mouse button somewhere in the middle of the display area. Do not hold down the middle mouse button.

6. Move your cursor slowly toward the bottom right.

   As you move your cursor, two white rectangles are drawn. The inside rectangle represents the original display area. The outside rectangle represents a zoom-out magnification. The further the outside rectangle is away from the inside rectangle, the greater the zoom-out. As you move your mouse, you see your work area temporarily refresh. This temporary redisplay represents what will be the new work area.



7. Select again with the middle mouse button or the left mouse button.

   The board in the window is redrawn to match the current zoom.

Note: To disable the dynamic zoom feature, set the `no_dynamic_zoom` environment variable in the *Input* category of the User Preferences Editor. By setting this variable, middle-button functionality is limited to zooming in or zooming out. See Lesson 5-3: Setting Environment Variables on page 162 for additional information.

Summary

You now know how to zoom in and out of a design using the middle mouse button on your mouse device, and how to disable the dynamic zoom feature.

You have learned the following:

- **New term:** zooming
- **New environment variable:** `no_dynamic_zoom`

For More Information

See the *Getting Started with Physical Design* user guide in your documentation set.

What's Next

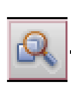
Go to Lesson 2-4: Using Other Methods to Zoom In and Out of a Design to learn the various methods for zooming in and out of a design.

Lesson 2-4: Using Other Methods to Zoom In and Out of a Design

Overview

In addition to using a mouse for zooming in and out of a design, you can also use other methods. In this lesson, you will use menu items, toolbar icons, console commands, and function keys to zoom in and out of a design.

Procedure

1. If it is not already displayed in the window, open `cds_routed.brd`.
2. Use one of these commands to specify a new display area by picking two diagonally opposed points:
   - ❑ From the menu bar, choose *View – Zoom By Points*.
   - ❑ At the console window prompt, type `zoom points.`
   - ❑ Click .

Notice that the `zoom points` command is listed in the Status bar. In the Command console window, you are prompted to pick the first corner of a new view window.

3. Click to place the first corner of the new window.

As you move your cursor, a rectangle with inscribed diagonals representing the new window forms.

4. Click again to fix the size of the new window.

The window zooms to display only the area you just outlined within the rectangle.

5. Use one of these commands to display the entire extents of the drawing in the window:

- ❑ From the menu bar, choose *View – Zoom World*.
- ❑ At the console window prompt, type `zoom world`.

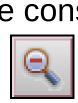
6. Use one of these commands to create a view that includes but is no larger than the board:

- ❑ From the menu bar, choose *View – Zoom Fit*.
- ❑ At the console window prompt, type `zoom fit`.
- ❑ Click .
- ❑ Press F2.

7. Use one of these commands to magnify or zoom into a smaller area of the drawing that remains centered about the same point:

- ❑ From the menu bar, choose *View – Zoom In*.
- ❑ At the console window prompt, type `zoom in`.
- ❑ Click .
- ❑ Press F11.

8. Use one of the these commands to increase the displayed area of the drawing.

- ❑ From the menu bar, choose *View – Zoom Out*.
- ❑ At the console window prompt, type `zoom out`.
- ❑ Click .

- ❑ Press F12.

  This shows more data in the window and makes objects become smaller.

9. Use one of these commands to fit the design in the window:

- ❑ From the menu bar, choose *View – Zoom Fit*.
- ❑ At the console window prompt, type `zoom fit`.
- ❑ Click .
- ❑ Press F2.

Summary

You now know how to zoom in and out of a design using menu items, toolbar icons, console commands, and function keys.

You have learned the following:

- **New menu commands:** *View – Zoom By Points, View – Zoom World, View – Zoom Fit, View – Zoom In, View – Zoom Out*
- **New console commands:** `zoom points, zoom world, zoom fit, zoom in, zoom out`
- **New toolbar icons:**    
- **New function key commands:** `F2, F11, F12`

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `zoom points` command in the *Allegro PCB and Package Physical Layout Command Reference*.
- `zoom world` command in the *Allegro PCB and Package Physical Layout Command Reference*.

- `zoom fit` command in the *Allegro PCB and Package Physical Layout Command Reference*.
- `zoom in` command in the *Allegro PCB and Package Physical Layout Command Reference*.
- `zoom out` command in the *Allegro PCB and Package Physical Layout Command Reference*.

What's Next

Go to Module 3: Using Allegro PCB Editor Control Functions to learn some control functions for Allegro PCB Editor.

3

Module 3: Using Allegro PCB Editor Control Functions

This module comprises these lessons:

- Lesson 3-1: Changing the Cursor Display on page 83
- Lesson 3-2: Controlling Color and Visibility on page 86
- Lesson 3-3: Controlling Etch Visibility on page 97
- Lesson 3-4: Controlling Colors and Dimming Graphics on page 100
- Lesson 3-5: Using the Control Panel to Manipulate Design Objects on page 104
- Lesson 3-6: Highlighting Objects on page 113
- Lesson 3-7: Listing Detailed Information About a Specified Object on page 115
- Lesson 3-8: Measuring Distance Between Objects on page 118

Completion Time

It should take approximately 150 minutes to complete the written lessons in this module.

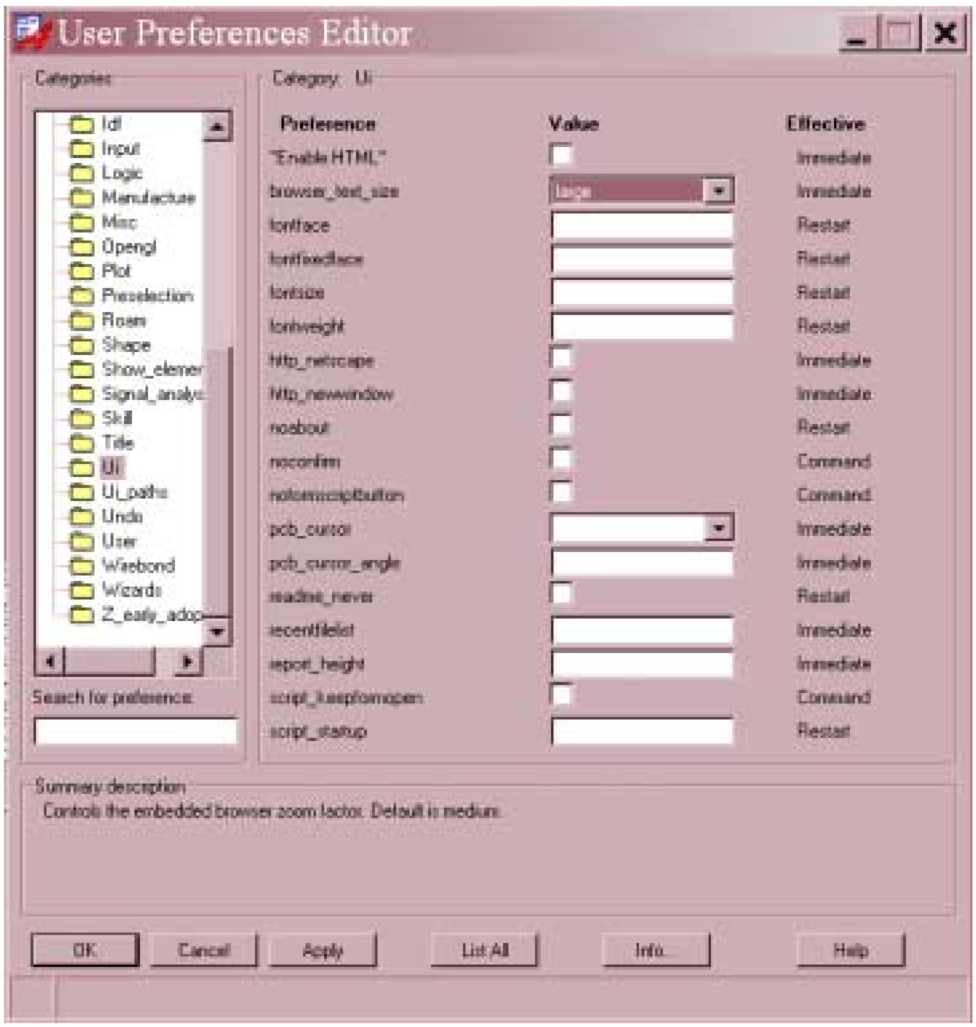
Lesson 3-1: Changing the Cursor Display

Overview

By default, Allegro PCB Editor sets the cursor to cross hair. In this lesson, you will change the cursor to infinite so that you can better line up components in a design.

Procedure

1. If it is not already displayed in Allegro PCB Editor, open `cds_routed.brd`.
2. Use one of these commands to display the User Preferences Editor:
   - ❑ From the menu bar, choose *Setup – User Preferences.*
   - ❑ At the console window prompt, type `enved`.



3. Click *UI* in the *Categories* section.

4. In the *Value* column, click the drop-down list next to the *pcb_cursor* preference and choose *infinite*.

   The *Effective* field states that this change takes effect immediately. The *Summary description* field at the bottom left corner displays a short description of the action that occurs when you click a check box or place the cursor in a text box. The *List All* button displays a viewer listing all current environment variables. The *Info…* button summarizes help descriptions for all environment variables.

5. Click *OK* to save the change and close the dialog box.

   The cursor changes to infinite and now spans the height and width of your window. The change you made is stored in the `env` file located in the `pcbenv` directory in your home directory. This setting is used each time you start up Allegro PCB Editor. For additional information on setting preferences, see Lesson 5-3: Setting Environment Variables on page 162.

6. Reverse your choices to change the cursor back to a cross hair.

Summary

You are now able to modify the cursor display.

You have learned the following:

- **New terms:** cross hair, infinite
- **New menu bar command:** *Setup – User Preferences*
- **New console command:** `enved`
- **New dialog box:** User Preferences Editor

For More Information

See:

- enved command in the *Allegro PCB and Package Physical Layout Command Reference.*
- *env File*.

What's Next

Go to Lesson 3-2: Controlling Color and Visibility to learn about classes and subclasses, and how to turn on the visibility of classes and subclasses, apply colors to subclasses, customize colors, and save color settings.

Lesson 3-2: Controlling Color and Visibility

Overview

Design elements are categorized by class and subclass. Classes are major categories defining the purpose of the design elements. Subclasses further define the purpose of the design elements. You can control how each class and subclass appears in the Design window. You can determine which colors to use for the subclasses.

Allegro PCB Editor supports a palette of 192 modifiable colors, 96 of which display at one time in a primary color palette, which is the Cadence default, and another 96 which display in secondary palette, used for customization. The first 24 positions are reserved for colors used in pre-16.0 databases.

Choose *Display – Color/Visibility* (`color192` command) and use the Color dialog box to:

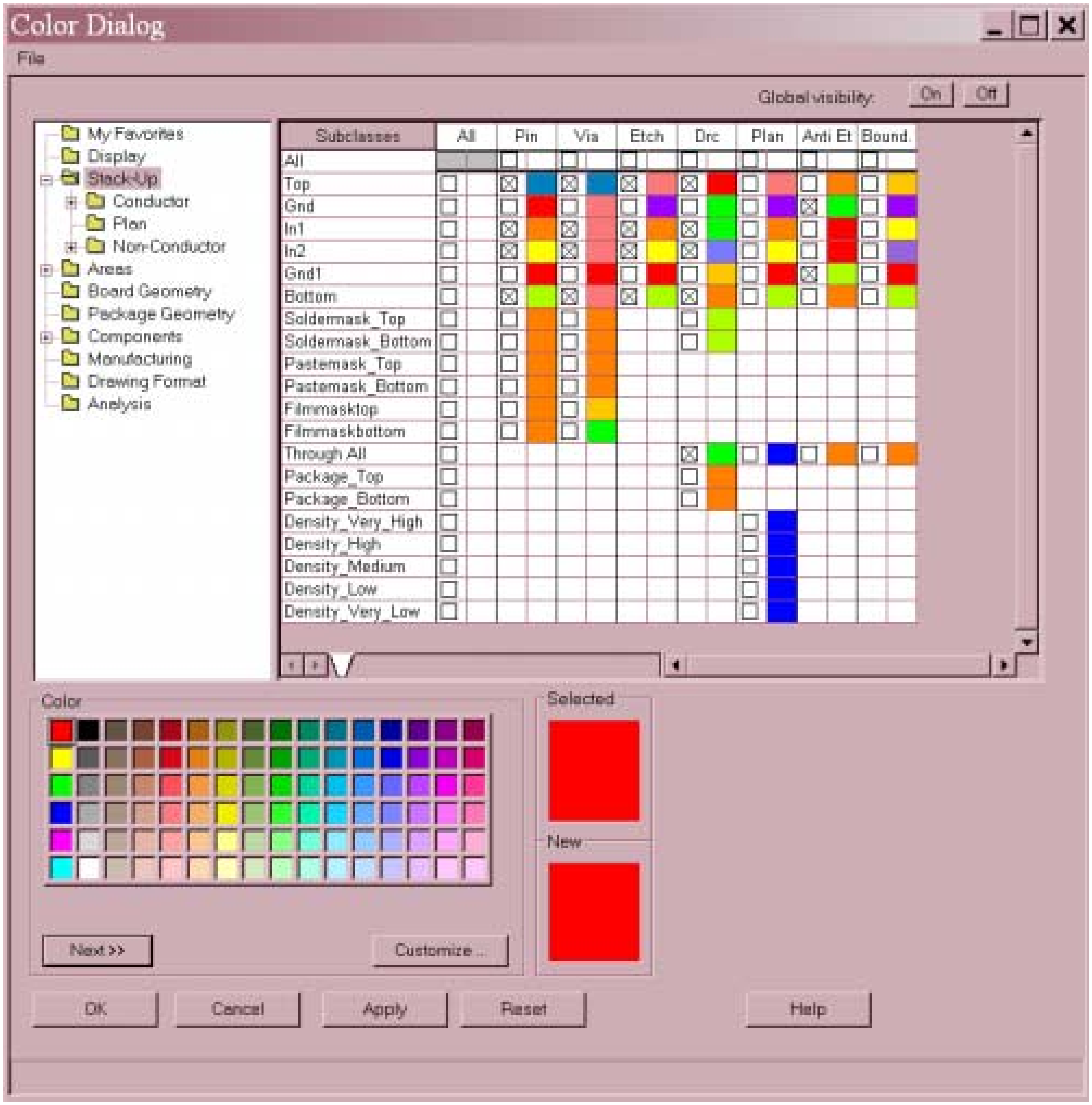
- Assign colors to subclasses, ratsnest lines, and highlighting schemes
- Control visibility of individual classes and subclasses
- Set graphics transparency levels
- Create a *My Favorites* folder to store frequently used subclasses for which visibility or color changes often
- Apply differentiating colors to side-centric graphical views of ratsnests
- Save customized color palettes for reuse
- Prioritize subclasses display
- Customize shades and hues of color

The Left Pane displays each class associated with a group. The color and visibility of the subclasses associated with that class display horizontally.

For all but the *Display* group, which has no associated classes or subclasses, each row lists a subclass. An X indicates the subclass is visible. The color box indicates the color assigned to the subclass element.

Clicking the *All* column or *All* row enables visibility for the entire row or column.

Clicking the intersection of the *All* row and *All* column cell (*All/ll* cell) enables visibility globally. By default, subclasses are visible.



In the following procedures, you will learn how to perform these tasks:

- Enabling Visibility of Classes and Subclasses

- Customizing Colors

Procedures

Enabling Visibility of Classes and Subclasses

1. From the menu bar, choose *File – Save As* to save the `cds_routed.brd` file as `cds_routed1.brd`.

   Note: OrCAD Demo CD users will not be able to save the `cds_routed.brd` database.

2. With `cds_routed1.brd` displayed in the Design window, use one of these commands to display the Color dialog box:

   - From the menu bar, choose *Display – Color/Visibility.*
   - At the console window prompt, type `color192`.
   - At the icon toolbar, click .

   The dialog box shows all the groups and their associated classes and subclasses in Allegro PCB Editor. Groups include:

   - Display
   - Stack-Up
   - Areas
   - Board Geometry
   - Package Geometry
   - Components
   - Manufacturing
   - Analysis

3. Notice that the class names in each group are listed vertically beneath the group name in the left pane of the dialog box, and the subclass names are listed vertically at the right pane of the dialog box, with the class names across the top. Each class also has an *All* check box. If checked, it automatically checks all the subclasses within the class. Each subclass has a visibility check box and a color push button.

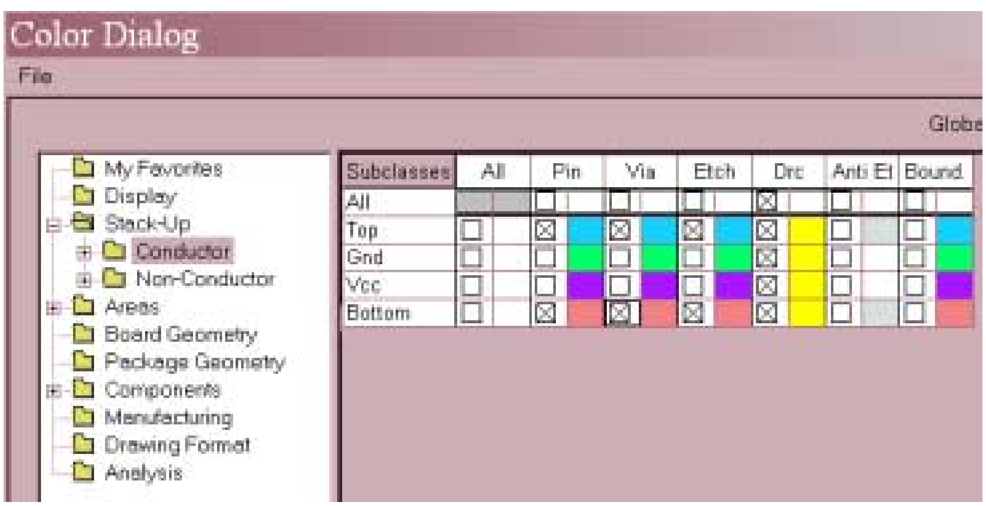
4. At the top right of the Color dialog box, click *Off* under *Global Visibility.*

   When the Allegro dialog box appears, click *Yes* to change all classes and subclasses to invisible. Notice that the check marks are removed from the visibility boxes. This action resets all the colors to off, so you can begin setting them to the colors you want.

5. Click *Apply*.
6. Choose the *Components* group.
7. Under the *Ref Des* class, check the visibility box for the subclass *Assembly_Top.*

   A check mark in the box indicates that the visibility for that subclass is turned on.

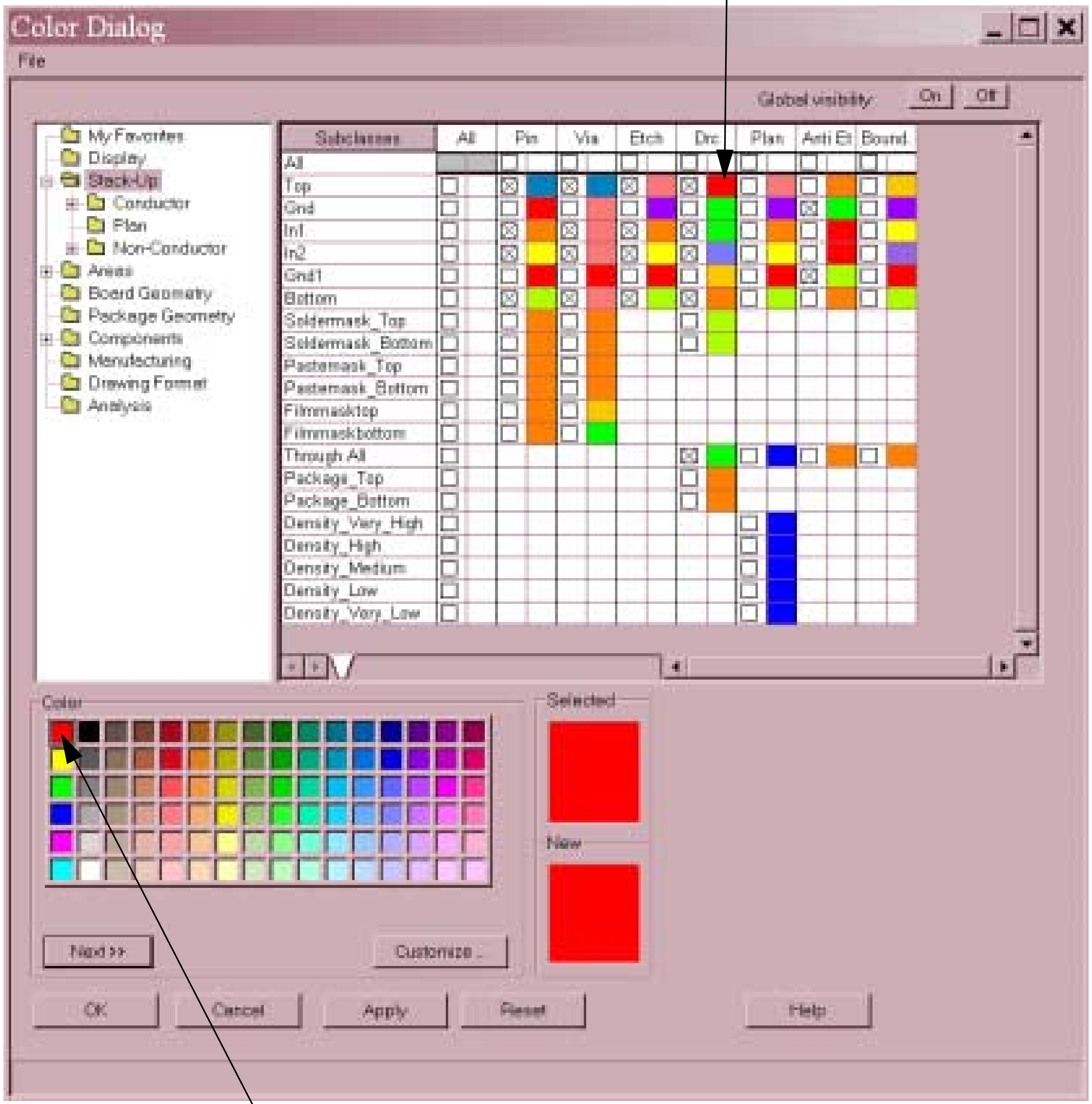
8. Under *Board Geometry*, check the visibility box for the *Outline* subclass.
9. Under *Package Geometry*, check the visibility box for the *Assembly_Top* subclass.
10. Under *Stack-Up*, check the visibility boxes or subclasses in this group, as shown in the Color dialog box below, then click *Apply*.
11. Close the Color dialog box by clicking *OK*.



Assigning Colors to Subclasses

1. If it is not already displayed in Allegro PCB Editor, open `cds_routed.brd`. With the Color dialog box open, choose a folder from the left pane that contains the subclass whose color you want to change.
2. In the *Color* section of the dialog box, use *Next >>* or *Prev >>* to display the primary or secondary color palettes.
3. Click the new color in the *Color* section. (This is the first click identified in the figure).

Second click



First click

The *New* color box shows the color you have chosen.

4. Click the color box next to the subclass whose color you want to change. (This is the second click in the figure.) The color box for

the subclass changes to the color you chose from the *Color* section.

5. Click *Apply* to update the drawing and continue changing colors.

   a. To apply the color to an entire row or column, click the color box next to the *All* column or *All* row.

   b. To apply the color globally, click the color box next to the intersection of the *All* row and *All* column cell (*All/All* cell).

6. Click *OK* to save changes and close the dialog box.

   The *Options* foldable window pane displays the color assigned to a subclass in a color box next to the subclass name.

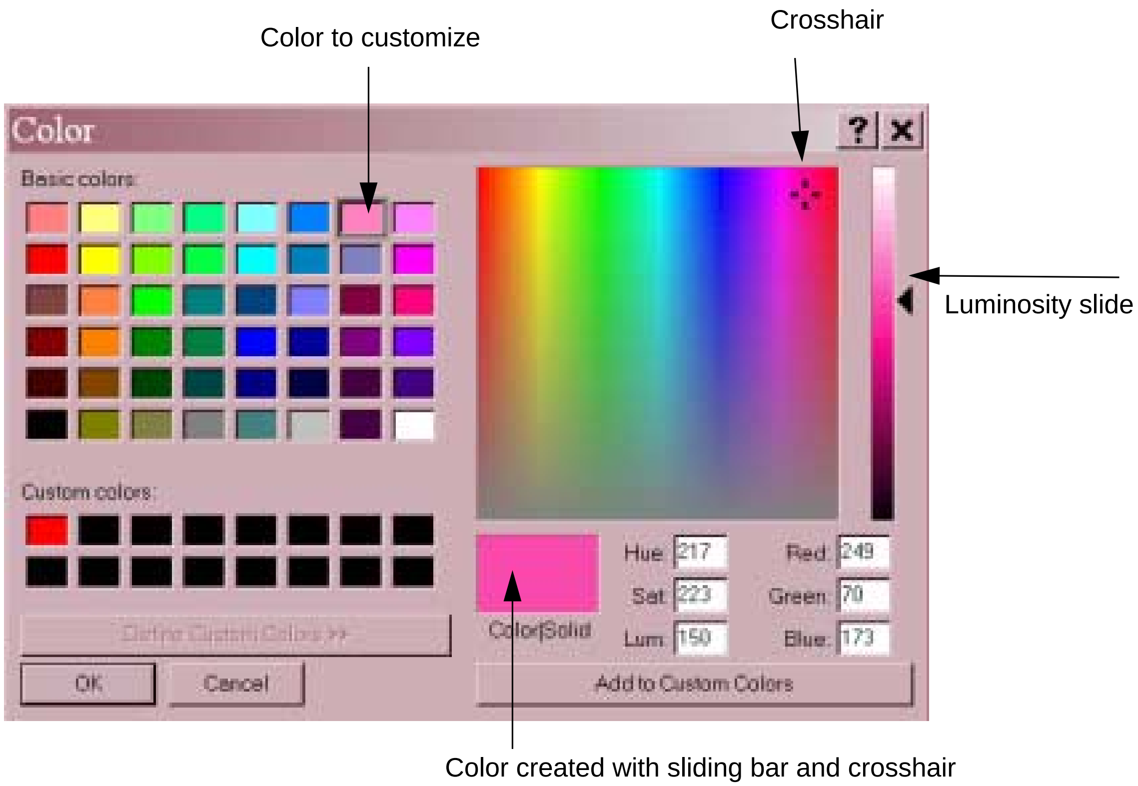
Customizing Colors

1. With `cds_routed1.brd` displayed in the Design window, choose *Display – Color/Visibility*.

   The Color dialog box appears.

2. In the *Color* section of the dialog box, click the color box for the color you want to change.

3. Click *Customize*.



4. Choose a new color from the *Basic Colors* section or the *Custom Colors* section.

5. Move the control on the right hand side vertical sliding bar for luminosity away from the extremes of white or black. The *Hue, Sat, Lum, Red, Green,* and *Blue* fields display the numerical color values for the color chosen.

6. Move the crosshair around the spectrum until you have created the color you want.

   The *Color* | *Solid* box displays the color you created with the vertical sliding bar and crosshair.

   All the fields in the dialog box reflect the correct number for the color in the crosshair. You can also type values in the fields to choose a color.

7. Click *Add to Custom Colors*. The color box in the *Custom Colors* section dialog box shows the new color.

   Note: *Define Custom Colors* >> is a Microsoft Windows feature that is unavailable.

Customized color you created

8. Click *OK* to save the changes and close the dialog box.
9. Click *Apply* in the Color dialog box to update the design with your color changes.

Saving a Customized Color Palette

After you customize a color palette, you can save these settings for use with other designs and for future use with the current design.

1. Choose *Display – Color/Visibility*.

   The Color dialog box appears.

2. Click *Apply* after making your color changes.
3. Choose *File – Save Color Palette.*

   A file browser appears with the filter set to `*.col` in the current local working directory. You can manually browse to other directories to save a color file.

4. Name the customized color palette and click *Save*.

The current design's customized color palette is saved.

Note: To revert to the default Cadence color palette, choose *File – Load Default Cadence Color Palette.*

Importing a Customized Color Palette

1. Choose *Display – Color/Visibility*.

   The Color dialog box appears.

2. Choose *File – Load Color Palette.*

   A file browser appears with the filter set to `*.col` and a list of all `.col` files available in the current local working directory. You can manually browse to other directories to open a color file.

3. Choose a customized color palette from the list and click *Open*.

   The customized color palette is applied to the current design.

To revert to the default Cadence color palette, choose *File – Load Default Cadence Color Palette.*

Adding Subclasses to the My Favorites folder

1. Run the `color192` command.

   The Color dialog box appears.

2. Choose a folder from the left pane.

3. Hover your cursor over the color box associated with the subclass you want to add to My Favorites.

4. Right-click and choose *Add to My Favorites* from the pop-up menu.

5. Add as many subclasses as necessary.

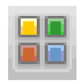
   The subclasses are copied (not moved) to the My Favorites folder.

   Note: A subclass stored in the My Favorites folder can be removed by hovering your cursor over the color box associated with the subclass, right-clicking and choosing *Remove from My Favorites* from the pop-up menu.

Summary

You now know how to turn on the visibility of classes and subclasses, apply colors to subclasses, and customize colors.

You have learned the following:

- **New menu bar command:** *Display – Color/Visibility*
- **New console command:** `color192`
- **New toolbar icon:** 

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `color192` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- *env File*

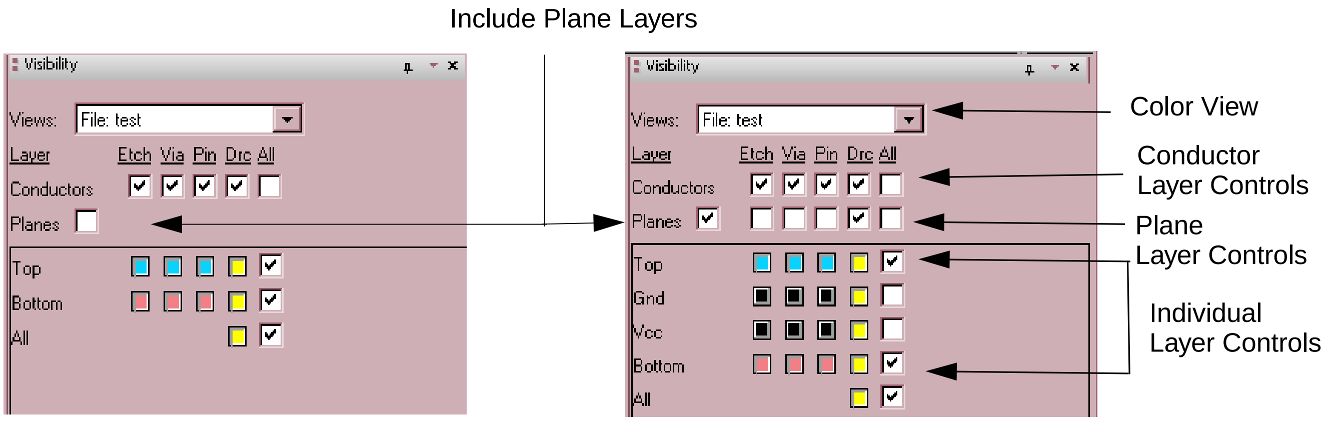
What's Next

Go to Lesson 3-3: Controlling Etch Visibility to learn how to control etch visibility, turn on or off layers or design elements, and separately control the etch routing layers from plane layers as well Etch, Pins, Vias, and DRCs classes.

Lesson 3-3: Controlling Etch Visibility

Overview

The *Visibility* foldable window pane on the Control panel lets you turn on or off layers or design elements. Once you assign colors to each class of design element (see Lesson 3-2: Controlling Color and Visibility on page 86), you can use the *Visibility* foldable window pane to selectively display Etch, Pin, Via, and DRC classes on each layer of the design. The *Visibility* foldable window pane displays the color assigned to a design element when that element is visible and the background color of the Design window when the design element is invisible.



In this lesson, you will learn how to turn on or off layers or design elements and separately control the etch routing layers from plane layers, as well as Etch, Pin, Via, and DRC classes.

Procedure

1. Start Allegro PCB Editor and open `cds_routed_DRC.brd`.
2. Use one of these commands to zoom into your design:
   - From the menu bar, choose *View – Zoom Fit.*
   - At the console window prompt, type `zoom fit`.
   - At the icon toolbar, click .
   - Press F2.
3. In the *Visibility* foldable window pane of the Control Panel, uncheck the boxes in the *All* column in the *Conductors* row.

   This turns off visibility for all etch, pins, vias, and DRCs for the conductor layers.
4. Check the box next to *Planes*, if it is not already checked.

   This displays the visibility check boxes for etch, pins, vias, and DRCs subclasses for the Plane layers.

5. Check the *All* box in the *Planes* row to turn on the visibility for etch, pins, vias, and DRCs for the plane layers.
6. Uncheck the *All* box in the *Planes* row.
7. Check the *Etch* box in the *Conductors* row to control an individual element. Look for changes in the Design window.
8. Check the *Pin* box in the *Conductors* row to control an individual element. Look for changes in the Design window.
9. Check the *Via* box in the *Conductors* row to control an individual element. Look for changes in the Design window.
10. Check the *DRC* box in the *Conductors* row to control an individual element. Then zoom into the area around the D1 diode (x, y coordinates of 2700, 1910). View the DRCs.
11. Uncheck the box next to *Planes*.

    Notice that the plane layers are removed from the *Visibility* foldable window pane. This may make it easier to view, depending on the layers in your design.

Summary

You now know how to turn on or off layers or design elementselements and to separately control the etch routing layers from plane layers, as well as *Etch, Pins, Vias*, and *DRCS* classes.

For More Information

See the *Getting Started with Physical Design* user guide in your documentation set.

What's Next

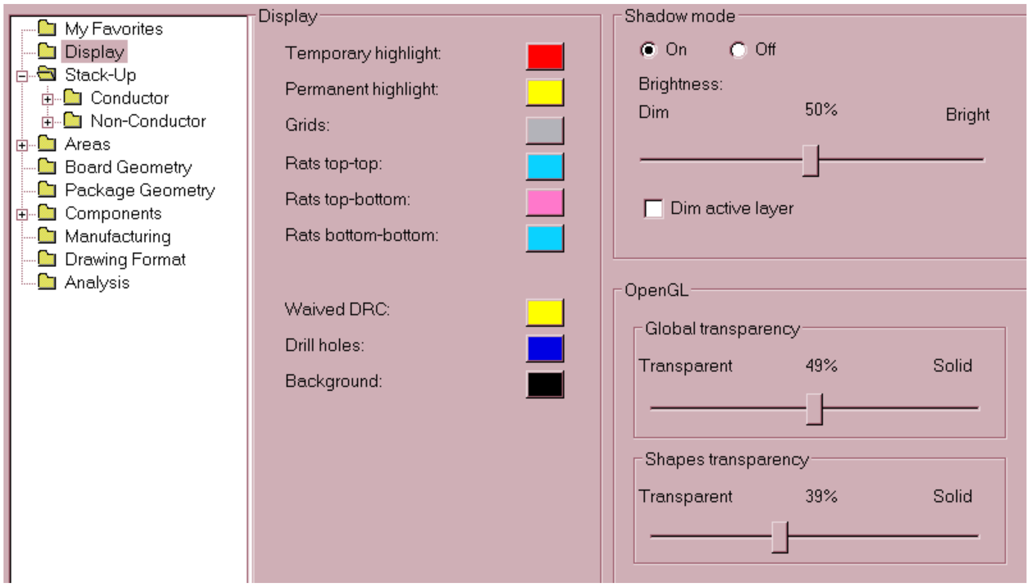
Go to <u>Lesson 3-4: Controlling Colors and Dimming Graphics</u> to learn how to use levels of visibility based on the importance of the element.

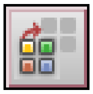
Lesson 3-4: Controlling Colors and Dimming Graphics

Overview

The Graphics Dimming or Shadow Mode option provides distinct levels of visibility based on the element importance. You use Shadow Mode with the `hilight` and `dehilight` commands, as well as various interactive commands.

Transparent graphics powered by the OpenGL engine provide the opportunity to work with a greater number of layers visible, including planes that often block the graphics on other layers. Separate graphics transparency slide bars can be used to set varying degrees of transparency for the entire design or shapes only.

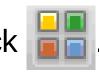




Toggles Shadow Mode on or off

In this lesson, you will learn how to use levels of visibility in your design based on the importance of the element.

Procedures

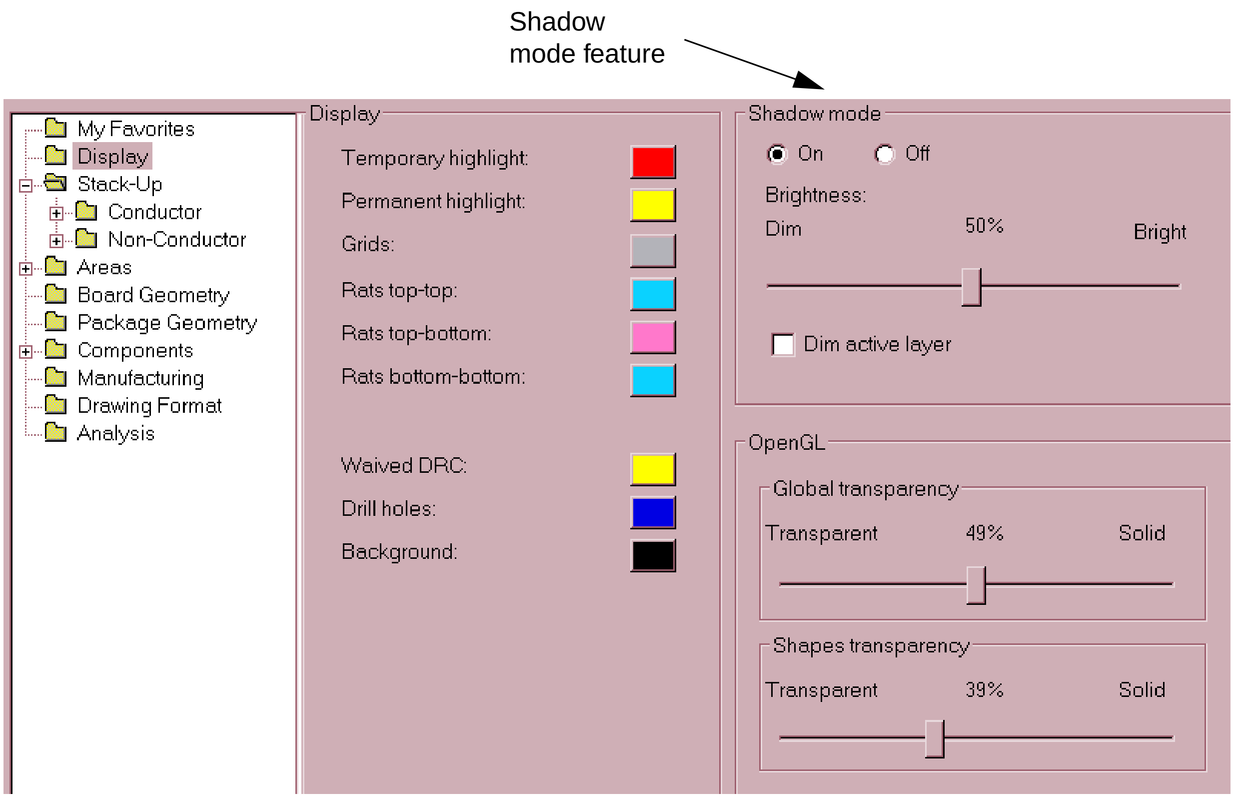
1. With `cds_routed.brd` displayed in the Design window, use one of these commands to display the Color dialog box:
   - ❑ From the menu bar, choose *Display – Color/Visibility.*
   - ❑ At the console window prompt, type `color192`.
   - ❑ At the icon toolbar, click .
2. Click *Display*.
3. To change the background color, choose a color from the Color section of the dialog box by clicking on it.
4. Click the color box next to *Background*. The color box changes to the color you chose in the *Color* section.
5. Click *Apply* for the change to take effect and leave the dialog box open.
6. Click on *Shadow mode* (shadow mode is normally disabled).
   - ❑ The *Brightness* setting slide bar moves to its last applied percentage of brightness. The initial default percentage setting is 50%.
   - ❑ The *Dim active layer* check box lets you dim the active layer of a design. Dimming the active layer if it contains a large number of elements displayed normally (non-highlighted) increases the effectiveness of shadow mode. You can dim the active layer using the check box in the Color dialog box or the *Options* foldable window pane when shadow mode is turned on and you have clicked *Apply*.
7. Drag the *Brightness* slide bar in either direction.

   The colors in the design dim to the chosen percentage of brightness in the slide bar when you click *Apply*.
8. Click *OK* to apply the changes and close the Color dialog box.
9. Notice how the color of the current *Active Class and Subclass* as defined in the *Options* foldable window pane displays at the normal color, while all others are drawn at the dimmed color.
10. Change the A*ctive Class* in the *Options* foldable window pane to *Board Geometry* and the *Active Subclass* to *Outline*.

Notice now that the board outline is drawn at the normal color and everything else displays at the dimmed color. Be sure that *Dim active layer* is unchecked in the *Options* foldable window pane when you are using shadow mode.

11. To turn off shadow mode, use one of the following:

- ❑ At the console window prompt, type `shadow toggle`.
- ❑ Click .



Setting Transparency Globally

1. Choose *Display – Color/Visibility* to display the Color dialog box.

2. Choose the *Display* folder from the left pane.

3. In the *OpenGL* section, use the *Global transparency* slider bar to vary the level of intensity for the entire drawing.

a. Slide the bar completely to the right for a pre-16.0 graphics display.

b. Slide the bar completely to the left to cause previously filled geometry, such as clines and pads, to display with less intensity.

The change takes effect when you click *Apply*.

Setting Transparency for Shapes

1. Choose *Display – Color/Visibility* to display the Color dialog box.
2. Choose the *Display* folder from the left pane.
3. In the *OpenGL* section, use the *Shapes transparency* slider bar to vary the level of intensity.

   a. Slide the bar completely to the right for a pre-16.0 graphics display.

   b. Slide the bar completely to the left to to cause previously filled geometry to display with less intensity.

The change takes effect when you click *Apply*.

Summary

You now know how to use levels of visibility based on the importance of the element. With Shadow Mode turned on, you can control the color intensity of the non-important elements. The higher the brightness percentage, the less difference in color between the important and the non-important elements.

Visibility can also be controlled with the Global Transparency slide bar or the Shapes Transparency slide bar to display elements semi-transparently and view any elements that lie beneath other elements, for either the entire design or shapes only, respectively.

You have learned the following:

- **New terms:** shadow mode, OpenGL
- **New console commands:** `shadow toggle`

- **New toolbar icon:** 

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `shadow toggle` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 3-5: Using the Control Panel to Manipulate Design Objects to learn how to find element types, elements by name, and elements by property.

Lesson 3-5: Using the Control Panel to Manipulate Design Objects

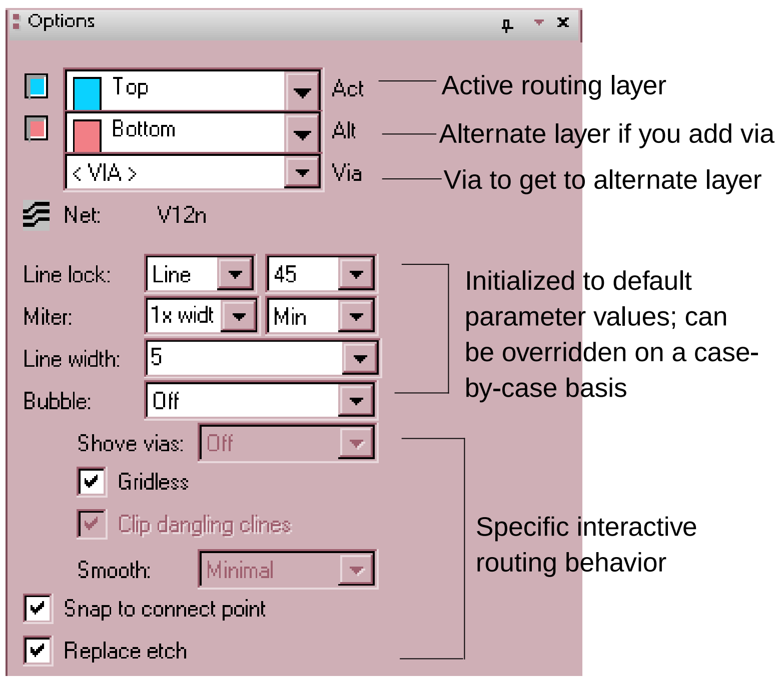
Overview

In Allegro PCB Editor, the area to the right of the Design window is the Control Panel. You can toggle among the *Options*, *Find*, and *Visibility* foldable window panes.

Based on the command you are running, the parameters in the *Options* foldable window pane change. When you choose a command, the *Options* window pane changes to reflect the appropriate class and the default subclass (the first subclass on the list for that class). The parameters and values you set in the *Options* pane take effect immediately. They override definitions for the same parameters and values that you may have already defined using the menu bar or console commands.

For example, Allegro PCB Editor looks to the Design Parameter Editor for the rotation and text values. If a different value exists in the *Options* pane, however, Allegro PCB Editor ignores the information in the Design Parameter Editor dialog box, available by choosing *Setup – Design Parameters* (`prmed` command).

When you update values in the Design Parameter Editor, the values in the *Options* pane change as well. See Lesson 4-4: Choosing Drawing Options on page 134 for additional information.



By default, the Control Panel is docked to the right of the Design window. You can change the position of the Control Panel. For additional information, see Lesson 5-1: Customizing Your View and Toolset on page 155.

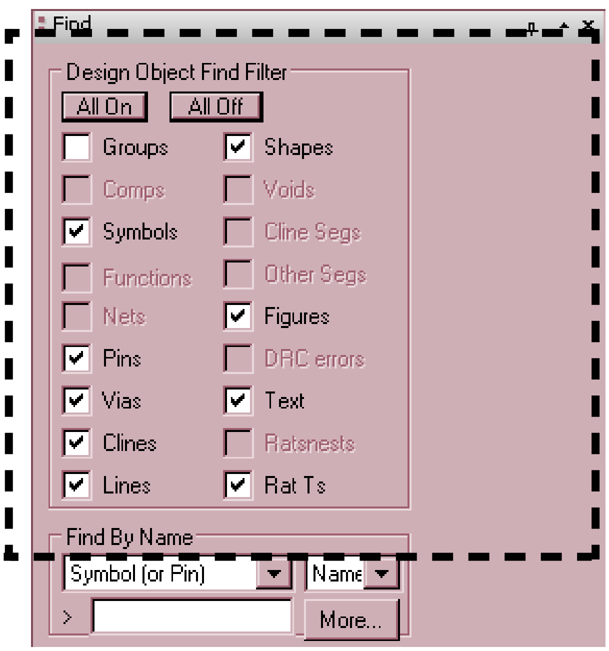
In the next three procedures, you will learn how to perform these tasks:

- Finding Objects by Type
- Finding Objects by Name
- Finding Objects by Property

Procedures

Finding Objects by Type

1. With `cds_routed.brd` displayed in the Design window, click the *Find* foldable window pane in the Control Panel to bring the Find window to the front of the display.
2. Use one of these commands to run the `move` command:
   - ❑ From the menu bar, choose *Edit – Move*.
   - ❑ At the console window prompt, type `move`.
   - ❑ Click .
3. In the *Find* window pane, check boxes of all elements eligible for use with the `move` command are toggled on. All but one of the available elements has been chosen in the *Find* pane. Other elements are not available (grayed out) for this command.



4. Zoom into the left side of the `cds_routed.brd` file and click on the reference designator *U3* (x, y coordinates 25, 1775).

Part U3 snaps to your cursor. In the *Design Objects Find Filter* section of the *Find* window pane, *Symbols* is checked, or toggled on. Allegro PCB Editor treats the reference designator you selected as *part of* the package symbol. Because *Symbols* is higher in the selection hierarchy than the reference designator *Text*, Allegro PCB Editor selects the item at the higher level.

When you check more than one element in the *Design Object Find Filter* section, Allegro PCB Editor prioritizes selection by going from top to bottom in the left column of elements and then top to bottom in the right column of elements to find the chosenchosen design elements that are of the highest priority element type.

5. With the cursor in the Design window, click the right mouse button.

A pop-up menu appears with options for the active `move` command.

Note: When you have multiple selections in the Find window and your cursor is clicked in a location that has multiple elements, such as a symbol and a cline, you can click the right mouse button and choose *Reject* from the pop-up menu. The Reject Item Selection dialog box appears. You can choose one of the other elements listed in the dialog box to be acted upon by the current command even though the element is lower in the hierarchy shown on the Find window.

6. Choose *Oops* from the pop-up menu.

Part U3 snaps back to its original location. The `move` command remains activated.

7. In the *Find* pane, click *All Off,* then click only the box next to *Text*.

All items in the *Find* foldable window pane are unchecked except for *Text*.

8. Click the reference designator text for *U3* again.

This time, part *U3* does not snap to the cursor. Instead, only the reference designator text snaps to the cursor.

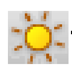
Because of the change you made in the *Design Objects Find Filter* section, the reference designator you selected is treated as a text element, and the symbol is not chosen.

9. Click the right mouse button and choose *Cancel* from the pop-up menu.

Text *U3* snaps back to its original location and the `move` command is deactivated.

Finding Objects by Name

1. With `cds_routed.brd` displayed in the Design window, use one of these commands to run the `hilight` command:

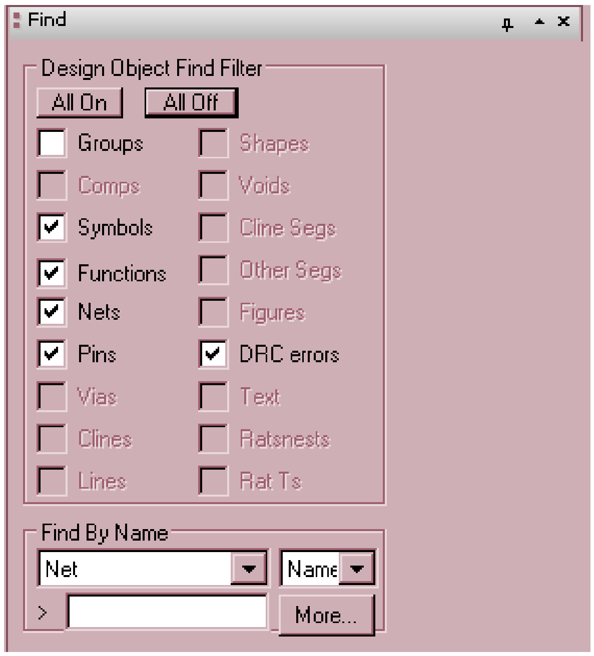
- ❑ From the menu bar, choose *Display – Highlight.*
- ❑ At the console window prompt, type `hilight`.
- ❑ Click .

2. Click on the *Options* foldable window pane in the Control Panel to check your highlighting color (*Perm highlight)*.

3. Click the *Find* foldable window pane. In the *Find by Name* box, verify that *Net* is chosen from the drop-down menu.

4. In the *Find by Name* box, verify that *Name* is chosen from the drop-down menu.

You can choose either *Name* or *List*. If you choose *List*, the data typed in the foldable window pane is not a design element, but the name of a text file that contains a list of the names for the design element. Each name in the file must be on a separate line.



5. In the blank text box below *Net*, type Q1 and press `Enter`.

   You can use wildcard characters such as * or ? as part of the name or by itself. The selections in the *Design Object Find Filter* section are ignored.

6. The net named *Q1* highlights, and Allegro PCB Editor zooms the design to the location of the Q1 net.

   Note: If you enabled the `no_zoom_to_object` environment variable in the *Input* category in the User Preferences Editor dialog box, Allegro PCB Editor does not zoom into the design.

7. If it is difficult to see the highlighted net, follow these steps and then view the highlighted Q1 net:

   a. Choose *Setup – Design Parameters*.

   b. Click the *Display* tab to bring it forward.

   c. In the *Enhanced Display Modes* section, click *Filled Pads*.

   d. Click *OK*.

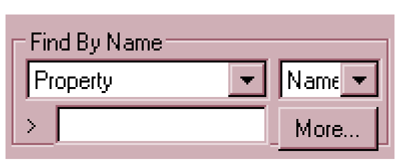
8. Use one of these commands to run the `dehilight` command:
   - ❑ From the menu bar, choose *Display – Dehilight.*
   - ❑ At the console window prompt, type `dehilight`.
   - ❑ Click .
9. Click the Q1 net.

   All highlighting is removed.
10. With the cursor in the Design window, click the right mouse button and choose *Done* from the pop-up menu.

Finding Objects by Property

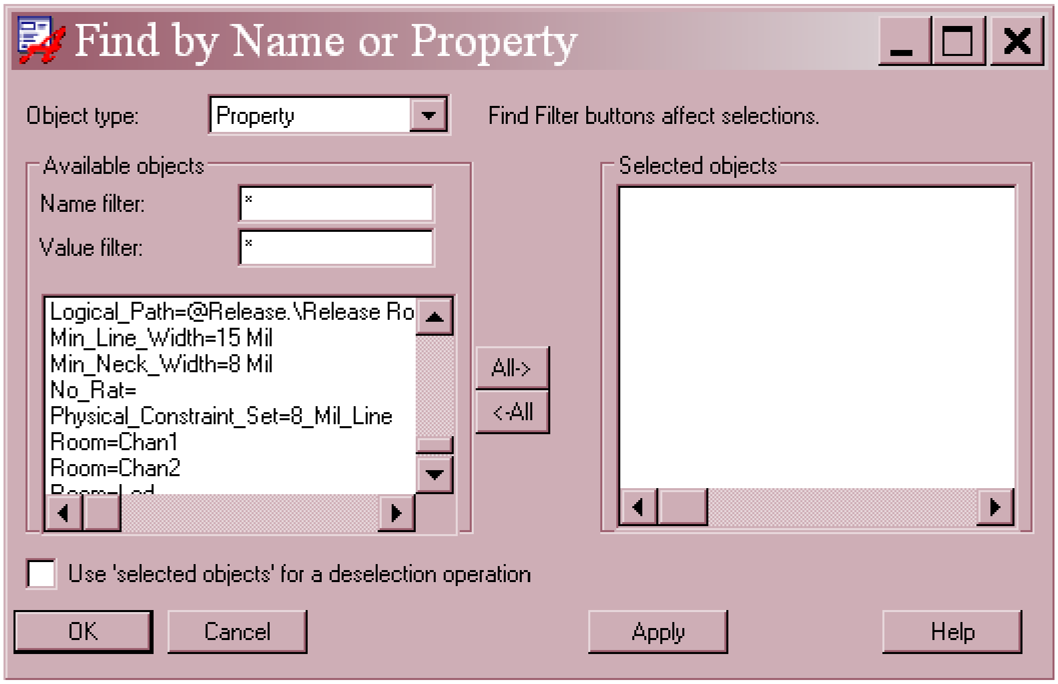
You can find elements by specifying the properties attached to them. A property is a name or a value pair assigned to a particular element. For Allegro PCB Editor, the property name is an identifier, a string of not more than 32 characters that includes letters, digits, and underscores
(_) and starts with a letter. Some examples of property names are: SIZE, ROUTE_PRIORITY, and PART_NAME.

1. With `cds_routed.brd` displayed in the Design window, use one of these commands to run the `hilight` command:
   - ❑ From the menu bar, choose *Display – Highlight.*
   - ❑ At the console window prompt, type `hilight.`
   - ❑ Click .
2. Click the *Find* foldable window pane in the Control Panel.
3. Under the *Find by Name* text box, choose *Property* from the drop-down list if it is not already chosen.



The *Property* option under the *Find by Name* box uses the active selections in the *Design Object Find Filter* section, Only relevant check boxes are toggled on, limited to *Symbols*, *Functions*, *Nets*, *Pins*, and *DRC errors*.

4. Click *More* to display the Find By Name or Property dialog box listing properties that exist in your design.



To obtain a complete listing of available properties, make sure that all the buttons in the *Design Object Find Filter* section are toggled on.

5. Scroll down and choose the *MIN_LINE_WIDTH=15* property and click *Apply*.

   You just highlighted your special voltage nets. All nets with an assigned MIN_LINE_WIDTH property of 15 are highlighted in the Design window. The V12N, GND_EARTH, AGND, and V+12 nets in this design have a MIN_LINE_WIDTH property attached to them.

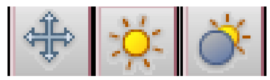
6. Click *Cancel* to close the Find by Name or Property dialog box.

7. With the cursor in the Design window, click the right mouse button and choose *Cancel* from the pop-up menu.

   The `hilight` command is no longer active.

Summary

You now know how to how to select element types in your design based on the selections active in the Find Filter, and find an element by name or property.

- **New menu commands**: *Edit – Move, Display – Highlight, Display – Dehilight, Setup – Design Parameters*
- **New console commands:** `move, hilight, prmed, dehilight`
- **New toolbar icons:** 
- **New dialog box:** Find by Name or Property
- **New environment variable:** `no_zoom_to_object`

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `move` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `hilight` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `prmed` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `dehilight` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 3-6: Highlighting Objects to learn how to highlight elements.

Lesson 3-6: Highlighting Objects

Overview

You can highlight and display database elements in certain colors when the location is unknown in your design, or so that you can see where the elements are placed or how they are routed. Highlighting is particularly useful on very large, densely populated designs.

The type of database element highlighted is based on the selections active in the Find window pane. You can choose your highlight color from up to 24 different colors in the *Options* window pane. Once highlighted, the elements remain highlighted until you dehighlight them.

Procedure

1. With `cds_routed.brd` displayed in the Design window, zoom into the area around the *U3* part (x, y coordinates 25, 1775), located at the left side of the design near the center.

   Note: If you enabled the `no_zoom_to_object` environment variable in the User Preferences Editor dialog box, Allegro PCB Editor does not zoom into the design. If you set the `display_nohilitefont` environment variable, Allegro PCB Editor controls how elements are highlighted. The default is to display the highlighted elements with a combination of the highlight color and the color assigned to the element. For additional information on setting environment variables, see Lesson 5-3: Setting Environment Variables on page 162.

2. Use one of these commands to run the `hilight` command:
   - From the menu bar, choose *Display – Highlight.*
   - At the console window prompt, type `hilight.`
   - Click .

3. Click the *Options* pane in the Control Panel.

   The *Options* pane changes to display the available colors and the current permanent highlight color.

4. Click on the red color button to designate red as the active color for permanent highlighting.

5. Ensure the *Find* pane is visible in the Control Panel.

6. Change the setting in the *Find by Name* list to *Symbol (or Pin)* as shown and type `U3` in the fill in (**>**) text box.

7. Press the `Enter` or `Return` key.

U3 becomes highlighted. You can also see the highlighted part in the World View window.

8. Use one of these commands to run the `dehilight` command:

   - ❑ From the menu bar, choose *Display – Dehighlight*.
   - ❑ At the console window prompt, type `dehilight`.
   - ❑ Click .

9. Enter ***** in the fill in (>) text box under *Find by Name* in the Find Filter.

10. Press the `Enter` or `Return` key.

    You just removed all the permanent highlights from your design.

11. Click the right mouse button in the Design window and choose *Done* from the pop-up menu.

You can use highlights for elements, critical nets, pins, properties, or any of the items available in the Find pane.

Summary

You now know how to highlight and dehighlight elements in a design.

You have learned the following:

- **New environment variable:** `display_nohilitefont`

For More Information

See the *Getting Started with Physical Design* user guide in your documentation set.

What's Next

Go to Lesson 3-7: Listing Detailed Information About a Specified Object to learn how to obtain detailed information about a specified element.

Lesson 3-7: Listing Detailed Information About a Specified Object

Overview

In Allegro PCB Editor, you can obtain information about an item in the design. Using the Find Filter, you can determine the type of information that displays. Based on the Find settings, you can choose a net name, a component's reference designator, the padstack a pin uses, and so on. Allegro PCB Editor selects the highest-level element that is associated with that selection. If you disable the higher-level elements, Allegro PCB Editor selects lower-level elements. For example, a pin can be part of a function, net, symbol, component, or group. When determining the proper element to highlight, Allegro PCB Editor uses this hierarchy:

- Groups

- Components
- Symbols
- Functions
- Nets
- Pins

Procedure

1. With `cds_routed.brd` displayed in the Design window, zoom into a view area around the *U2* component (x, y coordinates 755, 1980).

   U2 is a long DIP component located just left of the board center and to the right of the three SOICs at the left side of the design.

2. Use one of these commands to run the `show element` command:

   - From the menu bar, choose *Display – Element.*
   - At the console window prompt, type `show element`.
   - Click .

   The check boxes for all elements are toggled *on*. Only the *Groups* box remains unchecked because there are no groups in the design.

3. Move the mouse to place the cursor on one of the pins of the *U2* component that contains etch connected to the pin, and click the left mouse button to select it.

   The Show Element report appears. If your Show Element report window is covering the Find pane, move it so you can also see the Find pane.

   At the top of the Show Element report is a description of the type of element that is chosen, *<COMPONENT INSTANCE>*. The data in this report corresponds to a description of the component instance of the *Comps* item in the Find window because the *Comps* category is higher in the selection hierarchy than pins or etch.

4. In the Find Filter, disable the check box next to *Comps*.

5. Select the same pin on the same component again.

   This time, the Show Element report refreshes to display <*SYMBOL*> information for this component package.

   This report focuses on the characteristics of the physical package symbol, and corresponds to the *Symbols* entry in the Find Filter. *Symbols* is now the priority item in the Find window. If more than one item in the Find window is turned *on*, then the priority goes to the highest active item in the list.

6. In the Find window, disable *Symbols* and select the same pin again.

   The Show Element report refreshes to display <*FUNCTION INSTANCE*> information for this package. This information corresponds to the Functions entry in the Find window. The pin you selected is seen as part of a function or gate within this package.

7. In the Find window, disable *Functions* and select the same pin again.

   The Show Element report refreshes to display <*NET*> information for this pin. This information corresponds to the *Nets* entry in the Find window. Notice the information about etch length and any attached properties.

8. In the Find window, disable *Nets* and select the same pin again.

   The Show Element report refreshes to display <*CONNECT PIN*> information. This information corresponds to the *Pins* entry in the Find Filter. Notice the padstack information.

9. In the Find window, disable *Pins* and select the same pin again.

   The Show Element report refreshes to display <*CONNECT LINE*> information for the connection to the pin. This information corresponds to the *Clines* (etch) entry in the Find window.

10. Close the Show Element report window.

11. Click the right mouse button in the Design window and choose *Cancel* from the pop-up menu.

    Selecting the same element generates different information, based on the settings in the Find window. It is not just *which* item you select, but also the *selection priority* in the Find window that matters.

When using the *Display – Element* menu command, disable all the elements in the Find window. Then enable only the element(s) that generate the information you want to see.

Summary

You now know how to list the attributes of a specified element.

You have learned the following:

- **New menu bar command:** *Display – Element*
- **New console command:** `show element`
- **New toolbar icon:** 
- **New dialog box:** Show Element

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `show element` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

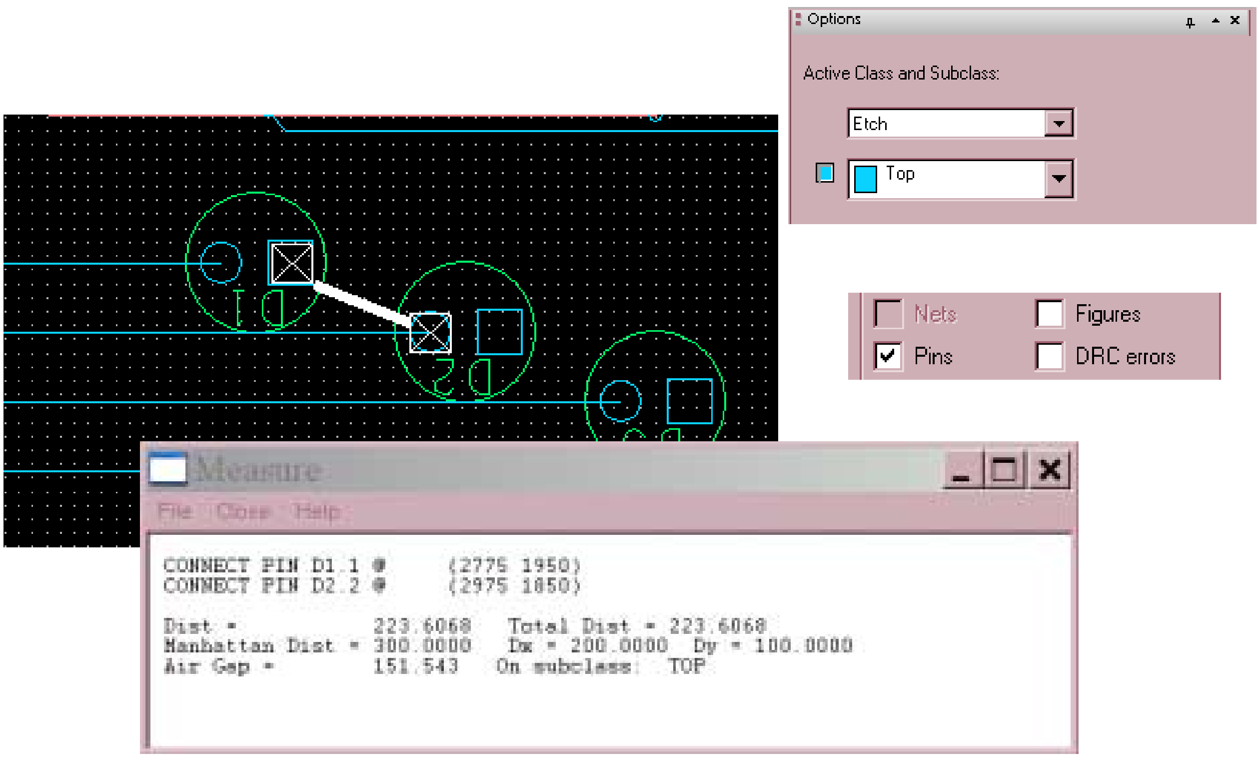
Go to Lesson 3-8: Measuring Distance Between Objects.

Lesson 3-8: Measuring Distance Between Objects

Overview

In Allegro PCB Editor, you can measure distance between two elements. The Find window settings determine which database elements are chosen. If the selection point does not contain any items that match the Find window settings, then Allegro PCB Editor uses the closest grid point to determine the distance. After you select the two points, a window appears detailing information about the distance between the two elements. Information displayed includes total distance, manhattan distance, the delta X and delta Y, and the air gap.

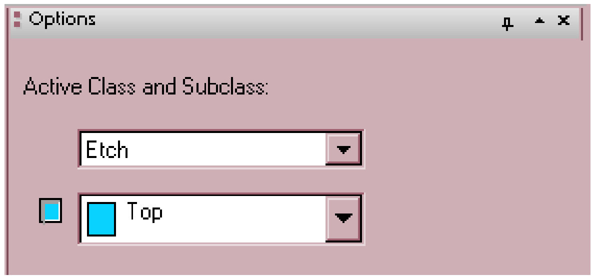
The manhattan distance is the orthogonal distance between two points. The distance is calculated as the sum of the distance between the points along the X axis and the distance between the points along the Y axis: DX + DY. The air gap is the closest, shortest straight line between the two elements. It displays only if the two chosen elements reside on the same class and subclass.



Procedure

1. With `cds_routed.brd` displayed in the Design window, click the *Options* foldable window pane in the Control Panel.

2. Set the Active Class to *Etch* and the Subclass to *Top*, as shown below.



3. Use one of the following to run the `show measure` command:

   ❑ From the menu bar, choose *Display – Measure.*

   ❑ At the console window prompt, type `show measure`.

   ❑ Click .

   The Allegro PCB Editor command console window prompts you as follows:

   `Make two picks for the distance calculator`

4. Choose two elements between which you want to measure the distance. Remember to check the settings in the Find window.

   The Measure report appears, showing information about the elements chosen, the manhattan distance, and air gap information.

5. To exit this mode, click the right mouse button and choose *Done* from the pop-up menu.

6. Choose *File – Exit.*

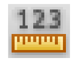
   An Exit window appears, asking if you want to save any of the changes made to your current design.

7. Click *No*.

   Allegro PCB Editor closes. You exit Allegro PCB Editor software.

Summary

You now know how to measure the distance between two design elements.

- **New terms:** manhattan distance, air gap
- **New menu bar command:** *Display – Measure*
- **New console command:** `show measure`
- **New toolbar icon:** 

For More Information

See:

- `show measure` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Module 4: Using Allegro PCB Editor Design Editing Functions to learn some basic design editing functions.

4

Module 4: Using Allegro PCB Editor Design Editing Functions

This module comprises these lessons:

- Lesson 4-1: Naming a Symbol and Setting Drawing Parameters on page 124
- Lesson 4-2: Setting the Grid for a Design on page 128
- Lesson 4-3: Creating a Board Outline on page 130
- Lesson 4-4: Choosing Drawing Options on page 134
- Lesson 4-5: Defining the Stackup on page 139
- Lesson 4-6: Associating Design Objects with Classes and Subclasses on page 144
- Lesson 4-7: Adding Arcs to a Design on page 146
- Lesson 4-8: Adding Circles to a Design on page 148
- Lesson 4-9: Adding Text to a Design on page 149
- Lesson 4-10: Using Zcopy on page 151

Completion Time

It should take approximately 90 minutes to complete the written lessons in this module.

Lesson 4-1: Naming a Symbol and Setting Drawing Parameters

Overview

Lessons 4-1 through 4-3 have procedures to set parameters for a design and create a board outline. Although these procedures do not follow the recommended methodology for creating a board outline, they provide a useful exercise to introduce symbol mode and let you create a mechanical drawing.

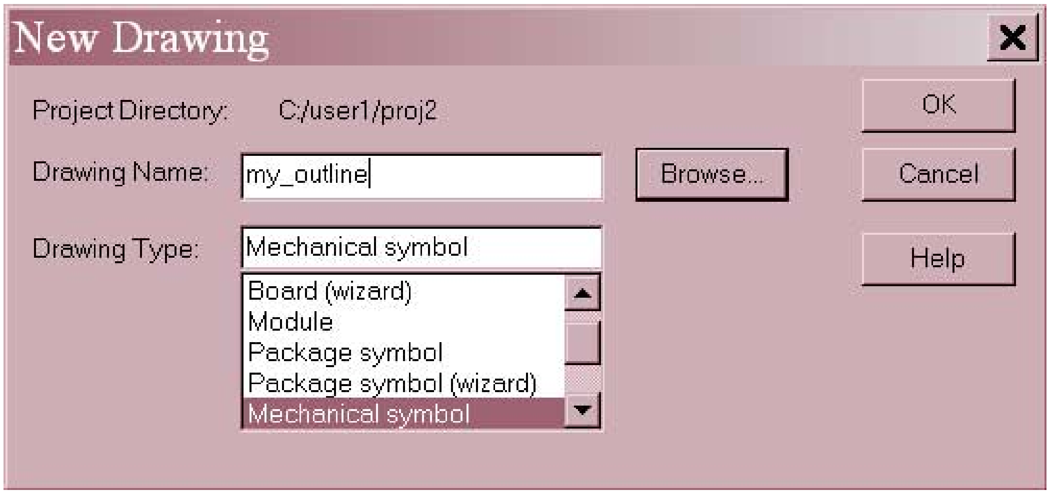
In this lesson, you will name a symbol and set drawing parameters for a design.

Procedure

1. Start Allegro PCB Editor software. If you need additional information, see Lesson 1-2: Starting Up Allegro PCB Editor on page 49.
2. Use one of these commands to start a new drawing:
   - ❑ From the menu bar, choose *File – New*.
   - ❑ At the console window prompt, type `new`.
   - ❑ At the icon toolbar, click .

   The New Drawing dialog box appears.
3. In the Drawing Name text box, type `my_outline`.

4. Choose *Mechanical symbol* from the scrolling list of drawing types, as shown below.



5. Click *OK* to close the New Drawing dialog box.

Allegro PCB Editor displays symbol mode. The title bar changes to (Mechanical) Allegro.

6. Use one of these commands to set up drawing parameters:

a. From the menu bar, choose *Setup – Design Parameters*.

b. At the console window prompt, type `prmed`.

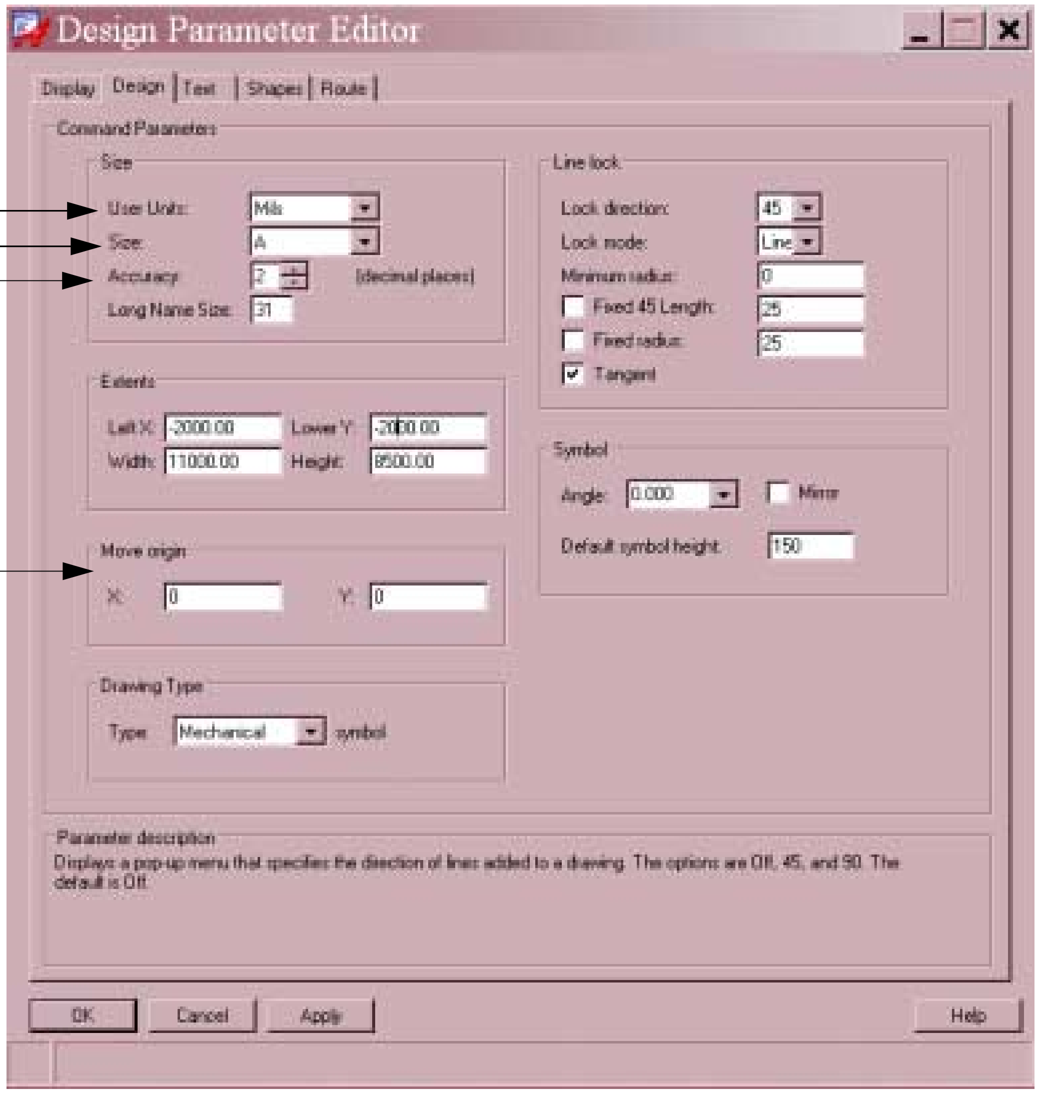
7. Click the Design tab of the Design Parameter Editor dialog box, which controls the size, origin, number of decimal places, and user units of the drawing.

A, B, C, D, or Other. A1, A2, A3, A4 for metric units. Default is 1.

Can be mils (default) inches, millimeters, centimeters, or microns

Number of decimal places (0 to 2) Default is 1.

Use these fields to place the drawing origin inside the drawing area.



8. From the *Size* drop-down list, choose *A*.

9. In the *Accuracy* list box, click the arrow until 2 appears in the box.

The accuracy should match or be less than the accuracy of your board file.

10. Change the Left X and Lower Y text boxes in the *DRAWING EXTENTS* section to match the values in the Drawing Parameters dialog box shown above.

    These settings cause the drawing origin to be placed 2 inches (2000 mils) up and to the right of the lower-left corner of the drawing.

    If you type a value in the *MOVE ORIGIN* section, it causes cumulative results. An easier method for setting the origin point for this instance is to change the coordinates in the *DRAWING EXTENTS* text boxes.

    Note: To advance to the next field in any Allegro PCB Editor dialog box, use the `Tab` key. Do not press the `Enter` key. Pressing the `Enter` key has the same results as clicking *OK*. It closes the dialog box and executes the commands.

11. Click *OK* to save the drawing parameters and close the Design Parameter Editor dialog box.

Summary

You now know how to name a symbol and set drawing parameters.

You have learned the following:

- **New menu bar command:** *File – New*
- **New console command:** `new`
- **New toolbar icon:** 
- **New dialog boxes:** New Drawing, Design Parameter Editor

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.

- new command in the *Allegro PCB and Package Physical Layout Command Reference*.

What's Next

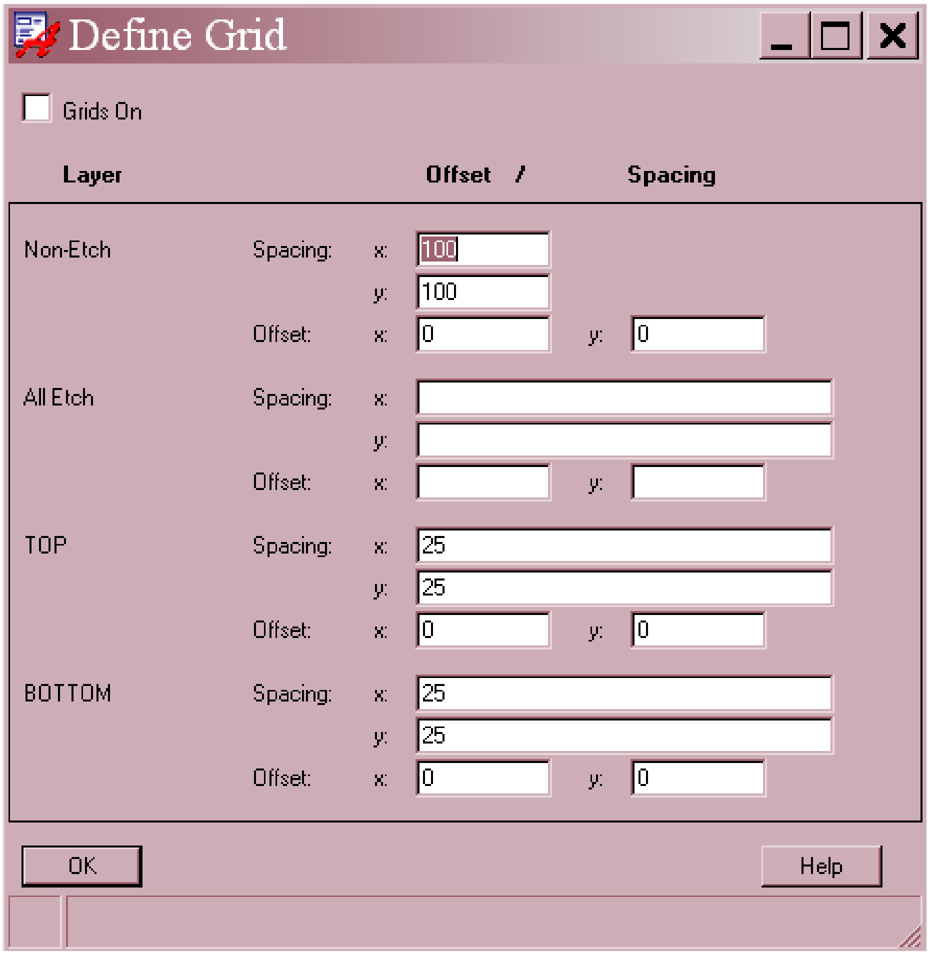
Go to Lesson 4-2: Setting the Grid for a Design to learn how to define a grid.

Lesson 4-2: Setting the Grid for a Design

Procedure

1. Use one of these commands to define the grid for `my_outline.dra`:
   a. From the menu bar, choose *Setup – Design Parameters*.
   b. At the console window prompt, type `prmed.`
2. Click the Display tab of the Design Parameter Editor dialog box.
3. Click Setup Grids in the Grids section.

The Define Grid dialog box appears.



4. In the *Non-Etch* section at the top of the dialog box, make the following spacing changes:
   a. Click in the *Spacing: x* text box, type `25`, and press `Tab`.
   b. Click in the *Spacing: y* text box, type `25`, and press `Tab`.
5. Click *OK* to save the changes and close the dialog box.

Summary

You now know how to define a grid.

You have learned the following:

- **New dialog box:** Define Grid

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.

What's Next

Go to Lesson 4-3: Creating a Board Outline to learn how to create a board outline.

Lesson 4-3: Creating a Board Outline

Overview

In this lesson, you will use the `add line` command to create a board outline.

Note: The datum (0,0) point for this outline is inside the lower-left corner of the board.

Procedure

1. Use one of these `add line` commands to add a board outline to `my_outline.dra`:
   - ❑ From the menu bar, choose *Add – Line*.
   - ❑ At the console window prompt, type `add line`.
   - ❑ At the icon toolbar, click .
2. In the *Options* window pane of the Control Panel, click *Active Class* to *Board Geometry* and *Subclass* to *Outline,* if necessary.

For information about classes and subclasses, see **Lesson 4-6: Associating Design Objects with Classes and Subclasses** on page 144**.**

3. Use the drop-down lists and text box to specify these values in the *Options* tab of the Control Panel:
   - *Line lock* – Line 45
   - *Line width – 8*
   - *Line font* – Solid
4. In Allegro PCB Editor, you can pick points on the screen to specify coordinates, or you can specify the coordinates at the console window prompt. Type each line of values at the console window prompt and press `Enter` after each entry. In this list, first you set absolute values for the x and y coordinates; then you set relative values–just the increment for a specified coordinate. `ix` means to increment the `x` coordinate by the specified value; `iy` means to increment the `y` coordinate by the specified value.

   x -1000 0

   x -150 0

   x -150 -200

   ix 4100

   iy 4500

   ix -4100

   iy -200

   ix -850

   iy -4100

   Or you can use the `pick` and `ipick` commands, or click the P button in the status bar.

   a. At the console window prompt, type `pick.`

   b. In the dialog box, type the value for the x and y coordinates. Be sure to leave a space between the numbers.

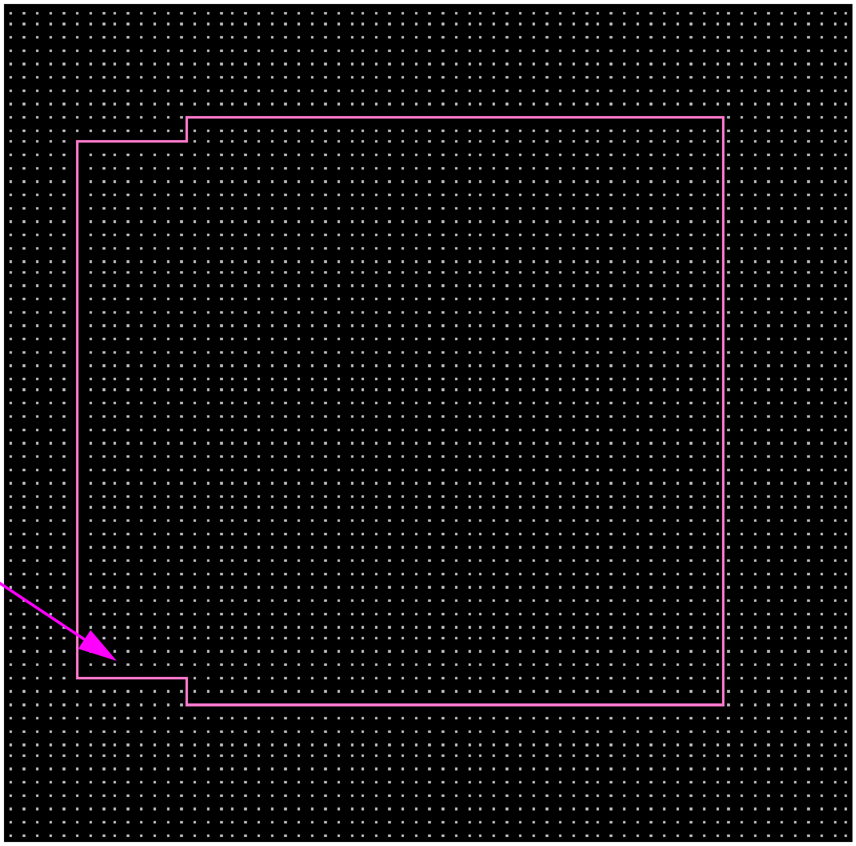
   Note: The dialog box is labeled Pick and the fields differ if you are running a release later than Release 15.0.

   c. Click *Close*.

   d. For the incremental coordinates, type `ipick` at the console window prompt.

   e. In the dialog box, type the incremental value for the x and y coordinate and then click *Close*.

5. Click the right mouse button and choose *Done* from the pop-up menu. Your outline should look like the outline shown below.

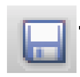


Note: If you are running a release later than Release 15.0, you do not need to perform step 1. The symbol and drawing are both saved when you perform step 2.

1. Use one of these commands to save the symbol (`my_outline.bsm`):

   ❑ From the menu bar, choose *File – Create Symbol.*

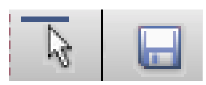
   ❑ At the console window prompt, type `create symbol`.

2. Use one of these commands to save the drawing (`my_outline.dra`):
   - ❑ From the menu bar, choose *File – Save.*
   - ❑ At the console window prompt, type `save`.
   - ❑ Click .

Summary

You now know how to use the add line function to create a board outline.

You have learned the following:

- **New terms:** ix, iy
- **New menu commands:** *Add – Line, File – Save*
- **New console commands:** `add line, save, pick, ipick`
- **New toolbar icons:** 

For More Information

See:

- `create symbol` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `add line` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `save` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `pick` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `ipick` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 4-4: Choosing Drawing Options to learn how to control Allegro PCB Editor display, set the line lock parameters, and run online design rule checks on your design.

Lesson 4-4: Choosing Drawing Options

Overview

In your design, you can:

- Control the Allegro PCB Editor display.
- Set the line lock parameters.
- Run online design rule checks (DRCs).
- Specify the angle that Allegro PCB Editor uses when you place symbols. You can also mirror symbols that you add to a drawing.

In this lesson, you will learn how to set drawing options for your design.

Procedure

1. Open the `cds_routed.brd` file.

   Be sure that the *Files of type* list box in the Open dialog box is set to *All Files*.

2. Using panning and zooming functions, find the *U7* component (x, y coordinates: 15, 3545) located in the upper left part of the board and view the pads.

3. Use one of these commands to display the Design Parameter Editor dialog box:

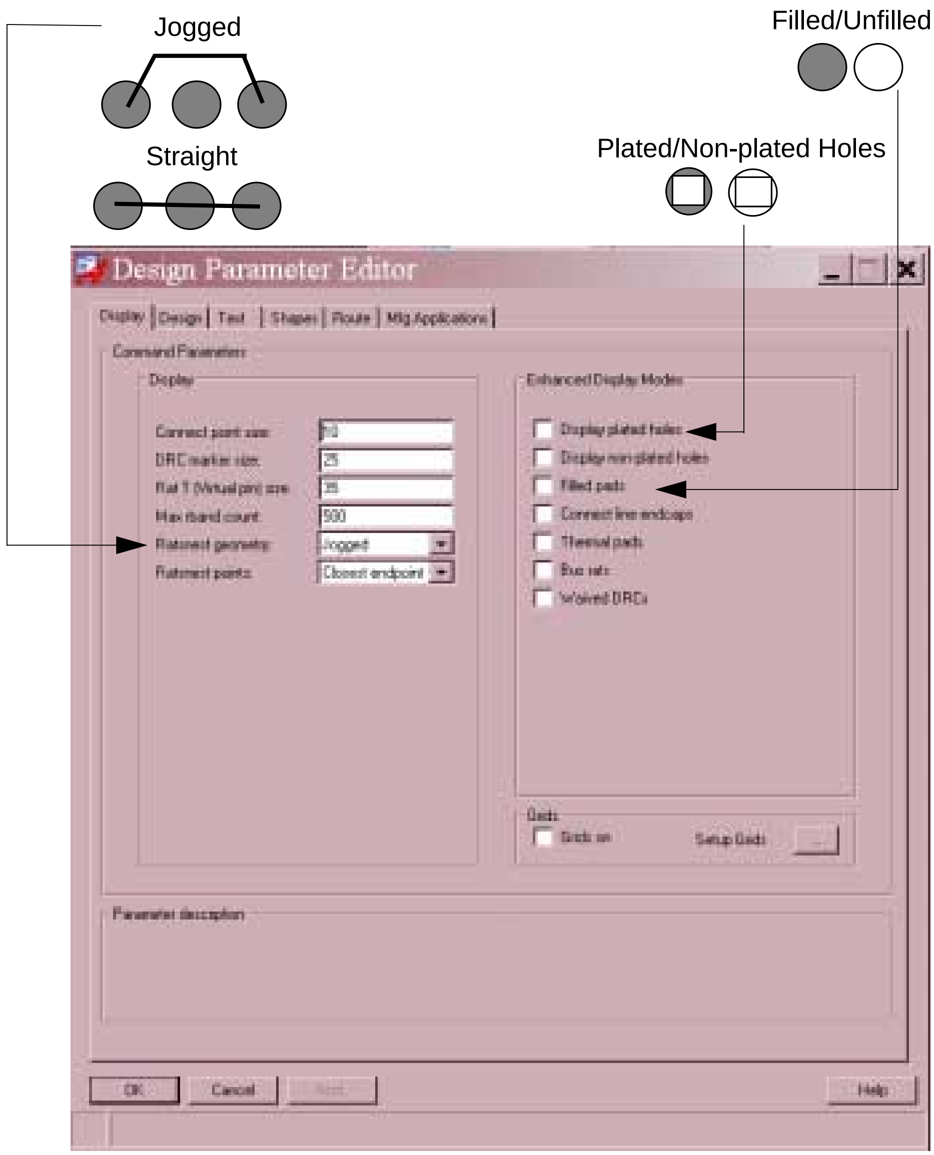
   a. From the menu bar, choose *Setup – Design Parameters*.

   b. At the console window prompt, type `prmed`.

   c. At the icon toolbar, click .

4. Click the Display tab of the Design Parameter Editor dialog box, which shows current settings for various design operations.

5. Click *Filled Pads, Connect line endcaps,* and *Display plated holes*, as shown below, then click *Apply*.



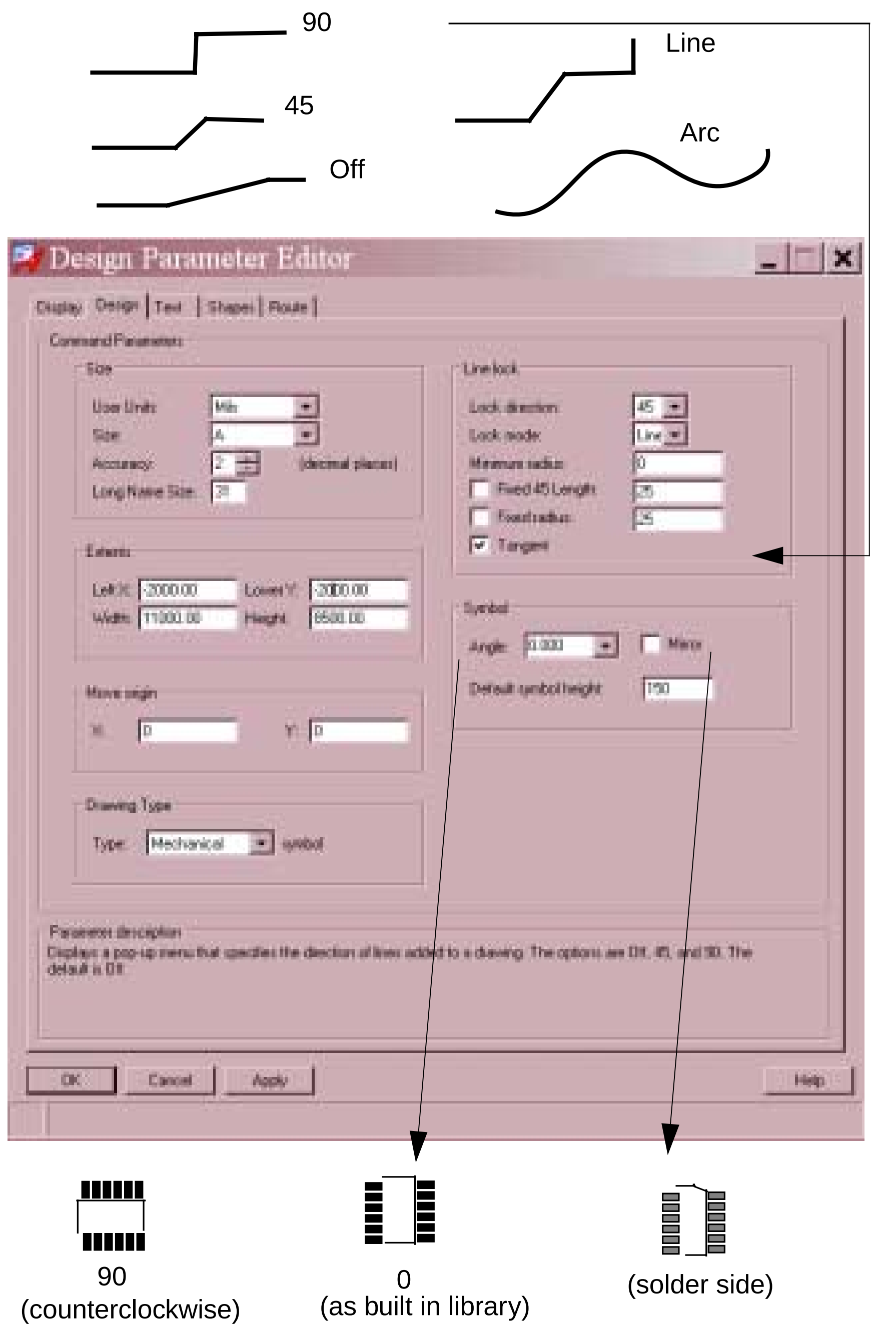
The U7 pin pads now resemble donuts.

6. After viewing the changes on the *U7* component, open the Display tab of the Design Parameter Editor dialog box again and reset the options to their previous disabled (unchecked) states.

You can also control the size of connect points, DRC markers, and Rat Ts as well as the number of rubberbands displayed, and the geometry of the ratsnest lines.

7. Click the *Text* tab and review the fields for controlling text in your design.

8. Click the *Design* tab, where the Line Lock section specifies default values when you add lines to a design. Override these values by modifying fields in the *Options* window pane of the Control Panel.



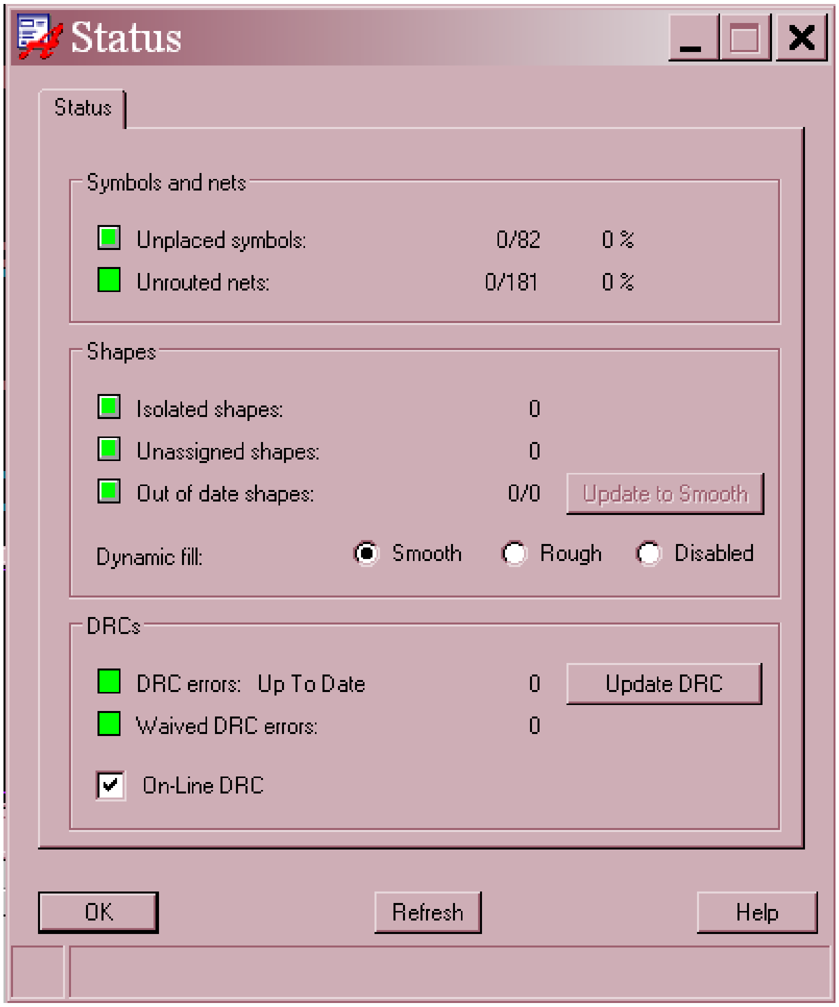
9. In the *Symbol* section, you can specify the default values when you place symbols, or you can override these values if you modify fields in the *Options* window pane of the Control Panel. Click *OK* to close the Design Parameter Editor.

10. To get information about your design, use one of these commands to display the Status dialog box:

a. From the menu bar, choose *Display – Status*.

b. At the console window prompt, type `status`.

You can use the *Status* tab to verify the current state of dynamic shapes and DRCs and update them if they are out-of-date. If the *DRC errors* box is red, choose *Update DRC* to rerun DRC checks to bring status up-to-date and change the *DRC errors* box to green.

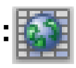


11. Click *Help* for additional information on the dialog box. Click *OK* to save changes and dismiss the dialog box.

Summary

You now know how to set the drawing options for a design.

You have learned the following:

- **New terms:** jogged, DRCs
- **New menu bar command:** *Setup – Design Parameters, Display – Status*
- **New console commands:** `status` and `prmed`
- **New toolbar icon:** 

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `status` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `prmed` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 4-5: Defining the Stackup to learn how to define the stackup or cross section.

Lesson 4-5: Defining the Stackup

Overview

A layer is an insulated plane in the design that contains lines of etch. The ordered list of layers in the design is called the stackup or cross section. In Allegro PCB Editor, you define and sequence the layers, identify the type of material used for that layer, and assign a name to each of the layers in the stackup.

The cross-section worksheet presents the layers of the active design using a spreadsheet where rows represent the primary layer material and columns represent the various properties of the layer. You can resize the dialog box to display a larger range of layers in the design (the default size presents ten layers).

The Layout Cross Section dialog box automatically displays default values from the `materials.dat` (Allegro PCB Editor) or `mcmmat.dat` (APD or AP SI) file. These read-only files contain typical industry fabrication materials. They are located in directories specified in the search path defined by the $MATERIALPATH environment variable.

You can modify most attributes by entering a new value in the appropriate cell. Exceptions to this are the extreme outer layers that have a fixed name called SURFACE (APD) and no definable attributes, and the extreme outer CONDUCTOR layers, which have a fixed name of TOP and BOTTOM. By default, all new design files are created with just two layers: TOP and BOTTOM. You cannot change the name TOP and BOTTOM, but you can change the values on those layers.

In this lesson, you will add more layers to the stackup.

Procedure

1. Use one of these commands to display the New Drawing dialog box:
   - ❑ From the menu bar, choose *File – New.*
   - ❑ At the console window prompt, type `new`.
   - ❑ At the icon toolbar, click .

   Click *No* to indicate that you do not want to save any changes in `cds_routed.brd` when closing it.
2. Type `example1` in the *Drawing Name* text box.
3. Choose *Board* in the *Drawing Type* list and click *OK*.
4. Use one of these commands to display the Layout Cross Section dialog box:
   - ❑ From the menu bar, choose *Setup – Cross-section.*

❑ At the console window prompt, type `define lyrstack`.

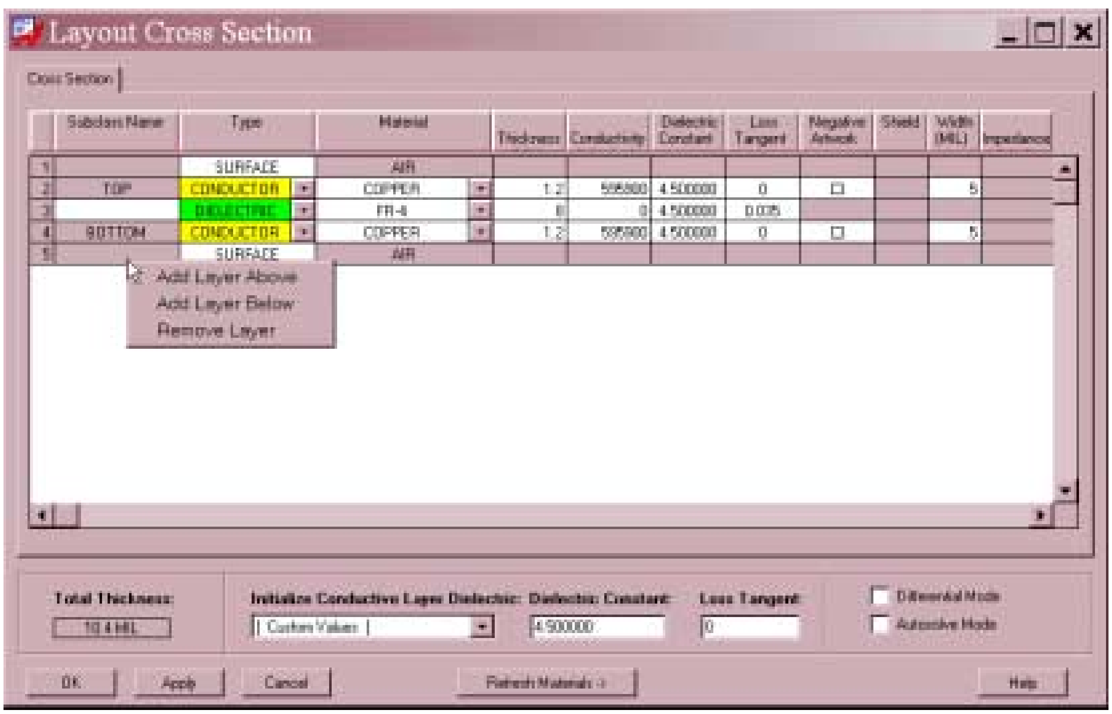
❑ At the icon toolbar, click .

The Layout Cross Section dialog box appears.

Notice that a TOP and BOTTOM layer are already defined by default as conductor layers. This dialog box displays one line for each layer of the layout cross section. The lines are in the physical order of the layers, from TOP/SURFACE to BOTTOM/ BASE as they exist in the layout.

To avoid performance issues when adding layers, you need to first set a sufficient number of planes in the stack (typically every 4th layer).

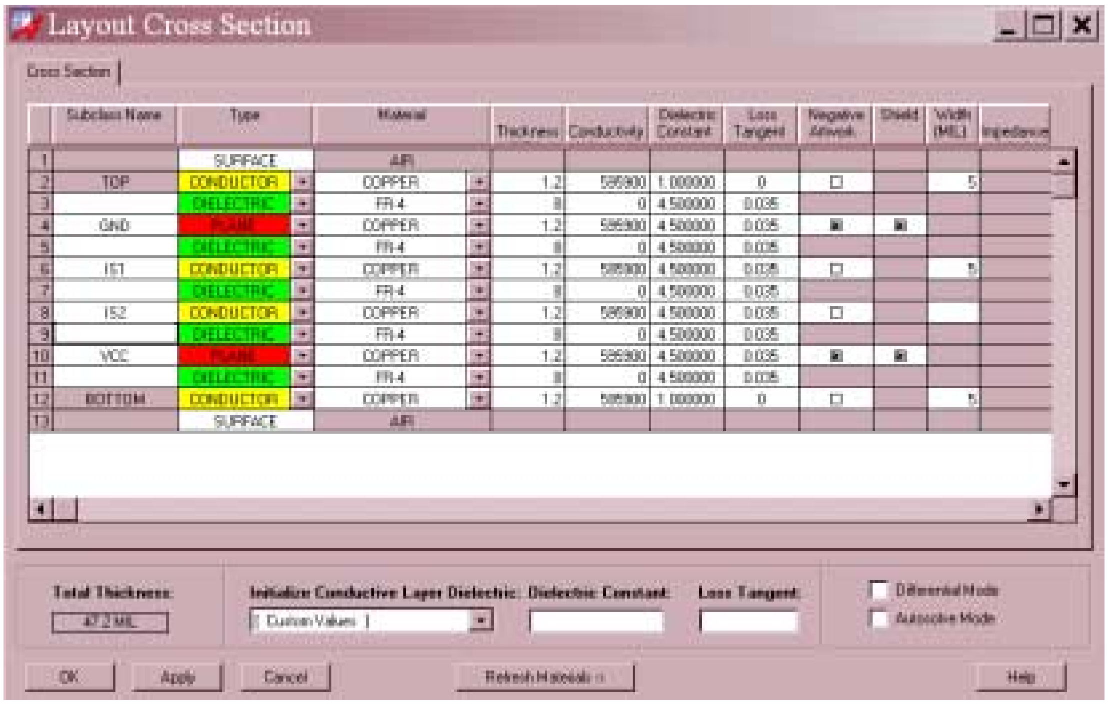
This example shows only the routing layers. You need to add dielectric and core layers to fully model the real design and perform signal integrity analysis on the design.



5. In the *Subclass Name* column, find the row labeled *BOTTOM*. Right-click and choose Add Layer Above from the pop-up menu.

The tool adds a new dielectric layer above the existing layer. You can then change the layer name and type as well as other attributes.

6. Set up your stackup to match the layer specifications shown below by repeating step 5 until there are nine layers between the TOP and BOTTOM layers.



The GND and VCC planes are both labelled negative for the Manufacturing (artwork) output, and DRC checking treats those as negative planes. In this master design file, you added a power and a ground plane and two inner layers for routing. All designs created from this mechanical template start as six-layer boards. However, for this design, only a four-layer board is required.

In the *Thickness* text box at the bottom of the dialog box, note the default value for the thickness of the layer. These values differ for conductor and dielectric layers. This field is important for high-speed designs. When using the impedance calculator, it considers layer types and thickness for its calculations.

7. Right-click and choose Remove Layer from the pop-up menu to delete the dielectric layer above IS1 from the stackup.

8. Repeat step 7 to delete these layers: IS1, the dielectric layer below IS1, and IS2.

   The layers are deleted from the design leaving a four-layer design. You can now save the board template so it can be used again.

   Note: Be sure that you delete all objects on a layer before deleting the layer. Otherwise, an error message appears indicating that you must delete all objects on the layer before deleting the layer.

9. Click *OK* to close the Layout Cross Section dialog box.

10. Use one of these commands to save the changes:
    - ❑ From the menu bar, choose *File – Save.*
    - ❑ At the console window prompt, type `save`.
    - ❑ Click .

Summary

You now know how to define the cross section or stackup.

You have learned the following:

- **New terms:** stackup, cross section,
- **New menu bar command:** *Setup – Cross-section*
- **New console command:** `define lyrstack`
- **New toolbar icon:** 
- **New dialog box:** Layout Cross Section

For More Information

See:

- the *Preparing for Layout* user guide in your documentation set.
- `define lyrstack` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 4-6: Associating Design Objects with Classes and Subclasses to learn how to associate design objects with classes and subclasses.

Lesson 4-6: Associating Design Objects with Classes and Subclasses

Overview

A design file is a composite of a number of design objects. The design objects are categorized by class and subclass. Classes are major categories defining the purpose of the design objects. Subclasses further define the purpose of the design objects. For example, if you are running the `add line` command on class *Board Geometr*y, subclass *Outline,* Allegro PCB Editor adds a simple geometric graphic element, such as board outline, to a design. However, if you are running the `add line` command and *Etch* is the active class, Allegro PCB Editor adds lines of etch to the design.

There are 21 classes to which you can associate design objects. These classes come with some pre-defined subclass names that are most commonly used in board design. You can also define your own subclasses for 11 of the 21 classes.

In this lesson, you will define a subclass and associate design objects with classes and subclasses.

Procedure

1. With `example1.brd` displayed in the Design window, use one of these commands to access the Define Subclasses dialog box:
   - ❑ From the menu bar, choose *Setup – Subclasses*.
   - ❑ At the console window prompt, type `define subclass`.
2. In the Define Subclasses dialog box, click *Board Geometry*.

3. In the Define Non-Etch Subclass dialog box, type `my_subclass` in the *New Subclass* text box and press the `Enter` or `Return` key.

   Note: If you press the arrow next to the name of the new subclass, you can delete it. However, you need to delete all the design objects on the subclass before you can delete the subclass.

4. Click *OK* in the Define Subclass dialog box to save the subclass and dismiss the dialog boxes.

5. Verify in the *Options* window pane of the Control Panel that the active class is *Board Geometry* and the subclass is *My_Subclass*.

6. Use one of these commands to add a rectangle:

   ❑ From the menu bar, choose *Add – Rectangle.*

   ❑ At the console window prompt, type `add rect`.

   ❑ At the icon toolbar, click .

7. In the Design window, start drawing a rectangle and click another point to complete the drawing.

8. Click the right mouse button and choose *Done* from the pop-up menu to exit the command.

9. From the menu bar, choose *File – Save*. Click yes to the confirmer dialog box that asks to overwrite `example1.brd`.

   The rectangle design object is associated with the *Board Geometry* class and *My_Subclass.*

Summary

You now know how to associate a design object with a class and subclass pair.

You have learned the following:

■ **New terms:** class, subclass

■ **New menu commands:** *Setup – Subclasses, Add – Rectangle*

- **New console commands:** `define subclass, add_rect`
- **New toolbar icon:** 
- **New dialog boxes:** Define Subclasses, Define Non-Etch Subclass

For More Information

- the *Getting Started with Physical Design* user guide in your documentation set.
- `define subclass` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `add_rect` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 4-7: Adding Arcs to a Design to learn how to add arcs to a design.

Lesson 4-7: Adding Arcs to a Design

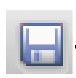
Overview

In Allegro PCB Editor, adding an arc requires three points: a point to start the arc, an end point, and a third point to determine the radius of the arc. To create an arc, specify three points either by mouse click or typing cursor coordinates at the command line.

Procedure

1. With `example1.brd` displayed in the Design window, verify that *Active Class* is *Board Geometry* and *Subclass* is *My_Subclass* in the *Options* window pane of the Control Panel.
2. Use one of these commands to add an arc:

- ❑ From the menu bar, choose *Add – 3pt Arc*.
- ❑ At the console window prompt, type `add arc`.

3. Specify these values in the *Options* window pane of the Control Panel:
   - ❑ *Line Width –* `5`
   - ❑ *Font – Solid*
4. Pick the start point of the arc, the end point, and a third point that dynamically establishes the radius of the arc.
5. Click the right mouse button and choose *Done* from the pop-up menu to make the arc permanent, or pick another three points for the next arc.
6. Use one of these commands to save the file:
   - ❑ From the menu bar, choose *File – Save.*
   - ❑ At the console window prompt, type `save`.
   - ❑ Click .
7. Click *Yes* to overwrite the file.

Summary

You now know how to add arcs to a design.

You have learned the following:

- **New menu bar command:** *Add – 3pt Arc*.
- **New console command:** `add arc`

For More Information

See the *Preparing for Layout* user guide in your documentation set.

What's Next

Go to Lesson 4-8: Adding Circles to a Design to learn how to add circles to a design.

Lesson 4-8: Adding Circles to a Design

Procedure

1. With `example1.brd` displayed in the Design window, verify that the *Active Class* is *Board Geometry* and the *Subclass* is *My_Subclass* in the *Options* window pane of the Control Panel.

2. Use one of these commands to add a circle:

   - From the menu bar, choose *Add – Circle.*
   - At the console window prompt, type `add circle`.

   The following message appears:

   ```
   Pick center point of circle
   ```

3. Specify these values in the *Options* window pane of the Control Panel:

   - *Line Width* – 5
   - *Font – Solid*

4. Move the cursor to the position you want to be the center of the circle, and click the left mouse button.

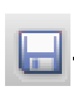
   The following message appears:

   ```
   Pick a perimeter point on the circle.
   ```

5. Repeat steps 4 for each circle you draw.

6. When all circles are complete, click the right mouse button and choose *Done* from the pop-up menu.

7. Use one of these commands to save the file:

   - From the menu bar, choose *File – Save.*
   - At the console window prompt, type `save`.
   - Click .

8. Click *Yes* to overwrite the file.

Summary

You now know how to add circles to a design.

You have learned the following:

- **New menu bar command:** *Add – Circle*
- **New console command:** `add circle`

For More Information

See:

- `add circle` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 4-9: Adding Text to a Design to learn how to add text to your design.

Lesson 4-9: Adding Text to a Design

Overview

You can add text to Allegro PCB Editor designs as simple notes and as logical labels of elements. Labels include reference designators, device type, value, tolerance, and user part number.

You can also specify text size before adding text to your design. For additional information, choose *Setup – Design Parameters* (`prmed` command), the *Text* tab of the Design Parameter Editor, and *Setup Text Sizes*.

Procedure

1. With `example1.brd` displayed in the Design window, verify that the *Active Class* is *Board Geometry* and the *Subclass* is

My_Subclass in the *Options* window pane of the Control Panel*.*

2. Use one of these commands to add text:

- ❑ From the menu bar, choose *Add – Text.*
- ❑ At the console window prompt, type `add text`.
- ❑ Click .

3. Specify these values in the *Options* window pane of the Control Panel:

- ❑ Disable *Mirror.*
- ❑ *Marker Size* – `50`
- ❑ *Rotate* – `0`
- ❑ *Text block* – `16`
- ❑ *Text just* – `Left`

4. Position the cursor and click at the location for the text and enter the text in the Design window.

Limit text lines to 200 characters, including spaces.

Note: To import a text file into the design, run add text, click right, and choose *Read from file*.

5. To correct errors, press the `Delete` or `Backspace` key.

6. Press the `Enter` key to start a new line of text with line spacing set by the parameter block.

7. When you have entered all text required for the current point, click the right mouse button and choose *Done* from the pop-up menu.

8. Use one of these commands to save the file:

- ❑ From the menu bar, choose *File – Save.*
- ❑ At the console window prompt, type `save`.
- ❑ Click .

9. Click *Yes* to overwrite the file.

Summary

You now know how to add text to a design.

You have learned the following:

- **New menu bar command:** *Add – Text*
- **New console commands:** `add text`
- **New toolbar icon:** 

For More Information

See

- the *Getting Started with Physical Design* user guide in your documentation set.
- `add text` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 4-10: Using Zcopy to learn how to copy a shape and add it to a different class and subclass at the same location in the design.

Lesson 4-10: Using Zcopy

Overview

The `zcopy shape` command lets you copy a shape, closed polygon, or rectangle and add it to a different class or subclass at the same location in your design. In this lesson, you will copy the outline of the *J1* component on the left side of `cds_routed.brd` to a new subclass.

Procedure

1. Open `cds_routed.brd`. Zoom to fit the design in the window.

2. Define a new subclass called *new_subclass* on the *Board Geometry* class. See Lesson 4-6: Associating Design Objects with Classes and Subclasses on page 144 for information.

3. To find out the design object used to create the *J1* component, use one of these commands:

   ❑ From the menu bar, choose *Display – Element.*

   ❑ At the console window prompt, type `show element`.

   ❑ Click .

   ❑ Press F4.

4. In the *Find* window pane of the Control Panel, click *All Off*. Then click on *Clines*, *Lines*, and *Shapes*.

5. Zoom in and select the *J1* component on the left side of the drawing.

   The Show Element dialog box shows that the object is *Line*.

6. Close the dialog box, click the right mouse button in the Design window, and choose *Cancel* to cancel the `show element` command.

7. Use one of these commands:

   ❑ From the menu bar, choose *Edit – Z-copy*.

   ❑ At the console window prompt, type `zcopy shape`.

8. In the *Options* window pane of the Control Panel, click the *Active Class* to *Board Geometry* and the *Subclass* to *new_subclass* to indicate where the copy will be located.

9. Select the *J1* component on the left side of the drawing.

   Note: You can use the *Temp Group* command in the pop-up menu to choose more than one element, provided that the items are in the same class and subclass. The command highlights the selected items and identifies the class and subclass of the selected items in the *Class/Subclass* list box in the *Options* tab.

   This message appears in the console window.

   ```
   Copied to: ("Board Geometry/New_Subclass"), 1 copies made
   ```

10. Click the right mouse button and choose *Done* from the pop-up menu.

11. Choose *File – Exit* to exit Allegro PCB Editor. Do not save the file.

Summary

You now know how to copy a shape and change its class and subclass.

You have learned the following:

- **New command:** *Edit – Z-copy*
- **New console command:** `zcopy shape`

For More Information

See:

- `zcopy shape` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Module 5: Customizing the Environment to learn how to customize the Allegro PCB Editor environment.

5

Module 5: Customizing the Environment

This module comprises these lessons:

- Lesson 5-1: Customizing Your View and Toolset on page 155
- Lesson 5-2: Defining Aliases and Function Aliases on page 159
- Lesson 5-3: Setting Environment Variables on page 162
- Lesson 5-4: Running Commands with Strokes on page 164
- Lesson 5-5: Scripting on page 167
- Lesson 5-6: Using Color Visibility Views on page 172

Completion Time

It should take approximately 90 minutes to complete the written lessons in this module.

Lesson 5-1: Customizing Your View and Toolset

Overview

In addition to customizing color selections, setting aliases and function aliases, and scripting, you can customize the Allegro PCB Editor Control Panel and icon toolbars. You can undock the Control Panel in Allegro PCB Editor; resize, stack, or relocate the foldable *Options, Find*, and *Visibility* window panes; and “pin” any window so it remains visible or hide it to maximize your working area. See Lesson 2-1: Identifying Parts of the User Interface about control the visibility of these windows by clicking an arrow to expand a docked window pane, clicking the X to hide it, or by using the *View – Windows* menu choices to hide or display it.

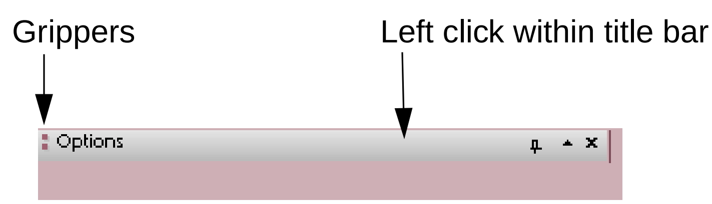
The toolbar contains functionally related icons, such as those for routing or placement, to access common commands.To learn a toolbar icon's function, position the cursor over the icon without depressing the mouse button and view its description in the tool tip that appears.You can also perform the following tasks to customize your toolbar:

- Add or remove groups of icons from the toolbar
- Choose specific icons from each toolbar group to display

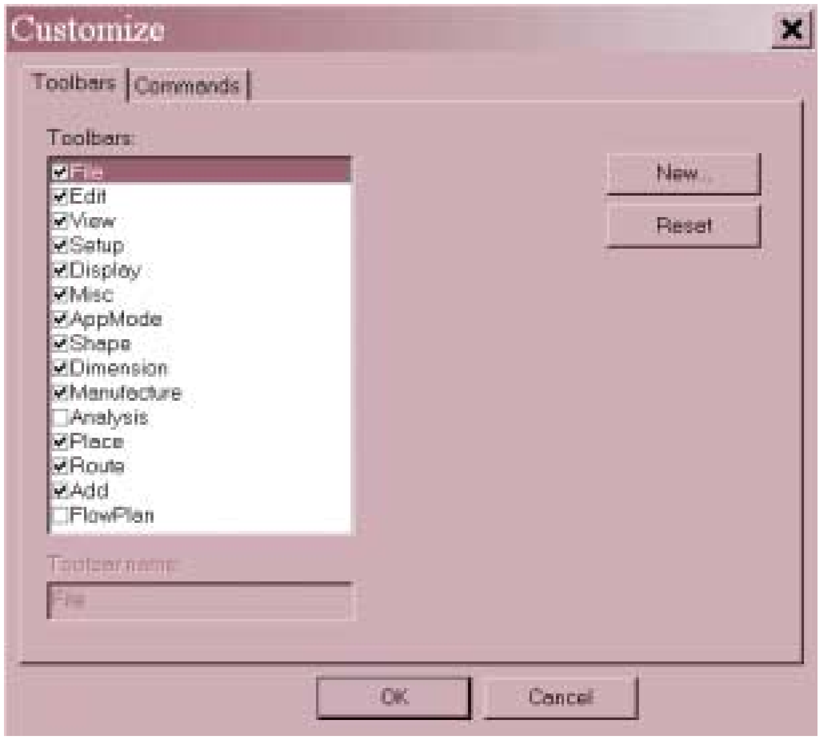
The toolbar settings are stored in an Allegro PCB Editor initialization (`allegro.ini`) file and are read each time you start Allegro PCB Editor. They are not stored in the Allegro PCB Editor database. See Allegro PCB Editor Initialization on page 21.

Procedure

1. Start Allegro PCB Editor. For additional information, see Lesson 1-2: Starting Up Allegro PCB Editor on page 49.
2. Dock or undock any window pane by left-clicking on and moving it anywhere within or outside the design window. You can also left-click on the small circles, or grippers, next to it choose it and move it .

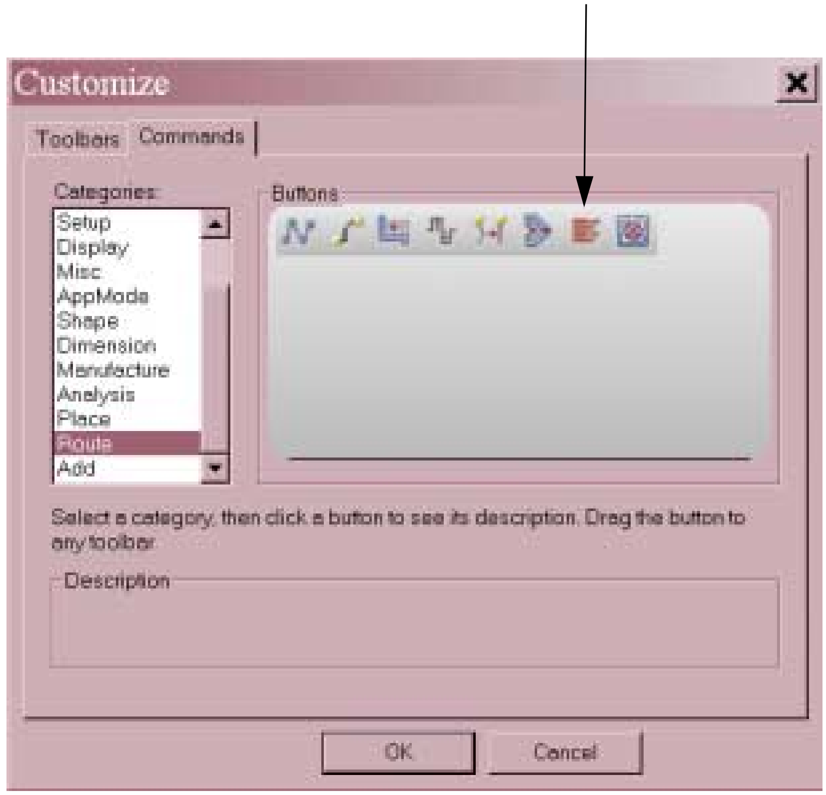


3. From the menu bar, choose *View – Customize Toolbar* to display the Customize dialog box.



4. Experiment by checking or unchecking the boxes to turn on and off the various toolbars on the *Toolbars* tab.
5. Click the *Commands* tab to bring it forward.
6. Click *Route* in the *Categories* list to display available route-related icons.
7. Position the cursor over any icon without depressing the mouse button and view its description in the tool tip that appears.

Create Fanout icon



1. Add the Create Fanout icon to your toolbar. Click and drag the icon from the *Buttons* section in the *Commands* tab to the toolbar area of Allegro PCB Editor.
2. Click *OK* to close the Customize dialog box.
3. Reset the options in the Customize dialog box to their defaults.

Summary

You now know how to position your Control Panel and customize the view of your toolset.

You have learned the following:

- **New menu commands:** *View – Customize – Toolbar*
- **New dialog boxes:** Customize

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.

What's Next

Go to Lesson 5-2: Defining Aliases and Function Aliases to learn how to set aliases and function keys that you can use as shortcut commands.

Lesson 5-2: Defining Aliases and Function Aliases

Overview

The `alias` command lets you create shortcuts for commands you use most often. In addition to using alphanumeric characters as an alias, you can also use function keys (create a function alias), with or without `Shift` and `Control` keys, to execute commands. The alias and function alias are alternative ways of entering the command, but they do not disable the full commands. You can still use the standard form of the command.

Note: The `funckey` command is available only in releases later than Release 15.0.

The `funckey` command lets you create a function alias using alphanumeric keys. The tools support groupings of up to four alphanumeric character keys for operation as a function alias. When keys operate as a function alias, you press the keys and you do not have to press the `Enter` key, provided that your cursor is not active in the Console window.

Aliases and function aliases work only in the Cadence tool, not at the operating system level. When you create an alias or a function alias, it is active only for the current work session. When you exit the tool and return to the operating system, aliases and function aliases are lost.

To use aliases and function aliases repeatedly, define and save them in your local environment file. Allegro PCB Editor creates a `pcbenv` directory with the `env` file at a location determined by the value of the environment variable `HOME`. See env File on page 22.

Procedure

1. At the command console window prompt, type `alias` or `funckey`.

   The Defined Aliases/Funckeys list appears. It lists the default aliases for the typed commands and the function keys. It includes any aliases entered at the site level as well as those that you entered in the local environment file.

   Note: If you are running a release later than 15.0, you can choose *Tools – Utilities – Aliases/Function Keys* from the menu bar to display the Defined Aliases/Funckeys list.

2. Close the list.

3. At the command console window prompt, type this shortcut and press either the `Return` or `Enter` key to set an alias for the `gloss param` command:

   ```
   alias glp gloss param
   ```

4. At the command console window prompt, now type `glp` and press either the `Return` or `Enter` key.

   The `gloss param` command runs as though you typed `gloss param`. The Glossing Controller dialog box appears.

5. Click *Close* to dismiss the dialog box.

   Note: You can do steps 6 through 8 only if you are running a release later than 15.0.

6. At the command console window prompt, type this shortcut to create a function alias for the `add line` command:

   ```
   funckey addl add line
   ```

7. At the command console window prompt, type `addl`. You do not need to press the `Enter` key. Be sure that the cursor is not active in the console window. The `add line` command becomes active as though you typed `add line` at the command console window prompt.

8. Click the right mouse button in the Design window and choose *Cancel* from the pop-up menu.

9. Press `SF6` (`Shift` key and `F6`).

   In the Status bar, note that you have activated the `move` command. `SF6` is a default function alias for the `move` command. Other functions keys have already been set as defaults.

10. Click the right mouse button in the Design window and choose *Cancel* from the pop-up menu to deactivate the `move` command.

11. Experiment with `F1` through `F12` keys and `SF1` through `SF12` keys to see if they are associated with commands.

    Note the command name in the Status bar. Except for `F1`, designated for Help, you can override the default settings for all the other keys.

12. If you are in command mode, click the right mouse button in the Design window, and choose *Cancel* from the pop-up menu.

Summary

You now know how to create and use aliases and function aliases.

You have learned the following:

- **New console commands:** `alias, funckey, gloss param`
- **New list:** Defined Aliases/Funckeys list

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `alias` command in the *Allegro PCB and Package Physical Layout Command Reference.*

- `funckey` command in the *Allegro PCB and Package Physical Layout Command Reference* (available only in releases later than Release15.0).

What's Next

Go to Lesson 5-3: Setting Environment Variables to learn how to set environment variables.

Lesson 5-3: Setting Environment Variables

Overview

You can set or remove values for Allegro PCB Editor preferences, also known as Allegro PCB Editor environmental variables. These variables set operating conditions for Allegro PCB Editor at the local and site levels.

Procedure

1. Based on your version of Allegro PCB Editor, use one of these commands:
   - ❑ At the command console window prompt, type `set`.
   - ❑ If you are running a release later than Release 15.0, choose *Tools – Utilities – Env Variables* from the menu bar.

   The Defined Variables list appears. This list includes all the defined environment variables, including those that you defined in your local `env` file as well as those defined at the command console window prompt with the `set` command.
2. Scroll down until you find `set pcb_cursor`. Note the setting.
3. Close the list.
4. Use one of these commands to display the User Preferences Editor dialog box:
   - ❑ From the menu bar, choose *Setup – User Preferences*.
   - ❑ At the command console window prompt, type `enved`.

5. In the *Categories* section of the User Preferences Editor dialog box, click *UI*.
6. In the *pcb_cursor* list, click the arrow to display the drop-down list and choose *infinite*.

   Notice the description in the *Summary description* section of the dialog box. Also notice the *Effective* field to the right of the *pcb_cursor* setting. Settings becomes effective immediately, after the next command, or after you restart Allegro PCB Editor.
7. In the *pcb_cursor_angle* text box, type `45`.
8. Click *OK* to dismiss the dialog box.
9. Use one of these commands:
   - ❑ At the command console window prompt, type `set`.
   - ❑ In the User Preferences Editor dialog box, click *List All.*

   The Defined Variables list appears.

   Note: If you are running a release later than 15.0, choose *Tools – Utilities – Env Variables* from the menu bar.
10. Scroll down the Defined Variables list until you reach *pcb_cursor* and *pcb_cursor_angle.* Note that the values are the ones that you just set in the User Preferences Editor dialog box.
11. Close the list.
12. Locate your local `env` file in the `pcbenv` directory, open it with a text editor, and check the settings for *pcb_cursor* and *pcb_cursor_angle.*

    All changes you make in the User Preferences Editor dialog box are also saved in your `env` file. If the `env` file does not exist, Allegro PCB Editor creates one when you set variables. For additional information on the env file, see env File on page 22.

    If you edit the `env` file, be sure that you add the new information after the `source $TELENV` statement and before the `###User Preferences` section. Otherwise, the Allegro PCB Editor may overwrite the entries.

13. Choose *Setup – User Preferences.* In the User Preferences Editor dialog box, change the value in the *pcb_cursor* list to *cross.*

14. Click *OK* to dismiss the User Preferences Editor dialog box.

Summary

You now know how to set environment variables in the User Preferences Editor, in the `env` file, and using the `set` command.

You have learned the following:

- **New menu bar command:** *Tools – Utilities – Env Variables*
- **New console command:** `set`

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `set` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 5-4: Running Commands with Strokes to learn how to run commands using strokes.

Lesson 5-4: Running Commands with Strokes

Overview

You can define graphical shapes that Allegro PCB Editor interprets as commands. Allegro PCB Editor analyzes the relative motion of the stroke created by the mouse movement, calculates a number for this motion, and then determines whether or not a command is assigned to the shape. Allegro PCB Editor provides a default stroke file, called `allegro.strokes,` which is a binary file.

You can also create your own stroke definitions. Stroke definitions and their associated commands are stored in stroke files that are created and read by the Stroke Editor. You can store these files anywhere on your workstation.

To create a stroke file, or edit an existing stroke file, see the `stroke editor` command in the *Allegro PCB and Package Physical Layout Command Reference.*

The Allegro PCB Editor default `allegro.strokes` file is located in the `$cdsroot\share\pcb\text` directory. Allegro PCB Editor looks for stroke files in this order:

1. Current working directory
2. `\pcbenv` directory
3. `$cdsroot\share\pcb\text` directory

If you create a new stroke file, store it in your current working directory or in the `\pcbenv` directory.

In this lesson, you will learn how to use pre-defined strokes to accomplish tasks.

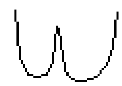
Procedure

1. Open `cds_routed.brd`.
2. Place your cursor in the Design window, then press and hold the `Ctrl` key on the keyboard while you press and hold the right mouse button (`Ctrl+RMB`).

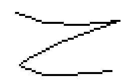
Note: You can set the `no_dragpopup` environment variable by choosing *Setup – User Preferences* from the Allegro PCB Editor menu bar and then choosing *Input* in the *Categories* section of the User Preferences Editor dialog box. By default, you must hold down the `Ctrl` key and depress the right mouse button at the same time when using strokes. Setting this environment variable lets you depress the right mouse button and drag the mouse when using strokes. With this option, however, you lose the ability to choose popup menu items by pressing the right mouse button and dragging the mouse. Instead, you have to click twice with the right mouse button: once to see the popup and a second time to select a popup item.

3. Draw the letter *W* with the cursor anywhere on the design.

   This *W* stroke has been associated through the `alias` command with the `zoom world` command that zooms to fit the entire layout drawing on the screen.



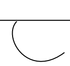
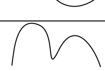
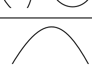
4. Use `Ctrl+RMB` to draw the letter *Z* across an area of the board.



   Allegro PCB Editor has associated the *Z* stroke with the `zoom in` command through the `alias` command. The extents of the zoom area are defined by the diagonal line connecting the upper left tip to the lower right tip of the *Z*. The Z stroke zooms into the area where you drew the *Z*.

5. Using the strokes listed below, experiment with the `copy`, `move`, and `delete` commands.

   The move, copy, and delete strokes select the object under the first point of the stroke. Remember to click the right mouse button in the Design window and choose *Cancel* from the pop-up menu to inactivate a command.

Stroke	**Equivalent Command**	**Key Combinations**
	Copy	F5
	Move	F6
	Zoom In	`F11`
	Zoom World	`Shift` + 12
	Delete	~D

Summary

You now know how to use mouse strokes to accomplish a task. You also know that you can use the Stroke Editor to create your own stroke file.

You have learned the following:

- **New terms:** stroke, Stroke Editor
- **New environment variable:** `no_dragpopup`
- **New console command:** `stroke editor`
- **New file:** `allegro.strokes` file

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `stroke editor` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 5-5: Scripting to learn how to record and play scripts.

Lesson 5-5: Scripting

Overview

You can use scripting for design setup and performing design functions. You can combine any sequence of commands in a script. Then you can use the `alias` or `funckey` commands to define shortcuts to replay the scripts. When you use the scripting feature, Allegro PCB Editor saves all your menu selections and mouse picks in a text file.

Note: The `funckey` command is available only in releases later than Release 15.0.

Macros are like scripts because they let you perform repetitive actions, such as complex geometric operations, on a drawing. The difference, however, is that scripts record from absolute coordinates while macros record from relative coordinate positions in a drawing, using the starting location you specify.

Two variables in the UI category of the User Preferences Editor affect how scripts work. The `noformscriptbutton` environment variable lets the script use `add` or `reset` commands. The `script_keepformopen` environment variable keeps the script form open after completing the script. This can be useful when repeating a script several times. For information about setting environment variables, see Lesson 5-3: Setting Environment Variables on page 162.

In this lesson, you will learn how to perform these tasks:

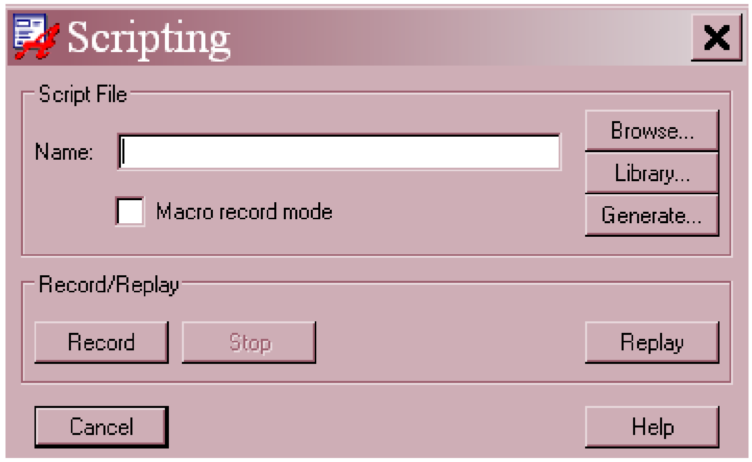
- Starting a Script File Recorder
- Stopping the Script File Recorder
- Testing the Script File (colors.scr)

Procedure

Starting a Script File Recorder

1. With `cds_routed.brd` displayed in the Design window, choose *File – Script* from the menu bar.

   The Scripting dialog box appears.



2. In the *Name* text box, type `colors`. Do not press the `Enter` or `Return` key.

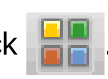
3. Click *Record*.

   The Scripting dialog box disappears and you are ready to begin recording. Everything you do from this point forward is entered into the `colors.scr` script file. Notice the *Rec colors* message in the Status bar.

4. Use one of these commands to display the Color and Visibility dialog box:

   ❑ From the menu bar, choose *Display – Color/Visibility.*

   ❑ At the command console window prompt, type `color192`.

   ❑ At the icon toolbar, click .

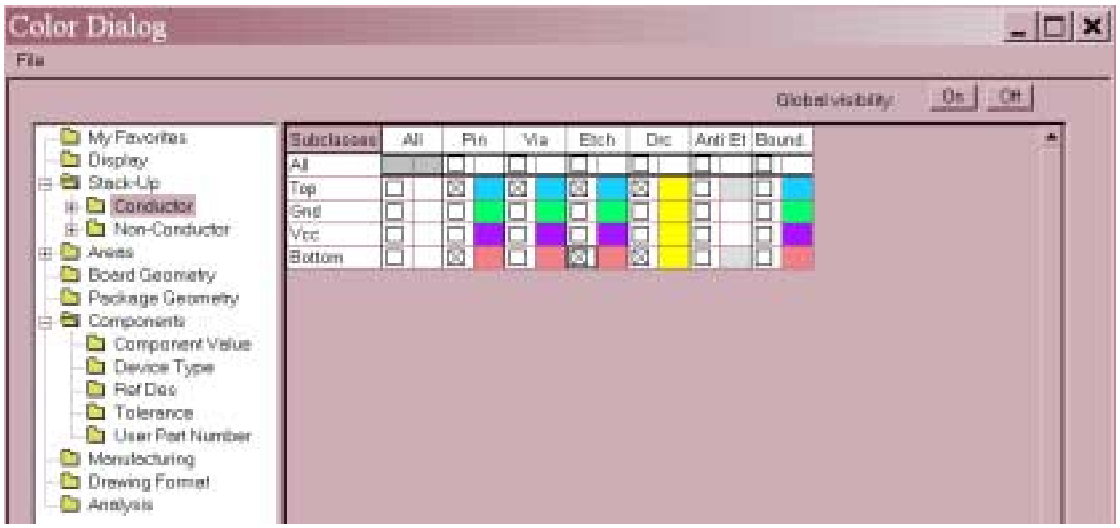
5. Near the top right of the Color dialog box, click *Global Visibility* to *Off*.

   When the confirmer dialog box appears asking if you want to make all classes invisible, click *Yes*. This action resets all the colors to *off*, so you can begin setting them to the colors you like.

6. In the left pane, click *Components*.

7. Under *Ref Des*, enable the visibility box for the subclass *Assembly_Top*. An X in the box indicates the subclass is turned *on*.

8. In the left pane, click *Board Geometry* and enable the visibility for the *Outline* subclass.
9. Under *Package Geometry*, enable the visibility for *Assembly_Top.*
10. Click *Stack-Up* and choose *Conductor*.
11. Enable visibility for subclasses as shown in the figure below, then click *Apply*, and then *OK*.



12. Notice the word *Rec colors* in the Status bar on the lower right. You are still in record mode.

Procedure

Stopping the Script File Recorder

1. From the menu bar, choose *File – Script* to display the Scripting dialog box.
2. Click *Stop* to stop the script file from recording.

   All the modified visibility and color assignments are captured in the *colors.scr* file.
3. Click *Cancel* to close the Scripting dialog box.

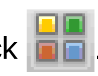
4. From the menu bar, choose *File – File Viewer* to view the `colors.scr` ASCII file.

   The file is located in your working directory (`proj2`). Be sure to change the file type in the browser menu from *(*`*.log`*)* to *All Files* `(*.*)` so the `colors.scr` file appears.

5. Close the `colors.scr` file when you are done viewing it.

Procedure

Testing the Script File (colors.scr)

1. Use one of these commands to display the Color dialog box:
   - ❑ From the menu bar, choose *Display – Color/Visibility.*
   - ❑ At the command console window prompt, type `color192`.
   - ❑ At the icon toolbar, click .
2. Near the top right of the Color dialog box, click *Global Visibility* to *Off*.

   When confirmer dialog box appears asking if you want to make all classes to invisible, click *Yes*.

3. Click *OK* to close the Color dialog box.

   Because the visibility for all classes is turned off, nothing displays in the Design window.

4. Use one of these commands to replay the script:
   - ❑ From the Scripting dialog box, type the name of the script in the *Name* text box and click *Replay*.
   - ❑ At the command console window prompt, type:

     ```
     replay colors
     ```

   This command replays the script file you created, and automatically sets the visibility and color assignments.

Summary

You now know how to start, stop, and test a script.

You have learned the following:

- **New menu bar command:** *File – Script*
- **New console command:** `replay`
- **New environment variables:** `noformscriptbutton`, `script_keepformopen`

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `replay` command in the *Allegro PCB and Package Physical Layout Command Reference.*

What's Next

Go to Lesson 5-6: Using Color Visibility Views to learn how to create and use a color visibility view and restore a previous view used in the current session.

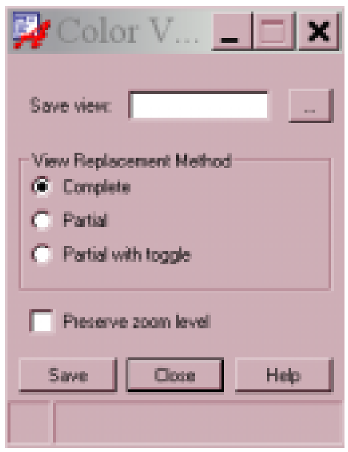
Lesson 5-6: Using Color Visibility Views

Overview

A color visibility view saves the visibility of the Allegro PCB Editor classes and subclasses as a collection of layer visibility settings that you can apply to subsequent Allegro PCB Editor designs. You save the settings in a file that is stored in the current directory with a `.color` extension. A color view also displays film record visibility settings stored in the current design.

Procedure

1. Use one of these commands to display the Color Views dialog box and save a color visibility view:
   - ❑ From the menu bar, choose *View – Color View Save.*
   - ❑ At the command console window prompt, type `colorview create`.



2. Click **...** to browse to your current working directory where the `cds_routed.brd file` is located.
3. Type the name of the color visibility view, `test`, in the *File name* text box and click *Save*.
4. Choose a method in the *View Replacement Method* frame in the Color Views dialog box.

   If you choose either of the *Partial* view replacement methods, you must change visibility settings in the Color dialog box (using the `color192` command) or in the *Visibility* window pane of the Control Panel.
5. To include zoom points with the color view, enable *Preserve Zoom Level*.
6. Click *Save* and then *Close*.
7. In the *Visibility* window pane of the Control Panel, turn off all the *Conductor* layers. Notice the changes in your drawing.

8. Use one of the following methods to restore the visibility layers you saved in the `test` file.

   - In the *Views* list box on the *Visibility* window pane of the Control Panel, choose *test*.

     Color views (`.color` files) appear in the Views list box as *File: <name>.* Film record names display appear as *Film: <name>*, unless you suppress the film record names from the list of color views in the Visibility window pane of the Control Panel. Suppress these names by selecting the `color_nofilmrecord` environment variable in the *Control_panel* section of the User Preferences Editor dialog box. See Lesson 5-3: Setting Environment Variables on page 162.

   - At the command console window prompt, type `colorview load` and then choose *test* in the Colorview Load dialog box. Click *Save*.

   The visibility layers are restored.

9. To toggle between the previous color visibility view and the one you just created, use one of these commands:

   - From the menu bar, choose *View – Color View Restore Last*.

   - At the command console window prompt, type `colorview restore`

10. Exit the Allegro PCB Editor. Do not save any changes to `cds_routed.brd`.

Note: You can also save or restore images, which are the same as views except that they also include zoom points with the color views. Use the `images` command at the Allegro PCB Editor command console window prompt.

Summary

You now know how to create and use a color visibility view, and restore the previous visibility view used in the current session.

You have learned the following:

- **New term:** color visibility view
- **New menu commands:** *View – Color View Save, View – Color View Restore Last*
- **New console commands:** `colorview create, colorview load, colorview restore, images`
- **New environment variable:** `color_nofilmrecord`
- **New dialog box:** Color Views

For More Information

See:

- the *Getting Started with Physical Design* user guide in your documentation set.
- `colorview create` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `colorview load` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `colorview restore` command in the *Allegro PCB and Package Physical Layout Command Reference.*
- `images` command in the *Allegro PCB and Package Physical Layout Command Reference.*

You have successfully completed the *Allegro PCB Editor Tutorial*.