

Allegro®

Project Manager User Guide

Product Version 23.1
September 2023

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Allegro® Design Entry HDL contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission. All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

1

<u>Introduction to the Allegro Project Manager</u>	7
<u>Project Manager Functions</u>	7
<u>Project Structure</u>	8
<u>Project Files</u>	9
<u>Local Project Files</u>	10
<u>Site Project File</u>	11
<u>Installation Project File</u>	16
<u>Project Flows</u>	17
<u>Starting Project Manager</u>	18
<u>Product Choices</u>	18
<u>Files Created for Your New Project</u>	20

2

<u>Creating Projects</u>	23
<u>Creating a Design Project</u>	23
<u>Creating a Library Project</u>	27

3

<u>Managing Library Projects</u>	29
<u>Library Management Flow</u>	30
<u>Setting Up a Template</u>	33
<u>Viewing a Symbol</u>	34
<u>Viewing and Verifying a Footprint</u>	34
<u>Importing from XML to Design Entry HDL</u>	34
<u>Converting XML to Capture</u>	35
<u>Importing from Capture to Design Entry HDL</u>	35
<u>Verifying VHDL/Verilog Support</u>	35
<u>Exporting from Design Entry HDL to XML</u>	36
<u>Exporting from Design Entry HDL to XML - Select Package</u>	36

Allegro Project Manager User Guide

<u>Converting from Capture to XML</u>	36
<u>Exporting from Design Entry HDL to Capture</u>	37
<u>Selecting Symbols</u>	37
<u>Verification Subflow</u>	39
<u>Verifying Design Entry HDL Libraries with Template</u>	40
<u>Verifying Design Entry HDL Libraries Using Rules Checker</u>	41
<u>Verifying Design Entry HDL Libraries in Verilog Simulation Flow</u>	42
<u>Verifying Design Entry HDL Libraries in Packaging Flow</u>	43
<u>XML to Design Entry HDL Conversion</u>	44
<u>Capture to XML Conversion</u>	46

4

<u>Project Manager Procedures</u>	49
<u>Working with Projects</u>	49
<u>Opening a Project</u>	49
<u>Copying a Project</u>	50
<u>Importing a Project</u>	61
<u>Exporting a Project</u>	61
<u>Closing a Project</u>	62
<u>Importing IFF Designs</u>	62
<u>Starting Tools from Project Manager</u>	64
<u>Starting a Tool from the Tools Menu</u>	64
<u>Starting a Tool from the Project Manager Toolbar</u>	65
<u>Starting a Tool from the Project Flows</u>	66
<u>Viewing Project Settings</u>	69
<u>Viewing Project Libraries</u>	72
<u>Viewing Running Tools</u>	74
<u>Customizing Project Manager</u>	74
<u>Customizing the Project Manager Tools</u>	74
<u>Customizing the Project Manager Display</u>	78
<u>Customizing the Project Manager Toolbar</u>	79
<u>Customizing the Project Flow</u>	79
<u>Adding a Customized Flow</u>	84
<u>Starting Project Manager from the Command Line</u>	86

5

<u>Setting Up Projects</u>	89
<u>Overview</u>	89
<u>Changing the Root Design for a Project</u>	89
<u>Creating a New Root Design for a Project</u>	89
<u>Editing the cds.lib File</u>	90
<u>Adding Libraries to the cds.lib File</u>	91
<u>Selecting Libraries for a Project</u>	92
<u>Adding Physical Part Table Files to a Project</u>	93
<u>Setting Up Tools</u>	94
<u>Specifying the Application Temp Directory</u>	95
<u>Selecting a Text Editor</u>	96
<u>Selecting a Property File</u>	96
<u>Setting Up a Log File</u>	97
<u>Selecting an Expansion Type</u>	98
<u>Selecting the Configuration for Expansion</u>	99
<u>Editing a Configuration</u>	99
<u>Creating a New Configuration View</u>	100
<u>Selecting Views for the Project</u>	100

A

<u>Menu Commands</u>	103
<u>File – New</u>	105
<u>File – New – New Design</u>	105
<u>File – New – New Library</u>	105
<u>File – Open</u>	105
<u>File – Close</u>	105
<u>File – Project Path</u>	105
<u>File – Copy Project</u>	106
<u>File – Import</u>	106
<u>File – Import IFF – RF-PCB</u>	106
<u>File – Import IFF – Standard</u>	106
<u>File – Export</u>	107
<u>File – File Viewer</u>	107

Allegro Project Manager User Guide

<u>File – Import IFF</u>	107
<u>File – Change Product</u>	107
<u>File – Exit</u>	107
<u>View – Toolbar</u>	107
<u>View – Status Bar</u>	108
<u>View – Project Settings</u>	108
<u>View – Project Libraries</u>	108
<u>View – Running Tools</u>	109
<u>View – Hide Flow</u>	109
<u>Tools – Library Tools</u>	109
<u>Tools – ViewCaptureSymbol</u>	110
<u>Tools – Variant Editor</u>	110
<u>Tools – Programmable IC</u>	110
<u>Tools – Simulate</u>	111
<u>Tools – EDIF 300</u>	111
<u>Tools – SigXplorer</u>	111
<u>Tools – Rules Checker</u>	111
<u>Tools – CRefer</u>	111
<u>Tools – Archive</u>	112
<u>Tools – Packager Utilities</u>	112
<u>Tools – Hierarchy Editor</u>	112
<u>Tools – Setup</u>	112
<u>Tools – Design Entry</u>	112
<u>Tools – Design Sync</u>	113
<u>Tools – PCB Editor</u>	113
<u>Tools – Allegro SI</u>	113
<u>Options – Customize</u>	113
<u>Web – Back</u>	113
<u>Web – Forward</u>	114
<u>Web – Home</u>	114
<u>Web – Stop</u>	114
<u>Web – Reload</u>	114
<u>Web – Add To Favorites</u>	114
<u>Web – Go to Favorites</u>	114
<u>Flows – Board Design</u>	115
<u>Flows – Library Management</u>	115

Allegro Project Manager User Guide

<u>Flows – Programmable IC</u>	115
--------------------------------	-----

B

<u>Project Manager Dialog Box Help</u>	117
<u>New Project Wizard</u>	118
<u>New Project Wizard – Name and Location</u>	118
<u>New Project Wizard – Libraries</u>	118
<u>New Project Wizard – Summary</u>	119
<u>Product Choices Dialog Box</u>	120
<u>Customize Dialog Box</u>	121
<u>Add Tool Dialog Box</u>	123
<u>Setup Template Dialog Box</u>	124
<u>View Symbol Dialog Box</u>	125
<u>Footprint Dialog Box</u>	126
<u>Import from XML to Design Entry HDL Dialog Box</u>	127
<u>Import from Capture to Design Entry HDL Dialog Box</u>	128
<u>VHDL/Verilog Support Dialog Box</u>	129
<u>Export from Design Entry HDL to XML Dialog Box</u>	129
<u>Export from Design Entry HDL to XML - Select Package Dialog Box</u>	130
<u>Convert from Capture to XML Dialog Box</u>	131
<u>Export from Design Entry HDL to Capture Dialog Box</u>	132
<u>Select Symbols Dialog Box</u>	132
<u>Verify Design Entry HDL Libraries with Template Dialog Box</u>	133
<u>Verify Design Entry HDL Libraries Using Rules CheckerDialog Box</u>	134
<u>Verify Design Entry HDL Libraries in Verilog Simulation Flow Dialog Box</u>	136
<u>Verify Design Entry HDL Libraries in Packaging Flow Dialog Box</u>	137
<u>Project Setup – Global Tab</u>	139
<u>Project Setup – Part Table Tab</u>	142
<u>Project Setup – Tools Tab</u>	144
<u>Project Setup – Expansion Tab</u>	147
<u>Project Setup – Views Tab</u>	150
<u>Index</u>	153

Allegro Project Manager User Guide

Introduction to the Allegro Project Manager

This chapter covers the following topics:

- [Project Manager Functions](#)
- [Project Structure](#)
- [Project Files](#)
- [Project Flows](#)
- [Starting Project Manager](#)
- [Product Choices](#)
- [Files Created for Your New Project](#)

Project Manager Functions

The Project Manager is the interface to the Cadence board design solution and library management. The Project Manager tool can be used for the following tasks:

- Create design projects or library projects

Design projects are projects created by designers where the basic aim is creation of designs. Library projects are created by librarians for the sole purpose of library creation and management.
- Set up projects

You can choose libraries for your project, as well as options such as Physical Part Table files, property files, expansion types, configurations, and view names.
- Import, export, and archive projects

- Launch tools such as Design Entry HDL, PCB Editor, Allegro SI, Library Explorer, Part Developer, and Pad Stack Editor.
- View project settings and libraries in a convenient tree form, and keep track of all tools running in a session.

The Project Manager flow can be customized to your requirements. You can add and remove tools from the flow and use custom icons. The flow can also be converted to a toolbar to conserve space on your desktop.

Project Structure

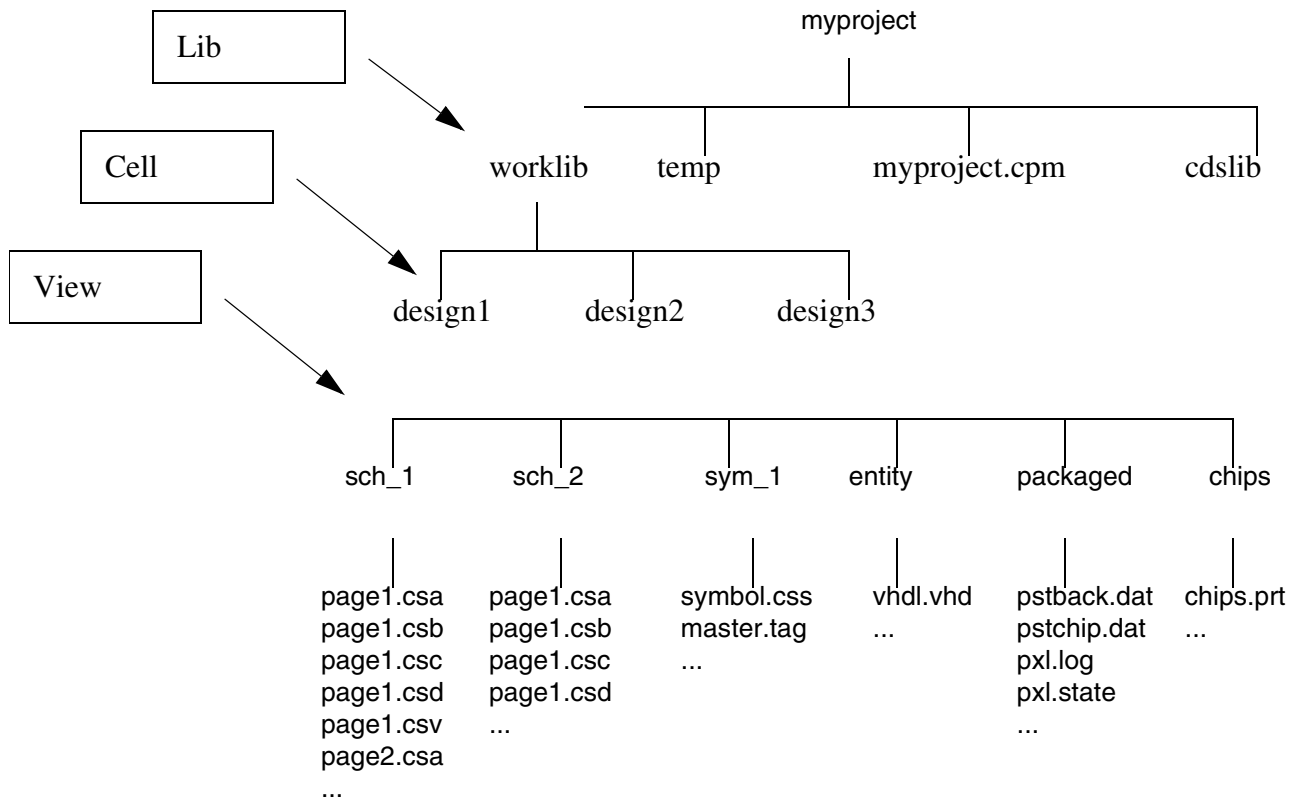
This design and library data is stored in what is called the `lib-cell-view` architecture.

The libraries get installed during the time of the setup of the Cadence tools. By default, the libraries get copied at `<your_install_dir>/share/library`. Each library has a directory, with subdirectories for each cell in the library.

Under each cell, there are further subdirectories representing views, such as entity, chips and so on, which show the cells in a unique manner - schematic, symbolic, or layout.

Each of the views contains files that store information about the view. The names of these files are fixed or may contain a variable portion controlled by tools (for example, multisheet schematics). For example, the `chips` folder stores the `chips.prt` file, which stores information like pin names and electrical information for the part.

Example



Project Files

The Project Manager manages all the information about a project—such as its libraries, physical part table files, log files, property files, and setup defaults for tools—through project files.

There is a hierarchy of files followed by the Project Manager while looking for directives set for a project:

- Local Project Files—named `<project>.cpm`
- Site Project File—named `site.cpm`
- Installation Project File—named `cds.cpm`

Note: If your site project file is named by `cds.cpm` by mistake, Project Manager assumes that the installation project file has been located and does not search the Cadence installation file.

Local Project Files

When you create a new project, the Project Manager creates a project file called `<projectname>.cpm` in the project directory. Each project has one project file. The `<projectname>.cpm` file contains all the setup information that you specified for your project. It has the following:

- The name of the top-level design and the library in which it is located
- The list of project libraries
- The physical part tables selection
- Changes to the default view names
- The name and location of the text editor for editing text files from Cadence tools
- The name and location of the property file
- The name and location of the log file
- The name and location of the application temp directory, which is the directory in which applications such as Design Entry HDL store temporary files.
- Setup directives for individual tools such as Design Entry HDL, Packager-XL, and the Project Manager.
- Directives for customizing the Project Manager (a customized Tools menu or customized flows).

The default setup information is maintained in an installation project file (`cds.cpm`) shipped by Cadence. The defaults in the `cds.cpm` file apply to all your projects. If you want to change these defaults, create a site project file (`site.cpm`) for your site.

When you open a project, the Project Manager gets the setup directives you specified for that project from the `<projectname>.cpm` file and the defaults for the others from the `site.cpm` and `cds.cpm` files. Your setup directives always have precedence over the `site.cpm` directives, which in turn have precedence over the `cds.cpm` directives.

You can view the project settings for a project with the *View – Project Settings* command.

Project File (.cpm) example

```
( Machine generated file created by SPI )  
( Last modified was 11:38:31 Thursday, October 09, 1997 )  
( NOTE: Do not modify the contents of this file. If this is regenerated by )  
( SPI, your modifications will be overwritten. )
```

```
START_GLOBAL  
use_library_ppt 'ON'  
design_name 'poa'  
design_library 'poa'  
library 'poa' 'standard' 'pic' 'poa_lib' 'element'  
temp_dir 'temp'  
cpm_version '@'  
session_name 'ProjectMgr12919'  
cdsprop_file ''  
ppt './ptf/poa.ppt'  
EXCLUDE_PPT  
INCLUDE_PPT  
END_GLOBAL  
  
START_PKGRXL  
state_wins_over_design 'ALL'  
END_PKGRXL
```

Site Project File

You create the site project file, called `site.cpm`, in the `<your_inst_dir>/share/local/cdssetup/projmgr` directory when you want to specify default setup options for all the projects at your site. The directives in this file have precedence over the installation project file (`cds.cpm`) and the local project file (`<projectname>.cpm`) has precedence over the `site.cpm` file.

You can customize the default settings for all your projects by creating the `site.cpm` file. To create a `site.cpm` file, either use a copy of an existing project file, or create a dummy project and use its project file to define your site settings.

To create a site project file for all the projects at your site,

1. Choose *Tools – Setup*.
2. In each tab of the *Project Setup* dialog box, specify the default setup information you want for all projects. For information about the setup options, click the *Help* button in the dialog box.
3. Click *Apply* to save your changes.
4. Close *Project Setup* by clicking *OK*.

5. Choose *File – Export*.

6. In the *Export Project* dialog box,

- ☐ Type `site.cpm` in the *File Name* box.
- ☐ In the *Folders* list, select `<your_inst_dir>/share/local/cdssetup/projmgr`, where `<your_inst_dir>` is the directory in which you have installed Cadence tools.
- ☐ Ensure that the *Save File as Type* box displays *Project Files (*.cpm)*.
- ☐ Ensure that the *Full Settings* option is not selected.

7. Click *OK* in the *Export Project Setup* dialog box.

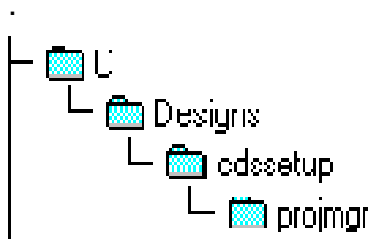
Creating a Custom Site Environment

The site.cpm File

If you do not place the `site.cpm` file in the `<your_inst_dir>/share/local/cdssetup/projmgr` directory, you must set a `CDS_SITE = location` environment variable that specifies the location of the site project file. The site location must have the following directory structure:

```
cdssetup/projmgr/site.cpm
```

For example, if you want to set your `CDS_SITE = C:\Designs`, you must create the following directory structure and place the `site.cpm` file in the `projmgr` directory:



The concepthdl.scr File

If you have set the `CDS_SITE` environment variable to another location, such as `/hm/common/`, you need to ensure that the `concepthdl.scr` file is at `/hm/common/cdssetup/concept/`. Otherwise, backannotation from Variant Editor will not work.

Allegro Project Manager User Guide

Introduction to the Allegro Project Manager

Flows

If you have any custom Project Manager flows, maintain them at `$CDS_SITE/cdssetup/projmgr/flows` using the same directory structure as at `<your_inst_dir>/share/cdssetup/projmgr/flows/`.

Other Customized Files

If you have customized any of the following files and want the changed version to be available for all projects at your site, copy them to the location listed in the following table. This will ensure that the customized information is available even when you install a newer version of Cadence PSD software.

Files and Descriptions	Location
<code>cds.lib</code> (lists libraries used in the project)	Place at <code>\$CDS_SITE/cdssetup</code> .
<code>bom.callouts</code> (mechanical parts to be added in the BOM reports)	Copy from <code><your_inst_dir>/share/cdssetup/</code> to <code>\$CDS_SITE/cdssetup/</code> .
<code>cdsinfo.tag</code> (project-specific information, including the name of the data management system, if any, used in the project)	Copy from <code><your_inst_dir>/share/cdssetup/</code> to <code>\$CDS_SITE/cdssetup/</code> .
<code>cdsprop.paf</code> (information about properties)	Copy from <code><your_inst_dir>/share/cdssetup/</code> to <code>\$CDS_SITE/cdssetup/</code> .
<code>cdsprop.tmf</code> (information about text macros)	Copy from <code><your_inst_dir>/share/cdssetup/</code> to <code>\$CDS_SITE/cdssetup/</code> .
<code>cjedectype.txt</code> (compatible JEDEC types in the Variant Editor tool)	Copy from <code><your_inst_dir>/share/cdssetup/</code> to <code>\$CDS_SITE/cdssetup/</code> .

Allegro Project Manager User Guide

Introduction to the Allegro Project Manager

propflow.txt (the default property flow setup in Packager Setup)	Copy from <your_inst_dir>/share/cdssetup/ to \$CDS_SITE/cdssetup/.
template.bom (the default template for BOM reports)	Copy from <your_inst_dir>/share/cdssetup/ to \$CDS_SITE/cdssetup/.
xilfam.dat (mapping information between a Xilinx family and a specific library and architecture)	Copy from <your_inst_dir>/share/cdssetup/ to \$CDS_SITE/cdssetup/.
xmodules.dat (modules that have to be excluded for cross-referencing and plotting)	Copy from <your_inst_dir>/share/cdssetup/ to \$CDS_SITE/cdssetup/.
concepthdl_key.txt (Design Entry HDL shortcut keys)	Copy from <your_inst_dir>/share/cdssetup/concept/ to \$CDS_SITE/cdssetup/concept/.
concepthdl_menu.txt (Design Entry HDL menus)	Copy from <your_inst_dir>/share/cdssetup/concept/ to \$CDS_SITE/cdssetup/concept/.
template.tsg (information related to the graphical attributes of the symbols and additional pin and symbol properties)	Copy from <your_inst_dir>/share/cdssetup/concept/genview/ to \$CDS_SITE/cdssetup/concept/genview/.
ceref.dat (template options of CRefer)	Copy from <your_inst_dir>/share/cdssetup/creferhdl/ to \$CDS_SITE/cdssetup/creferhdl/.
allegro.ilinit (PCB Editor SKILL initialization file)	Copy from <your_inst_dir>/share/pcb/text/ to \$CDS_SITE/pcb/text/.
cuimenu folder (updated PCB Editor menus)	Copy from <your_inst_dir>/share/pcb/text/ to \$CDS_SITE/pcb/text/.

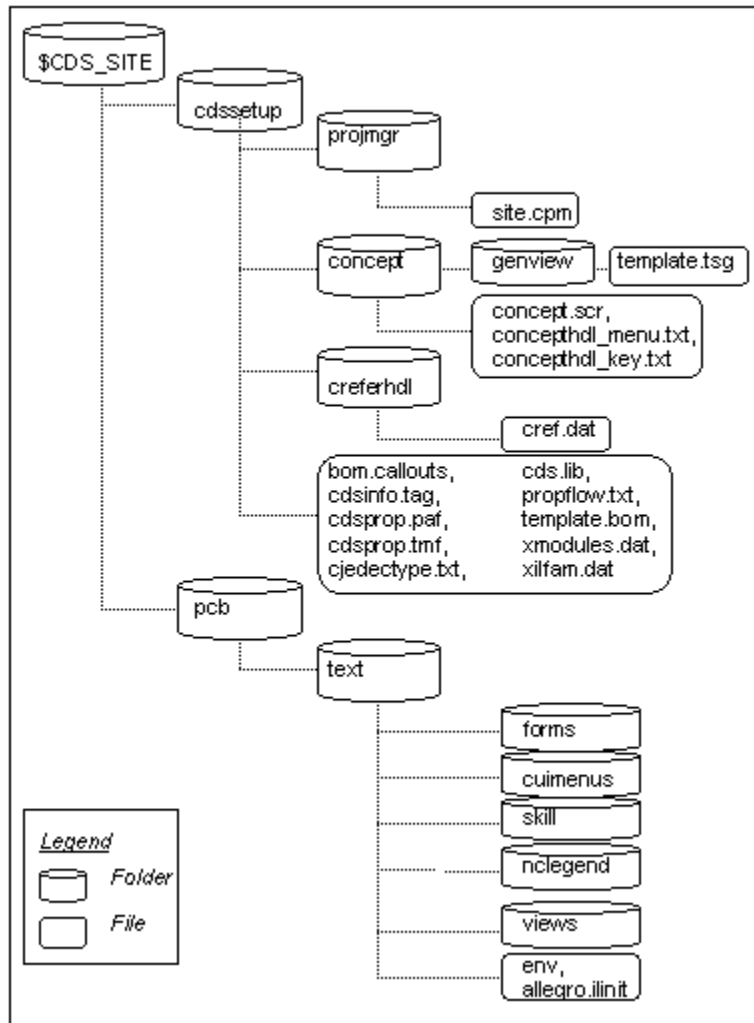
Allegro Project Manager User Guide

Introduction to the Allegro Project Manager

env (paths to PCB Editor libraries and other site settings for PCB Editor)	Copy from <your_inst_dir>/pcb/text/ to \$CDS_SITE/pcb/text/.
forms folder (all forms called from Allegro SKILL code)	Copy from <your_inst_dir>/share/pcb/text/ to \$CDS_SITE/pcb/text/.
nclegend folder (PCB Editor templates from NCDRIII legend)	Copy from <your_inst_dir>/share/pcb/text/ to \$CDS_SITE/pcb/text/.
skill folder (custom Allegro SKILL code)	Place at \$CDS_SITE/pcb/text/.
views folder (Allegro extract command files)	Copy from <your_inst_dir>/share/pcb/text/ to \$CDS_SITE/pcb/text/.

Note: The `cdsprop.txt` file need not be copied as you should not be modifying this file.

The directory structure for a custom site environment must be as depicted in the following chart:



Installation Project File

The installation project file, called `cds.cpm`, is shipped by Cadence and is in the `<your_inst_dir>/share/cdssetup/projmgr` directory. The `cds.cpm` file contains default setup directives for all projects and tools. The Project Manager obtains defaults from this file for setup options that are not defined in the `<projectname>.cpm` or `site.cpm` files. Do not modify this file. If you want to change the defaults for a set of projects, create a site project file (`site.cpm`).

The setup directives you specify (that is, the directives in the `<projectname>.cpm` file) always have precedence over the `site.cpm` directives, which in turn have precedence over the `cds.cpm` directives. When you open a project, the Project Manager gets the setup directives you specified for that project from the `<projectname>.cpm` file and the default values for the others from the `site.cpm` file. If they are not defined in the `site.cpm` file either, the Project Manager obtains the default values from the `cds.cpm` file.

Project Flows

The Project Manager is also an HTML browser and the project flow is defined in a simple HTML document. You can customize the project flow by replacing it with HTML pages that you create—for individual projects or for all the projects at your site.

The standard Cadence Board Design Flow consists of two HTML files, `main.htm` and `home.htm`, which are in the Cadence installation hierarchy at `<your_inst_dir>/share/cdssetup/projmgr/flows`. The `home.htm` file is loaded when no project file is currently open. It has links for opening an existing project or creating a new project. The `main.htm` file is loaded when a project file is opened.

The HTML files contain HREFs. When the Project Manager evaluates an HREF, it first looks at its own list of tools to see if the URL matches a tool name. If it does, the tool is launched as a separate process. If no match is found, the Project Manager attempts to load the referenced URL. This allows the Project Manager to be used as a flow manager, tool launcher, and internet/intranet browser.

For more information, see the following topics:

- [Customizing the Project Flow](#)
- [HTML Code for Default Board Design Flow](#)
- [Example: Creating a Simple Text-Based Flow](#)
- [Example: Creating a Graphical Multiple Page Flow](#)

To rectify the problem,

1. Close all Cadence tools
2. Set the environment variable `MWUSE_SYSTEM_COLOR_MAP` to 0. For example,

on C shell, type the following command:

```
% setenv MWUSE_SYSTEM_COLOR_MAP 0
```

and on sh, type the following command:

```
$ MWUSE_SYSTEM_COLOR_MAP=0; export MWUSE_SYSTEM_COLOR_MAP
```

3. Restart the tools.

Starting Project Manager

You start Project Manager using the following steps:

1. Type `projmgr` on the command line and press Enter.
2. The Project Manager Product Choices window appears offering the various choices.
See [“Product Choices”](#) on page 18 for more information.
3. Select a product.
4. Click *OK*. The Project Manager GUI opens showing three icons.
5. To open an existing project, click the Open Project icon. Click the Create Design Project icon to create a new design project or the Create Library Project icon to create a new library project.

Note: You must be on the Common Desktop Environment (CDE) on a Sun workstation to run the Design Entry HDL set of tools, including Project Manager.

See [Creating a Design Project](#) and [Creating a Library Project](#) for more information.

Product Choices

You can choose a product suite in which you want to run Project Manager. Changing product suites allows you to access components that are not available in the current product suite. The product suites available for use are displayed in the list.

How to Access

The Cadence Product Choices dialog box is invoked when:

- you are using the tool for the first time; and on all subsequent invocations unless you specify the default choice.
- you choose *File – Change Product*.

Setting a Default Product Choice

To prevent the Cadence Product Choices dialog box from appearing every time you run Project Manager, complete the following steps.

1. Select the product suite to be used as the default choice.
2. Select the *Use as Default* check box to invoke the selected product suite every time you invoke Project Manager.

Selecting the *Use as Default* check box writes the product choice in registry. The Project Manager interface changes to reflect the selected product suite and will open with this product suite until you change the default setting.

3. Click *OK*.

To change the default product suite:

1. Choose *File – Change Product* in Project Manager.
2. Select the required product suite from the list of choices in the Cadence Product Choices dialog box.
3. Select the *Use as Default* check box
4. Click *OK*.

Specifying Product Choice from Command Line

If you invoke Project Manager from command line, you can use the `-product` option to prevent the Cadence Product Choices dialog box from appearing every time.

The syntax for using this option is:

```
projmgr -product <license_string>
```

You can choose one of the following license strings:

- concept_hdl_expert
- concept_hdl_studio
- pcb_librarian expert
- concept_hdl_expert
- concept_hdl_studio
- allegro_performance

- Allegro_Design_Editor_620
- pcb_librarian_expert
- Allegro_Frontend_PCB_Solution
- Allegro_Venture_SDA
- Allegro_Enterprise_SDA
- Allegro_Enterprise_PCB_Designer
- Allegro_Venture_PCB_Designer
- specctraquest_si_expert

Note: License strings are not case-sensitive.

Disabling License Check

To ensure that only the product suites for which you have licenses available are displayed in the Cadence Product Choices dialog box, the application checks with the license server for available licenses. Displaying the list of available licenses takes some time.

If the time taken for displaying the Cadence Product Choices dialog box is high, you can use the CDS_IGNORE_LIC_FEATURE environment variable, with its value set to TRUE, to disable the procedure of checking for the available licenses. Using this variable ensures that the dialog box appears instantly, but displays all the licenses using which you can launch Project Manager. From the list, you need to select the product suite for which you have the license available. For information on the available licenses, contact your license administrator.

Files Created for Your New Project

When you create a new project, the Project Manager creates the following:

- A project file (*<projectname>.cpm*).

The project file contains all the setup information you have specified for the project—the name and location of the top-level design, libraries, view names, physical part tables, and tool setup directives. For more information, see [Project Files](#).

- Four configuration views in the design directory (cell).

The Project Manager creates four configurations for each new project – `cfg_package`, `cfg_verilog`, `cfg_pic`, and `cfg_vhdl`. Each is a directory and contains an `expand.cfg` file.

- An Application Temp directory (`temp`)

Temporary files created by applications such as Design Entry HDL are placed in the `temp` directory. You can delete the contents of this directory. You can also specify your own application temp directory from Project Setup.

In addition, if you created the project in a new directory or in a directory that does not contain a `cds.lib` file, the Project Manager creates the following:

- A `cds.lib` file

For design projects, the `cds.lib` file determines the list of available libraries from which you can choose the project libraries for your project. It contains the logical names of libraries and their physical locations. By default, it includes the path to the installed Cadence libraries.

For library projects, the `refcds.lib` contains all the reference libraries of a project, whereas the `cds.lib` file contains the libraries that you imported or checked out and can modify.

- A `worklib` directory.

The physical name of this directory is `worklib`. You can place your design directories (cells) in this directory.

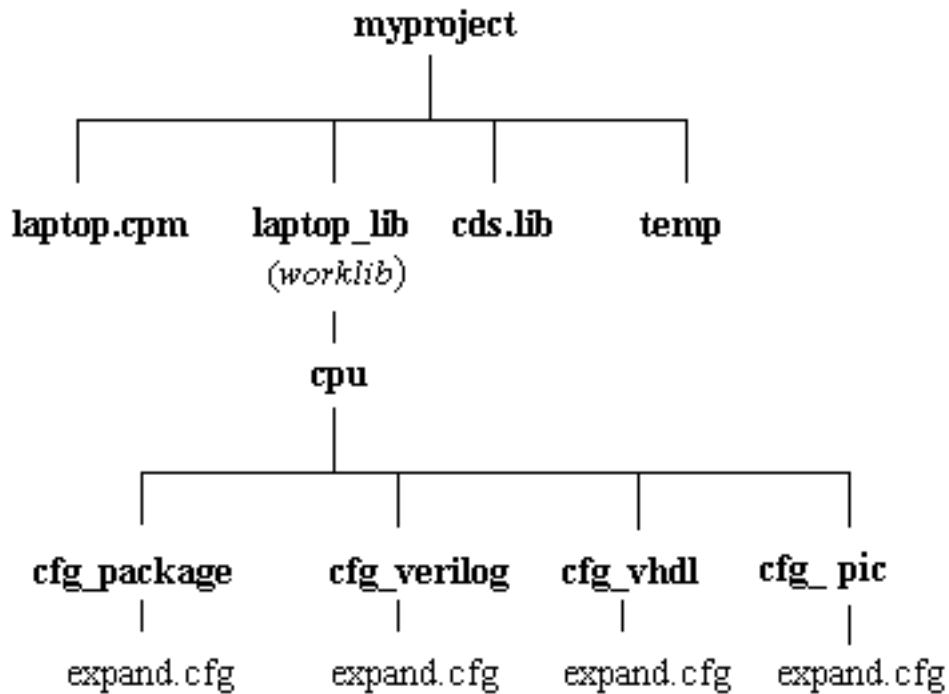
As you use the tools and create designs, non-view log files are added to the project directory, and design data and view-related log files are added to the `worklib` directory.

Allegro Project Manager User Guide

Introduction to the Allegro Project Manager

For example, if you create a new project `laptop` in a new directory `myproject`, and a design `cpu` in `laptop.lib`, you will have the following file structure:

:



Creating Projects

You use the New Project Wizard to create a design on library project. This section discusses the following topics:

- [Creating a Design Project](#)
- [Creating a Library Project](#)

Creating a Design Project

The Project Manager allows you to create a new project using the New Project wizard. To create a design project, perform the following steps:

1. Choose *File – New – New Design*.

The *New Project Wizard-Project Type* window appears. On the first page of the wizard, you specify the name and location of the project.

2. In the *Project Name* field, type your project name.

3. In the *Location* field,

- ☐ Type the complete path of the folder in which you want to create the new project.

- or -

- ☐ Click *Browse*, select a folder in the *Choose Directory* dialog box, and then click *OK*.

The *Location* field is filled in with the path of the folder you selected.

Note: If you want to create the project in a new folder, append a name for the new folder to the path (for example: /cpu). The Project Manager will create the folder.

4. Click *Next*.

The *Project Libraries* dialog box appears with the list of available libraries and project libraries. The New Project Wizard creates a `projectname_lib` library. This

`projectname_lib` is displayed in the *Project Libraries* list. If you created the project in a new folder or in a folder that does not contain a `cds.lib` file, a `cds.lib` file is automatically created. The `cds.lib` file contains an entry for the `projectname_lib` library.

5. Select the libraries for your project by placing them in the *Project Libraries* list.

- ☐ To add one library to the *Project Libraries* list, select the library in the *Available Libraries* list and then click *Add*.
- ☐ To add more than one library to the *Project Libraries* list, press **CTRL** and select the libraries. Then, click *Add*.
- ☐ To add all the libraries in the *Available Libraries* list, click *Add All*.
- ☐ To remove one library from the *Project Libraries* list, select the library and then click *Remove*.
- ☐ To remove more than one library from the *Project Libraries* list, press **CTRL** and select the libraries. Then, click *Remove*.
- ☐ To remove all the libraries from the *Project Libraries* list, click *Remove All*.

6. Choose the search order for your project libraries. The order in which libraries are listed in the *Project Libraries* list determines their search order.

- ☐ To move a library one level up, select the library and then click *Up*.
- ☐ To move a library one level down, select the library and then click *Down*.

Note: You cannot rearrange the order of the *Available Libraries* list.

7. Click *Next*.

8. In the *Design Name* dialog box, specify the top-level drawing for your design. You can choose an existing design from the project libraries or create a new one in any of the project libraries.

To create a new design,

- a.** In the *Library* list, select the library in which you want to create the new design.
- b.** In the *Design Name* field, type a name for the new design.

To select an existing design,

- a.** In the *Library* list, select the library that contains the design.
- b.** Click *Browse*, select a design from the *Existing Cell Names* list, and then click *OK*.

9. Click *Next*. The *Finish* dialog box displays your project specifications.

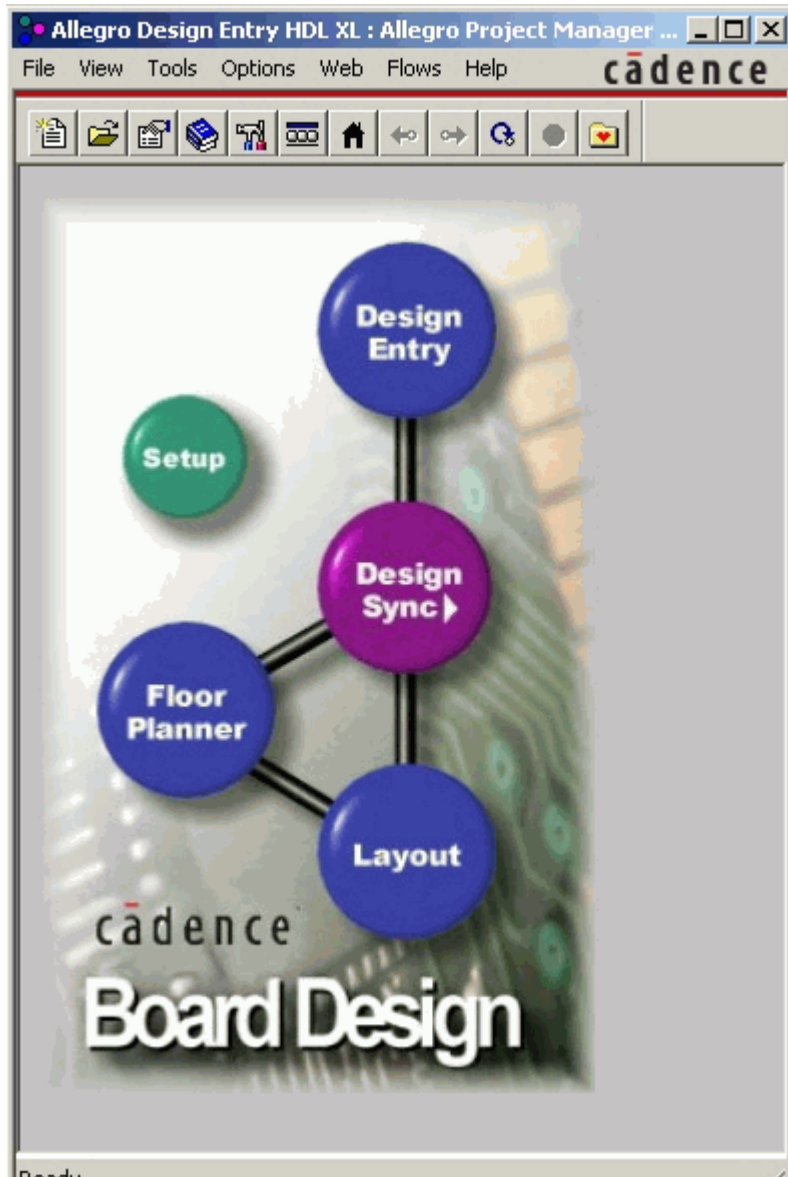
10. Do one of the following.

- ☐ To create the project, click *Finish*.
- ☐ To change the project name, project location, design library, design name, or project libraries, click *Previous* and edit the information you entered in each dialog box. When you finish, click *Next* until the *Finish* dialog box appears. Click *Finish* to create the project.

Allegro Project Manager User Guide

Creating Projects

Project Manager displays the default project flow, with icons for Design Entry HDL, PCB Editor, Design Sync, SI, and Setup.



Creating a Library Project

You can also create a new library project using the New Project wizard. This section discusses how to create a library project using the wizard.

To create a library project, perform the following steps:

1. Choose *File – New – New Library*. The *New Project Wizard - Project Type* window appears.
2. Click *Next*.
The *Project Name and Location* window appears.
3. Type the project name in the *Project name* field.
4. Enter the path to the location for your project in the *Location* field by typing it or by using the browse button to select it, and then click *OK*.
5. Click *Next*. The *Select Libraries* window appears with a list of reference libraries.
6. Select the libraries for your project by adding them to the list.

- ☐ To add a library to the list, click *Add* and select a library from the *Choose Directory* browser that appears.
- ☐ To import libraries from a `cds.lib` file, click *Import* and select a `cds.lib` file from the *Choose library file* browser that appears.
- ☐ To remove a library from the list, select the library and then click *Remove*.

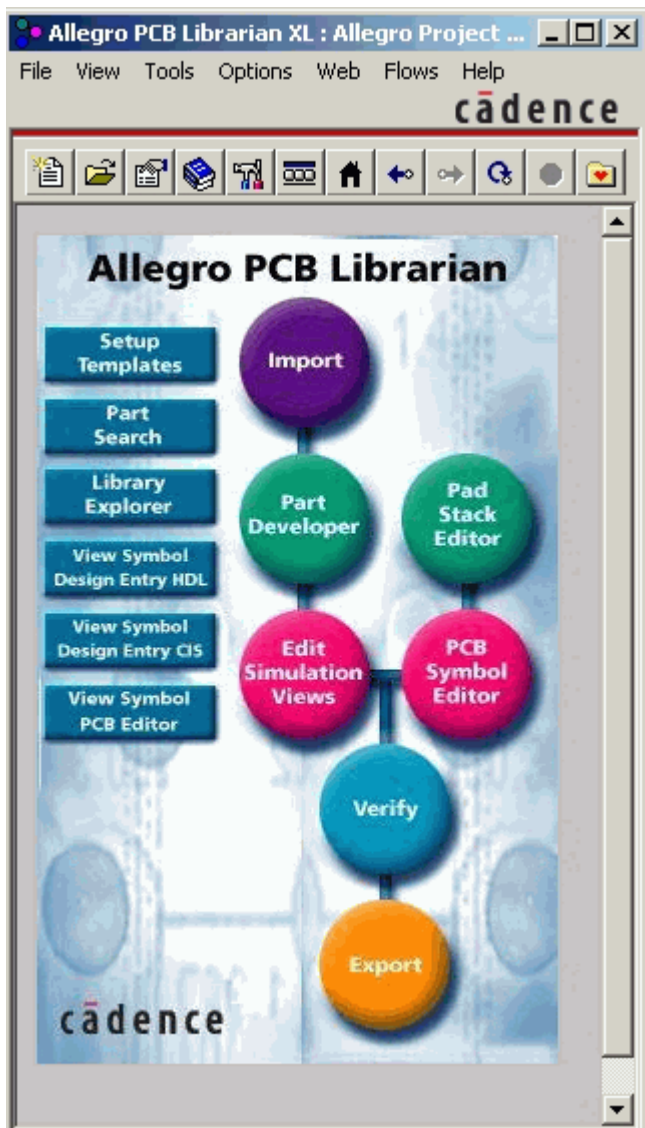
When you click a library, note that the non-editable Physical path to library field at the bottom of the window displays the path to the library name you clicked.

7. Click *Next*. The *Finish* dialog box displays your project specifications.
8. If you need to change any of these specifications, click the *Previous* button to go to previous pages of the wizard. Otherwise, click *Finish*.
9. The Project Manager displays a message box to confirm the successful creation of a library project. Click *OK*.
10. The Project Manager main window appears showing the Library Management project flow. The Flows menu shows the Library Management item checked. You can click the other items to switch to a different flow.

Allegro Project Manager User Guide

Creating Projects

Project Manager displays the *library management flow* as displayed in the following figure .



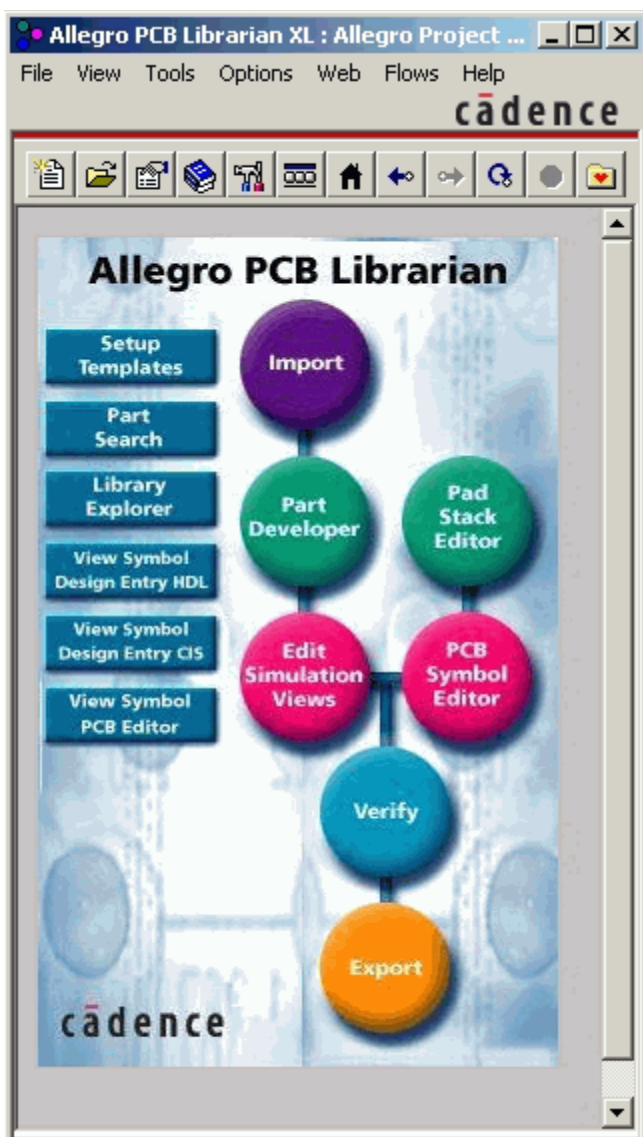
Managing Library Projects

This section discusses the following topics:

- Library Management Flow
- Verification Subflow

Library Management Flow

The interface for library management is as shown in the following image:



The various icons on this screen are briefly described:

Setup Templates

Helps you create a new template, edit an existing template, or extract a template from a part. When you click this button, the Setup Template Dialog Box appears.

Allegro Project Manager User Guide

Managing Library Projects

Library Explorer	Opens the Library Explorer window.
Part Search	Opens the Cadence Web page that provides information about the Part Browser utility and lets you freely download the utility.
View Design Entry HDL Symbol	Opens the <u>View Symbol Dialog Box</u> in which you can specify a symbol that you want to view and then view it in Design Entry HDL.
View Orcad Capture Symbol	Launches the OrCad Capture tool.
View PCB Editor Symbol	Opens the <u>Footprint Dialog Box</u> where you can specify a footprint and view it in PCB Editor.
Import	<p>Opens a submenu with the following three options:</p> <p>XML to Design Entry HDL: Opens the <u>Import from XML to Design Entry HDL Dialog Box</u> where you can specify an XML file to import it into a Design Entry HDL library.</p> <p>XML to Capture (available only on the Windows platform): Opens a browser where you can specify an XML file and convert it to a Capture part using the <u>Select Symbols Dialog Box</u>.</p> <p>Capture to Design Entry HDL (available only on the Windows platform): Opens the <u>Import from Capture to Design Entry HDL Dialog Box</u> where you can specify a Capture part from a .olb library file and import it into a Design Entry HDL library.</p>
Part Developer	Opens Part Developer, which helps you create, edit, and verify part data. It gives you one interface in which you can create and edit simulation views, schematic symbols, physical pin data, and part table data.
Edit Simulation Views	Opens the <u>VHDL/Verilog Support Dialog Box</u> where you can specify an existing or new VHDL or Verilog wrapper or mapfile for a part to be opened in Part Developer for modification.
Pad Stack Editor	Launches the Pad Stack Designer, which lets you create and edit padstacks and save them to your library.

Allegro Project Manager User Guide

Managing Library Projects

PCB Editor Symbol Editor Launches PCB Editor.

Verification Opens the Verification Subflow.

Export Opens a submenu with the following three options:

Design Entry HDL to XML: Opens the Export from Design Entry HDL to XML Dialog Box using which you can export a Design Entry HDL part to an XML file.

Capture to XML (available only on the Windows platform): Opens the Convert from Capture to XML Dialog Box in which you can specify a Capture part to be converted to XML file.

Design Entry HDL to Capture (available only on the Windows platform): Opens the Export from Design Entry HDL to Capture Dialog Box where you go through a series of steps to export a Design Entry HDL cell to Capture.

Setting Up a Template

The Setup Template Dialog Box offers three options:

Create a new template

1. Select this radio button to specify that you want to create a new `.tpl` template.
2. Click *OK*.
3. The *New Template* dialog box appears. You enter property, pin and symbol setup specifications in this dialog box. Click *Save*.
4. The *Save Part Template* browser appears. Type a new name for the template and click the *Save* button.
5. A message box appears asking you if you want to apply the template in current settings. Click *Yes* or *No*.

Edit an existing template

1. Select this radio button to specify that you want to edit a `.tpl` template.
2. Click *OK*.
3. The *Edit Template* dialog box appears. You modify property, pin and symbol setup specifications in this dialog box and click *Save*.
4. The *Save Part Template* browser appears. Modify the name of the template and click the *Save* button.
5. A message box appears asking you if you want to apply the template in current settings. Click *Yes* or *No*.

Extract template from an existing part

1. Select this radio button to specify that you want to extract a `.tpl` template from an existing part.
2. The *Library* and *Cell* drop-down lists appear enabled. Specify the Design Entry HDL library and a cell in it to extract the template from.
3. Click *OK*.
4. The *New Part Template* browser appears. Type a new name for the template and click the *Save* button.
5. The *Extracted Template Values* box appears showing you the values you specified for the new template. Click *OK*.

Viewing a Symbol

To view a symbol, do the following in the View Symbol Dialog Box:

1. Select either one of the radio buttons *Build Libraries* or *Reference Libraries* to specify the location of the symbol. You may notice that only one set of libraries, project libraries, appears for design projects.
2. Select a Design Entry HDL library from the *Library* drop-down list that displays the list of available build or reference libraries in the `cds.lib` or `refcds.lib` file.
3. Select a cell from the *Cell* drop-down list that displays the parts contained in the selected library.
4. Select a view from the *View* drop-down list that displays the available symbols in the selected part.
5. Click *View*. Design Entry HDL opens to display the symbol.

Viewing and Verifying a Footprint

To open a footprint, do the following in the Footprint Dialog Box:

1. Select a footprint from the *Footprint* list.
2. Click *View* or *Verify* to open PCB Editor and view the footprint there.
3. Click PCB Editor Setup to open the *User Preferences Editor* window where you can set or modify the `PSMPATH` directive. To open PCB Editor and view a footprint in it, you need to specify the `PSMPATH` directive in the project (`.cpm`) file.

Importing from XML to Design Entry HDL

To import an XML file to Design Entry HDL, do the following Import from XML to Design Entry HDL Dialog Box:

1. Specify the absolute path of the XML file in the *File* text box by typing the absolute path to the file or by selecting it using the browse button.
2. In the *Options* group box, select one of the three radio buttons to indicate whether you want to import only the master component and not the aliases, each alias as an individual primitive, or each alias as an individual cell.
3. In the *Design EntryHDL* group box, select a library from the *Library* drop-down box.
4. Click *OK*. The XML file you specified will be imported into the library you selected.

See [XML to Design Entry HDL Conversion](#) for more information on conversion from XML to Design Entry HDL.

Converting XML to Capture

To specify the XML file that you want to export to Capture, do the following:

1. Select an XML file from the browser.
2. Click *Open*. The [Select Symbols Dialog Box](#) opens where you need to specify symbols and then convert the XML file to a Capture part.

Importing from Capture to Design Entry HDL

To import a part from Capture to Design Entry HDL, do the following in the [Import from Capture to Design Entry HDL Dialog Box](#):

1. Specify a library in the *Library* text box by typing the absolute path to the file or by selecting it using the browse button. The *Part* drop-down box is populated with names of parts in the selected library.
2. Click a part to select it. The *Aliases* display box is populated with all aliases of the selected part.
3. In the *Options* group box, select one of the three radio buttons to indicate whether you want to import only the master component and not the aliases, each alias as an individual primitive, or each alias as an individual cell.
4. In the *Design EntryHDL* group box, select a library from the *Library* drop-down box.
5. Click *OK*. The Capture part you specified will be imported into the library you selected.

Verifying VHDL/Verilog Support

To view an existing or new VHDL or Verilog wrapper or mapfile for a part, do the following in the [VHDL/Verilog Support Dialog Box](#):

1. Select a Design Entry HDL Library in the *Library* drop-down list box, which lists the build libraries in the `cds.lib` file of the project.
2. Select a cell in the *Cell* drop-down list box, which displays the parts available in the selected Design Entry HDL library.

3. Select either the *Verilog Wrapper/Map File* radio button or the *VHDL Wrapper/Map File* radio button.
4. Select either one of the indented radio buttons to indicate if you want to create a new mapfile or wrapper or modify an existing one. If you select *Open existing*, select a file or wrapper from the adjacent list box.
5. Click *OK*. If you had selected the *Create new* radio button, Part Developer opens and shows the wrapper or file.

Exporting from Design Entry HDL to XML

To export a Design Entry HDL part to an XML file, do the following in the Export from Design Entry HDL to XML Dialog Box:

1. Select a library from the *Library* drop-down box, which lists the libraries in the `cds.lib` file of the project.
2. Select a cell you want to save as an XML file from the *Cell* drop-down box, which displays the cells available in the selected Design Entry HDL library.
3. Click *OK*. The Export from Design Entry HDL to XML - Select Package Dialog Box opens.

Exporting from Design Entry HDL to XML - Select Package

To select a package, do the following in the Export from Design Entry HDL to XML - Select Package Dialog Box:

1. The *Package* drop-down list is populated with packages in the cell you selected. Select a package. All the symbols in the package are automatically selected.
2. In the *XML* group box, specify the directory into which you want to export the selected part by typing the absolute path to the directory or by selecting it using the browse button.
3. Click *OK*. The specified cell is exported as an XML file into the directory you specified. The name of the XML file is derived from the name of the package you selected.

Converting from Capture to XML

To import a part from Capture to XML, do the following in the Convert from Capture to XML Dialog Box:

1. Specify a library in the *Library* text box by typing the absolute path to the file or by selecting it using the browse button. The *Part* drop-down box is populated with names of parts in the selected library.
2. Click a part to select it. The *Aliases* display box is populated with all aliases of the selected part.
3. In the *XML* group box, specify the directory into which you want to export the selected part by typing the absolute path to the directory or by selecting it using the browse button.
4. Click *OK*. The specified part is exported as an XML file into the directory you specified. The name of the XML file is derived from the name of the part.

See [Capture to XML Conversion](#) for more information on conversion from XML to Design Entry HDL.

Exporting from Design Entry HDL to Capture

To specify the Design Entry HDL cell that you want to export to Capture, do the following in the [Export from Design Entry HDL to Capture Dialog Box](#):

1. Select a library from the *Library* drop-down box, which lists the libraries in the `cds.lib` file of the project.
2. Select a cell you want to save as a Capture part from the *Cell* drop-down box, which displays the cells available in the selected Design Entry HDL library.
3. Click *OK*. The [Select Symbols Dialog Box](#) opens.

Selecting Symbols

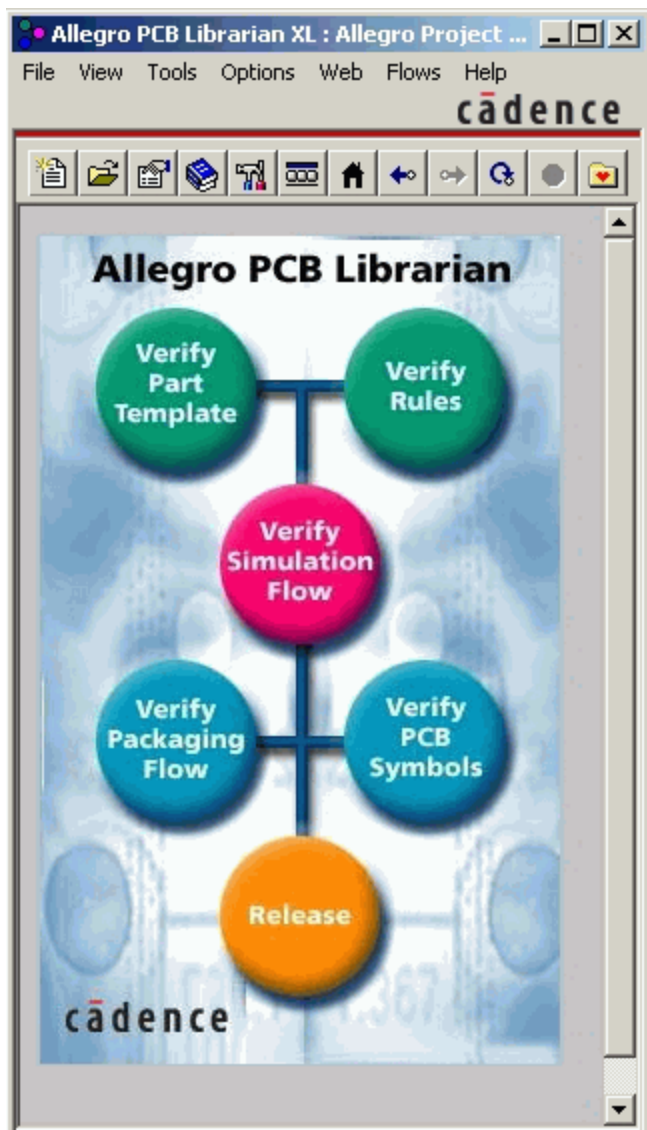
To select symbols, do the following in the [Select Symbols Dialog Box](#):

1. The *Package* drop-down list is populated with packages in the cell you selected. Select a package.
2. The *Symbols* box displays groups of symbols for the selected package. Select symbols by clicking the boxes to their left. You can select symbols from one of two groups at a time.
3. If you want pin names rather than pin text to be used for the selected cell in Capture, check the *Use Pin Name to write Capture Port Name* check box. If you do not check this check box, Capture will use pin text for each alias.
4. Specify the Capture library into which you want to export the selected cell in the *Library* text box by typing the absolute path to the file or by selecting it using the browse button.

5. Click *OK*. The specified cell is exported into the Capture library you specified.

Verification Subflow

The interface for the verification subflow is as shown in the following image:



The various icons on this screen are briefly described:

Verify Part Template

Opens the Verify Design Entry HDL Libraries with Template Dialog Box where you can specify a part and a template against which to verify it.

Allegro Project Manager User Guide

Managing Library Projects

Verify Rules Checker Rules	Opens the <u>Verify Design Entry HDL Libraries Using Rules Checker Dialog Box</u> to help you verify Design Entry HDL libraries or parts using Rules Checker.
Verify Simulation Flow	Opens the <u>Verify Design Entry HDL Libraries in Verilog Simulation Flow Dialog Box</u> to help you verify Design Entry HDL libraries or parts using the hlibsimsim utility.
Verify Packaging Flow	Opens the <u>Verify Design Entry HDL Libraries in Packaging Flow Dialog Box</u> to help you verify Design Entry HDL libraries or parts using the hlibftb utility.
Verify PCB Editor Symbols	Opens the <u>Footprint Dialog Box</u> where you can specify a footprint and verify it in PCB Editor.
Release	Takes you back to the upper-level flow, <u>Library Management Flow</u> , from which you started this subflow.

Verifying Design Entry HDL Libraries with Template

To verify Design Entry HDL libraries or parts with a template, do the following in the Verify Design Entry HDL Libraries with Template Dialog Box:

1. Select either of the radio buttons *Build Libraries* or *Reference Libraries* to indicate whether you want to run the verification on the build area libraries or the reference area libraries.

Note: For design projects, such an option will not be there because each project has only a single set of libraries, the project libraries.
2. From the library tree structure, select a cell on which to run the verification by clicking in the empty box to its left.
3. In the *Options* group box, select check boxes to indicate whether you want to verify symbol information, pin loads, or property values. You need to select at least one of these options.
4. Specify a template in the *Select a Template* field, by typing its name or by selecting it by using the browse button next to this field.
5. Click *OK*. The *Part Verification with Template* box appears showing a report of checks done and verification results obtained.
6. Click the *Save As* button to save the report as a .rep file.

Verifying Design Entry HDL Libraries Using Rules Checker

To verify Design Entry HDL libraries or parts using Rules Checker, do the following in the Verify Design Entry HDL Libraries Using Rules Checker Dialog Box:

1. Select either of the radio buttons *Build Libraries* or *Reference Libraries* to indicate whether you want to run the verification on the build area libraries or the reference area libraries.
2. From the library tree structure, select a cell or a library on which to run the verification by clicking in the empty box to its left.
3. In the *View Verification* group box, select appropriate options under *Minimal Checks* to run pre-supplied Rules Checker checks. Otherwise, select *Advanced Checks* and click the *Launch* Rules Checker button below it to start Rules Checker.
4. If you selected *Minimal Checks*, you need to select one or more of the following:
 - a. Select the *Symbol origin is centered* check box to check whether the origin always lies within the symbol and whether the symbol outline is at a distance less than the maximum allowed offset from the origin.
 - b. Select the *Tristated pins have Input and Output loads defined* check box to check for the presence of pin properties `OUTPUT_LOAD` and `INPUT_LOAD` for every tristate pin. The presence of tristated pins is denoted by the property `OUTPUT_TYPE =TS, TS`.
 - c. Select the *Consistent symbol name in symbol and package file* check box to check whether the cell name is the same as `BODY_NAME` in the `chips.prt` file.
 - d. Select the *Mandatory properties present in package file* check box to check whether the properties `BODY_NAME`, `PART_NAME`, `CLASS`, and `JEDEC TYPE` are present in the packages.
 - e. Select the *Consistent symbol and package in pin list* check box to check whether the pins are consistent across symbol and package views.
5. Click *OK*. The relevant checks are done and a verification report displayed.
6. Click the *Save As* button to save the report as a `.rep` file.

Verifying Design Entry HDL Libraries in Verilog Simulation Flow

To verify Design Entry HDL libraries or parts using the hlibsим utility, do the following in the Verify Design Entry HDL Libraries in Verilog Simulation Flow Dialog Box:

1. Select either of the radio buttons *Build Libraries* or *Reference Libraries* to indicate whether you want to run the verification on the build area libraries or the reference area libraries.
2. From the library tree structure, select one or more cells from one library or a complete library on which to run the verification by clicking in the empty boxes to their left.
3. Select the *Wrappers* or *Mapfiles* radio button.
4. Type the names of one or more wrapper or mapfile views in the *Wrapperview(s)/Mapview(s)* text box.
5. If you want the verification to stop right after netlisting, select the *Stop On Netlist* check box.
6. In the *Options* group box, select one of the three radio buttons.
 - a. If the file that contains the paths to Verilog Models and user-defined primitives (UDPs) is named `vlog_model_path.txt` and is in the default path, that is in the library directory, select the *Default paths* radio button.
 - b. If you want to specify the absolute paths to the Verilog Models and user-defined primitives (UDPs), select the *Specify paths* radio button. Then, specify the directory where the Verilog Models and UDPs for the selected library exist. To do this, type the paths in the Models and Udpes text boxes or specify the directory by using the browse buttons provided next to them.
 - c. If you have created an options file, which is a verilogcmd file that can be passed to the simulator as it is without any further processing, select the *Path to Options file* radio button. Then, type the path to this file in the text box or specify the path by using the browse button provided next to it.
7. Click *OK*. This calls the hlibsим utility, which runs the verification process. You can see the verification details by clicking the *Details* button in the *Verifying* process progress box.
8. A success or error message box appears and a log file is generated. You can view this file by clicking the *View Log File* button.

Verifying Design Entry HDL Libraries in Packaging Flow

To verify Design Entry HDL libraries or parts using the hlibftb utility, do the following in the Verify Design Entry HDL Libraries in Packaging Flow Dialog Box:

1. Select either of the radio buttons *Build Libraries* or *Reference Libraries* to indicate whether you want to run the verification on the build area libraries or the reference area libraries.
2. From the library tree structure, select one or more cells from one library or a complete library on which to run the verification by clicking the empty boxes to their left.
3. Select the *Use project ptf files for verification* radio button if you want to specify that the project part table files be used in instantiation and packaging. If no part table files are specified in the project file, the cell-level ptf is used by default.
4. Select the *Upto PCB Editor board (netrev)* radio button if you want the hlibftb utility to verify the cell or library for the complete Front-to-Back flow.
5. Select the *Generate pass/fail report* radio button if you want Part Developer to verify each part separately and you want a separate report for each part. A message box appears with a warning that the verification has to be run on each part separately and that this may take a long time. Click *OK*.
6. Click *OK*. This calls the hlibftb utility, which runs the verification process. You can see the verification details by clicking the *Details* button in the *Verifying* process progress box.
7. A success or error message box appears and a log file is generated. You can view this file by clicking the *View Log File* button.

Or

If you had selected the *Generate pass/fail report* radio button, a *Verification Results* dialog box appears and log files are generated. You can view a log file by clicking each part and then clicking the *View Log File* button.

XML to Design Entry HDL Conversion

The translation of XML to Design Entry involves the creation of the chips view and the symbols view from the XML part. The XML format exported out is in the schema of the E-Tools DTD. For more information on E-Tools DTD, refer to the E-Tools web site.

The translation supports normal flat parts, homogeneous parts (single cell in XML), and heterogeneous parts (multiple cells in XML having the same Source Package). For more information on types of Capture parts, refer to Cadence documentation on Capture.

Creating the Chips View (Physical)

The following information is translated from the Design Entry part:

- **Pin Names**

The pin name on the Capture symbol pin is translated into XML as the pin name.

- **Package Properties**

The properties on the cell in XML are placed as the Physical properties in the chips view.

- **Pin Types**

Capture Pin Types	XML Direction	XML Types
INPUT	Input	Input
Bidirectional	Bidirectional	UNSPEC
Output	Output	Output
Open Collector	Output	OC
Passive	UNSPEC	Passive
3-State	Output	HIZ
POWER	Unspecified	POWER
Open Emitter	Output	OE

- **Part Aliases (equivalent packages)**

In XML, the aliases are available as a series of values of the property PACKAGE_ALIAS(n), where n is the sequence number. These are converted as equivalent packages on the primitive section in the chips view.

■ Pin Text

If a property PIN_TEXT exists in the XML, it is taken as the PIN_TEXT property value. Otherwise, this is derived from the pin name.

Note: If the PIN_TEXT property does not exist, you might need to adjust the location of PIN_TEXT on the symbol after the symbol is created. For parts downloaded from spincircuit, the PIN_TEXT property does not exist.

Translating Symbol Information

■ Pin Properties

All pin properties are translated as they are. The XML schematic symbol block contains the references to the properties in the cluster ports. The properties on the cluster ports belong on the physical sections if and only if they are not referenced by a symbol port.

■ Symbol Properties

All pin properties in the schematic symbol section are translated as they are.

Limitations

The XML output applies the default Design Entry font information for all texts. The following graphics entities are not translated:

- Filled shapes (only outlines are translated)
- Bitmaps
- Font and color information

Capture to XML Conversion

The translation of Capture to XML includes the conversion of physical pin information and symbol information of Capture parts to XML parts. The XML format exported out is in the schema of the E-Tools DTD. For more information on E-Tools DTD, refer to the E-Tools web site.

The translation supports normal flat parts, homogeneous parts (single cell in XML), and heterogeneous parts (multiple cells in XML having the same Source Package). For more information on types of Capture parts, refer to Cadence documentation on Capture.

Translating Physical Information

The following information is translated from the Capture part:

- **Pin Names**

The pin name on the Capture symbol pin is translated into XML as the pin name.

- **Package Properties**

The properties on the cell in XML are placed as the Physical properties in the chips view.

- **Pin Types**

Capture Pin Types	XML Direction	XML Types
INPUT	Input	Input
Bidirectional	Bidirectional	UNSPEC
Output	Output	Output
Open Collector	Output	OC
Passive	UNSPEC	Passive
3-State	Output	HIZ
POWER	Unspecified	POWER
Open Emitter	Output	OE

- **Part Aliases (equivalent packages)**

The aliases are added as a series of values of the property PACKAGE_ALIAS(n), where n is the sequence number.

Translating Symbol Information

■ **Pin Properties**

All pin properties are translated as they are. The XML schematic symbol block contains the references to the properties in the cluster ports. The properties on the cluster ports belong on the physical sections if and only if they are not referenced by a symbol port.

■ **Symbol Properties**

All pin properties in the schematic symbol section are translated as they are and placed on the schematic block.

Limitations

The XML output applies the default Capture font information for all texts. The following graphics entities are not translated:

- Filled shapes (only outlines are translated)
- Bitmaps
- Font and color information

Allegro Project Manager User Guide
Managing Library Projects

Project Manager Procedures

This chapter covers the following topics:

- [Working with Projects](#)
- [Starting Tools from Project Manager](#)
- [Customizing Project Manager](#)
- [Starting Project Manager from the Command Line](#)

Working with Projects

This section covers the following project-related procedures:

- [Opening a Project](#)
- [Copying a Project](#)
- [Importing a Project](#)
- [Exporting a Project](#)
- [Closing a Project](#)
- [Importing IFF Designs](#)

Opening a Project

1. Choose *File – Open*. The *Open* dialog box appears.
2. In the *Files of type* list, click *Project Files (*.cpm)*.
3. In the *Look in* list, select the directory that contains the project.
4. In the list of folders below the *Look in* box, double-click the project folder to display its contents. The *Look in* box displays this folder.

5. Select the project file (`projectname.cpm`).

6. Click *Open*.

Your project flow is displayed in Project Manager.

Project Manager decides what kind of project it is depending on whether the `refcds.lib` file exists for that project or not. If it does exist, the project is considered to be a library project. Otherwise, it is be considered as a design project. The information about the flow last opened will be taken from the `.cpm` file and that flow will be shown. If this information is missing, the Library Management Flow is shown for library projects and the Board Design flow for design projects.

To view your project settings, choose *View – Project Settings*.

Note: You can also open a project by selecting the project file (`projectname.cpm`) in the File Manager and dragging it to the Project Manager window.

Copying a Project

When an existing project is copied to another location to create a copy of the source project with a different library and root design name, integrity and preservation of data, especially the packaged data, becomes a cause of concern. The same is also true if the root design or design library are renamed for an existing project. Design Entry HDL ensures that when you copy a project from one location to the other, the entire project hierarchy is copied from the source to the destination, thereby ensuring that there is no loss or corruption of data. Also, if a design library or a root design is renamed, Design Entry HDL ensures that the copied hierarchy is updated with respect to the root design cell and design library.

Classifying Data for Copying into a New Project

For the Copy Project functionality, project data is classified into the following categories:

- Essential Data
- Derived Data

Allegro Project Manager User Guide

Project Manager Procedures

Essential Data

Essential data is crucial for the design and is always copied to the new project. The project data in this category include:

Essential Data	Description
page*.csa files	These are the design database (ASCII) files that Design Entry HDL creates when you write a drawing.
cds.lib	The cds.lib file contains the list of libraries to be used in conjunction with the project specified in the project file.
.cpm file	Each project has one project file. The <projectname>.cpm file contains all the setup information, such as design name, design library, other libraries used, the name of the temporary directory.
dcf data	The dictionary and constraints (.dcf) file is a snapshot of electrical constraint information. It may include any user-defined properties, ECsets and their constraints, and net-related objects and their constraints (including ECset references). The dictionary and constraints file is proprietary to Cadence Design Systems and, as such, is not available for editing.
PTF data	PTF data is stored in a part table file (.ptf), which associates a logical part with physical parts having varying physical properties. Each row in a part table corresponds to a physical part.
Variant data	Variant data is stored in a variant database (variant.dat). It is an ASCII file, which contains the list of variant value entries for each and every variant component in a design. Since the schematic can also contain variant data, it is important to sync up the variant data between the variant database and the schematic through the packaged files
packaged data	Packaged data consists of pst*.dat files (pstxprt.dat and pstxref.dat). The pstxprt.dat file lists each reference designator and the section assigned to it. This information is also available in a simplified format in the pstxref.dat file. The pstxref.dat file also contains the logical-to-physical bindings, the schematic instance, and the component information used by Packager-XL for each part in the design.

Allegro Project Manager User Guide

Project Manager Procedures

Essential Data

Description

`module_order.dat` file Module ordering saves all ordering and the exclusion or inclusion information in a module order file named `module_order.dat` in the `<root design>/sch_1` directory. The `module_order.dat` file is read during cross referencing and hierarchical plotting.

Any local menu customizations

For more information on essential data contained in these files, see the following sections:

- [Design Entry HDL Files](#) of *Allegro Design Entry HDL Reference Guide*.
- [Project Creation and Setup](#) of *Allegro Design Entry HDL User Guide*.

Derived Data

Derived data is regenerated from the essential data. Derived data includes:

- `page*.csb`, `pc.db` files - `page*.csb` files contain the same information as the corresponding ASCII file but in a proprietary binary format that is quicker for Design Entry HDL to read and save. `pc.db` file contains information about the blocks used in the design.
- Packaged data
- Marker files
- BOM outputs - BOM-HDL reports are generated with inputs from project file, netlist files, the BOM template file, the variant database, and the callout file.
- CRefer Data - The `cref.dat` contains cross-references for page borders, custom offpage I/O flag bodies, and power signals.

Use Model

Using the Copy Project functionality you can copy an existing project to a new location or copy a project to the same location with different design and library names. Copy Project functionality can be used in the following two ways:

- [Copying a Project from Command Line](#)
- [Using the Copy Project GUI](#)

Copying a Project from Command Line

You use the `copyproject` command to copy a project from command line.

```
copyproject -proj "<project_cpm_file>" -copytopath "<location_to_copy_to>" -  
newprojname "<name_of_new_cpm_file>" -newlib "<new_library_name>" -newdesign  
"<new_design_name>" -product "<product_license>"
```

Using the Copy Project GUI

The Copy Project GUI provides a wrapper over the `copyproject` command. The GUI is an easy to use interface to copy a project. Use the GUI for:

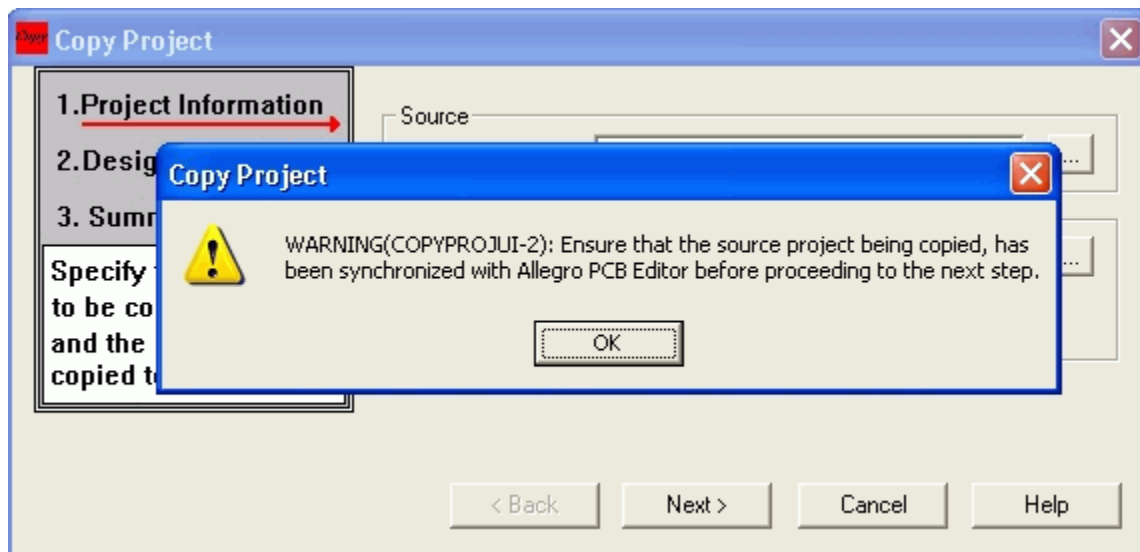
- Copying an Existing Project to a New Location
- Renaming an Existing Project

Copying an Existing Project to a New Location

1. Launch Project Manager.
2. Choose *File – Copy Project*.

The Copy Project dialog box is displayed.

3. Specify the location of the existing source project `.cpm` file in the *Project Name* field.
A message prompts you to ensure that the board and the schematic are in synch.



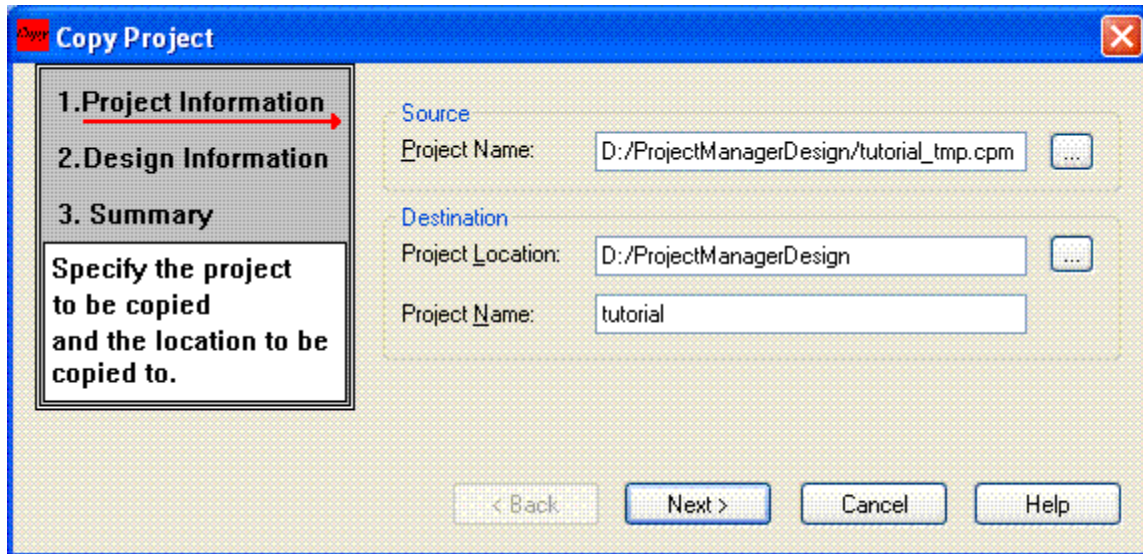
4. Specify the location of the destination project to which the project needs to be copied in the *Project Location* field.

Allegro Project Manager User Guide

Project Manager Procedures

By default, the location of the destination project is the same as the location of the source project. If you specify the name of a non-existent folder, it'll be automatically created for you.

5. Specify a name for the destination project in the Project Name field of *Destination* section.



If you are copying the project to a new location, project name can be the same as the existing project. However, if you are copying the project to the existing project hierarchy, the destination project file name must be different from the one for existing project.

6. Click *Next*.
7. Specify the design library name for the destination project in the *Design Library* field.
By default, the design library is the same as the source project design library.
8. Specify the design name for the destination project in the *Design Name* field.

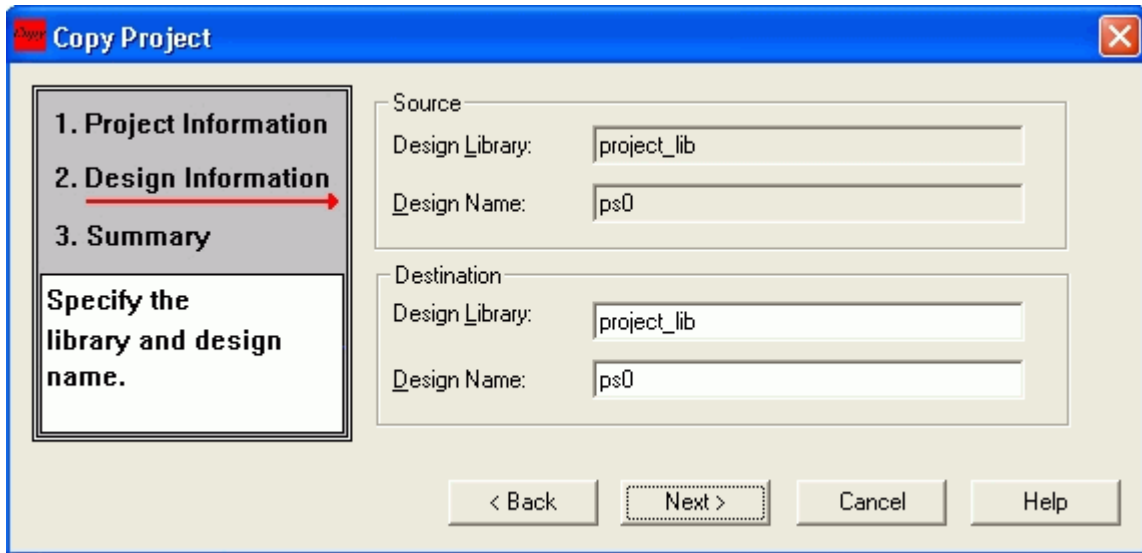
If you are copying the project to the existing project hierarchy, the destination root design name must be different from the source root design name. Only if the source and destination project location are different can the names be the same.

Note: If you specify a new name for the design being copied, make sure that there are no embedded spaces in the design name as spaces are not supported in the copied design names. Do not specify a new design name which is the same as the name of any of the blocks existing in a library other than the design library. This would result in a copied project having two different cells with the same name from two different libraries

Allegro Project Manager User Guide

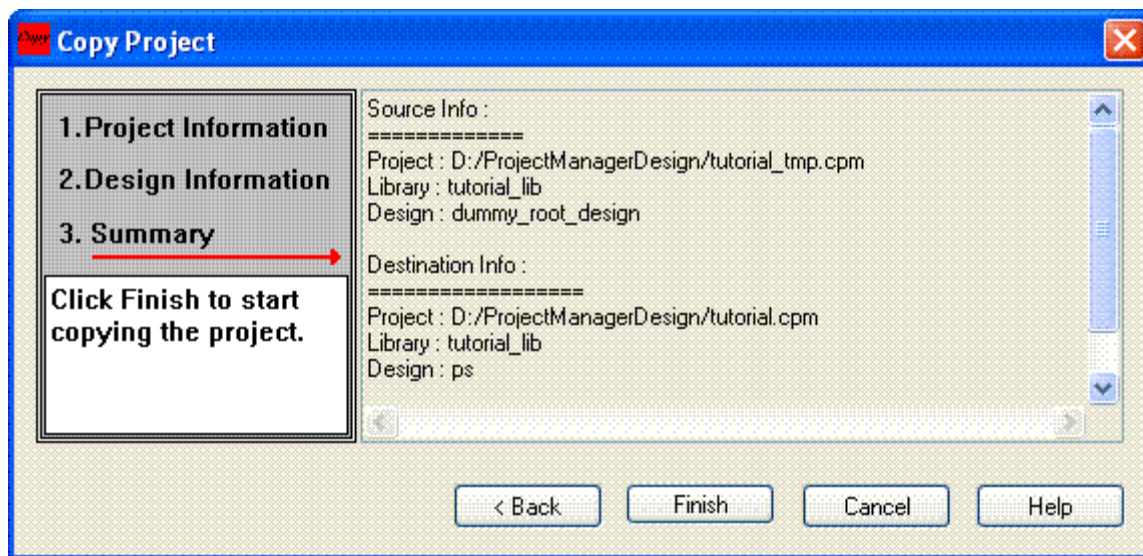
Project Manager Procedures

and the copied project would not work fine.



The 'Copy Project' dialog box is shown at the 'Design Information' step. On the left, a sidebar lists three steps: '1. Project Information', '2. Design Information' (which is highlighted with a red arrow), and '3. Summary'. Below the sidebar, a box contains the text 'Specify the library and design name.' The main area of the dialog has two sections: 'Source' and 'Destination'. Each section contains two text input fields: 'Design Library' and 'Design Name'. In the 'Source' section, 'Design Library' is set to 'project_lib' and 'Design Name' is set to 'ps0'. The 'Destination' section has identical values. At the bottom of the dialog are four buttons: '< Back', 'Next >', 'Cancel', and 'Help'. The 'Next >' button is highlighted with a dashed border.

9. Click *Next*.



The 'Copy Project' dialog box is shown at the 'Summary' step. The sidebar on the left now highlights '3. Summary' with a red arrow. Below the sidebar, a box contains the text 'Click Finish to start copying the project.' The main area of the dialog displays 'Source Info' and 'Destination Info' in a text area. The 'Source Info' section lists: Project : D:/ProjectManagerDesign/tutorial_tmp.cpm, Library : tutorial_lib, and Design : dummy_root_design. The 'Destination Info' section lists: Project : D:/ProjectManagerDesign/tutorial.cpm, Library : tutorial_lib, and Design : ps. At the bottom of the dialog are four buttons: '< Back', 'Finish', 'Cancel', and 'Help'. The 'Finish' button is highlighted with a dashed border.

The Summary page summarizes the information about the source and destinations designs.

10. Click *Finish* to start copying the project.

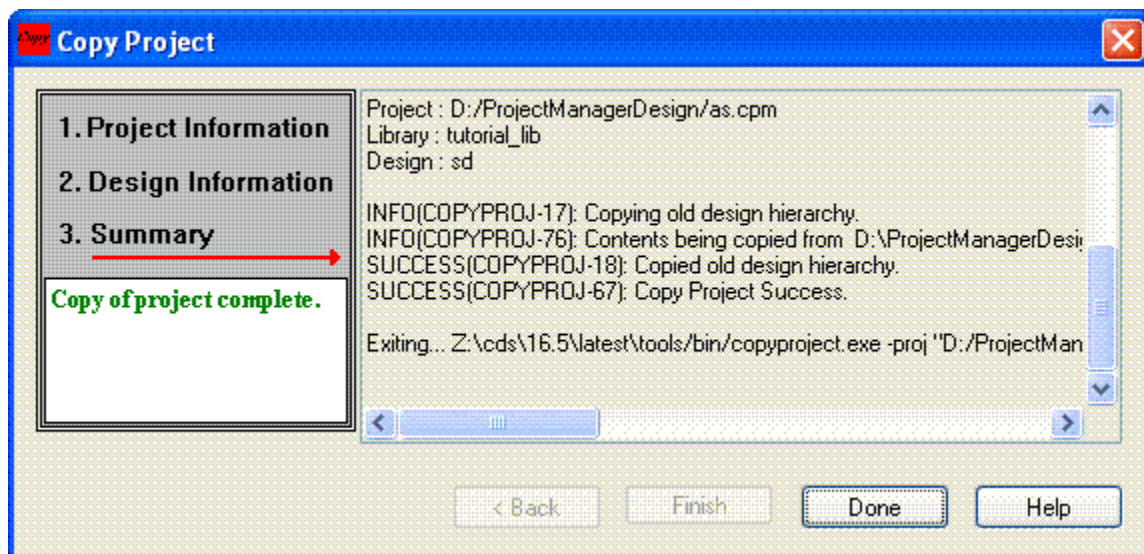
A few informational and warning messages are displayed prompting you to perform a set of activities before your copied design becomes fully functional.

Important

The following error is reported if the existent paths for accessing a library in `cds.lib` do not remain valid and referable in the copied destination, and the Copy Project UI exits.

"Unable to initialize the project with the contents of the `cds.lib` file. Ensure that the path to the libraries in the `cds.lib` file in the copied location is valid and referable. You can achieve this by using the Archiver tool on the source project to refer to the libraries locally. You can also edit the `cds.lib` file manually with paths relative to the copied location before running the Copy Project solution."

The final screen displays successful completion message.



A copy of the project is created at the specified location.

11. In Project Manager, choose *File – Open*.
12. Open the newly created *project.cpm* file.
13. Click *Setup*.
14. In the Project Setup dialog box, ensure that the relative path entries in the new project are appropriate.
15. Launch Design Entry HDL and open the newly created project.
16. Choose *File – Save Hierarchy* or type *hier_write* in the console window and press **ENTER**.

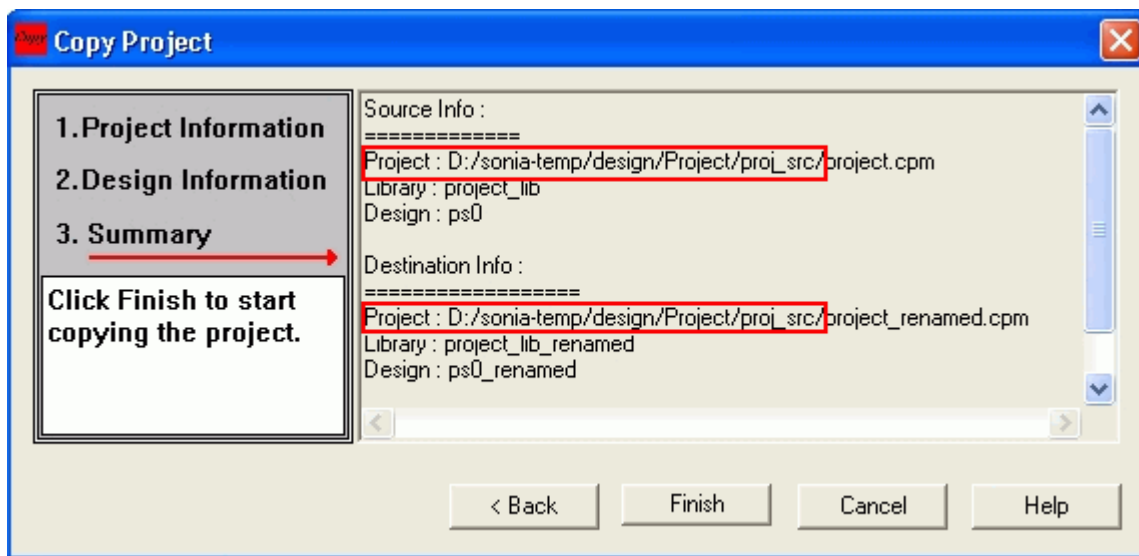
Allegro Project Manager User Guide

Project Manager Procedures

The hierarchy is saved and all files in `sch_1` view are created.

Renaming an Existing Project

You can also copy an existing project to the same location with different design and library names. This way you can rename an existing project. The steps for renaming a project are similar to copying a project. However, when you rename a project, the location of the source project remains the same as the location of the destination project.



A copy of the `.cpm` file is saved with the new name, and a copy of the `worklib` folder is saved with the design library name that you specified. A design with the new name is also renamed as per your specifications. The path to the new design library name is added in the `cds.lib` file.

Name	Size	Type	Date Modified
adw		File Folder	6/9/2005 12:
lib		File Folder	4/10/2006 10
part_tables		File Folder	4/10/2006 10
project_lib_renamed		File Folder	4/10/2006 10
temp		File Folder	6/9/2005 12:
worklib		File Folder	4/10/2006 10
cds.lib	1 KB	LIB File	4/10/2006 10
project.cpm	2 KB	CPM File	6/9/2005 12:
project_renamed.cpm	2 KB	CPM File	4/10/2006 10

After copying the project, you must launch Project Manager and ensure that the path to the design library in `cds.lib` is appropriate. You also need to open the new project in Design

Allegro Project Manager User Guide

Project Manager Procedures

Entry HDL and save the hierarchy. When you are sure that the project is working fine with the new design, library, and project name, then only you should delete the earlier design, library, and project.

Using Reuse Blocks in a Design

Consider the example of a project which has a root design (`des1`) with `GEN_SUBDESIGN` directive specified in the `.cpm` file. The root design also contains a `.mdd` file (`des1.mdd`) and a board file (`des1.brd`) created in the physical view. When you copy `des1` with a new name `des2`, the physical view, which is an essential view, is also copied as is. In PCB Editor, you need to explicitly rename the `.mdd` file from `des1.mdd` to `des2.mdd` to place the module on the board.

Contents of the Copied Project

When you copy a project, all the project related files such as the `.cpm`, `cds.lib`, root design cell, and various views are copied to the new project.

File Contents of the New Project

- `project.cpm`: All occurrences of source library name are updated to destination library name. The source design name is changed to the destination design name. In case the source and destinations locations are the same, a `.cpm` file, with the name that you specified, is created at the same location.
- `cds.lib`: Source library name is updated to destination library name if you copy an existing project to a new location. However, when the source and destination locations are the same, the path to the library name is appended to the `cds.lib` file.

Root Design Cell in the New Project

The following directories and files are preserved in the new design cell hierarchy. All the other directories (views) are removed.

Directory	Files
<code>sch_1</code>	<code>master.tag</code> , <code>*.csa</code> , <code>*.xcon</code> , <code>*.dcf</code>
<code>variant</code>	<code>master.tag</code> , <code>variant.dat</code>
<code>physical</code>	All files in the <code>physical</code> folder

Allegro Project Manager User Guide

Project Manager Procedures

Directory

metadata

tbl_1

Files

master.tag, revision.dat, revision.log,
revhistory.log, pinlist.txt

master.tag, *.xcon, *.dcf

Preserved Views in New Design Cell

Views are created in a specific sequence and essential and derived data is generated as described in the following table:

Directory

cfg_package

sch_1

constraints

variant

tbl_1

physical

metadata

Files

The `cfg_package` view is the first view to be created for the new library and the new design.

When you run the *hier_write* command to save hierarchy, the following files are generated in the `sch_1` view:

- `pc.db`
- `module_order.dat`
- `page.map`
- `*.csv`
- `viewprps.prp`

The `*.dcf` file is updated with all the constraints.

Updating variants involves updating of the `variant.dat` file.

Note: No other views in the older design which are created due to backannotation on to variant schematics are copied on to the new design location.

The `*.con` file is updated with the new library and design names.

No updates are required for the `physical` folder in the new project.

No updates are required for `metadata` in the new project.

Allegro Project Manager User Guide

Project Manager Procedures

Directory

packaged

Files

`pxl.state` file is preserved with changes in library and design names.

Note: The `sym_*` view of the root design being copied is not an essential view. You need to generate the symbol in the copied project, using the *Tools – Generate View* command of Design Entry HDL for the top-level design.

Points to Remember:

- After copying the project, always save the hierarchy to create all files in `sch_1` view.
- After copying the project, if you want to annotate base or variant schematic with the variant information (stored in `variant.dat`) from the copied design, first run *hier_write* (*File – Save Hierarchy*) and then package the design by running *File – Export Physical*. Only then can Variant Editor read the variant database.
- In a copied project, the top-level design includes all the necessary changes for library rename. If you try to make changes in a lower-level block in the copied project, at times you would need to delete the `cfg_*` views in the lower-level block before proceeding as the lower-level design `cfg_*` views would still be referring to the older library name.
- You cannot rename a design from `project1` to a newly created project `project2`, when the design is not the root design. If the design name is a sub-design of another design existing in `project1`, then intention is unclear as to whether the old blocks in `project2` are to be replaced with the new block OR a new design in `project2` is to be created with the same block name as earlier.

For example, If a project has two blocks `top` and `mid` and you perform the Copy Project operation with `mid` as root design and also change the library name, then `top` is also copied as it was (with `cfg_package`) present in `worklib`. In the copied project, if you change root to `top` and run *Export Physical*, a netlisting error occurs because `cfg_package` is not modified (it shows old library name for binding). To resolve this issue, delete the `cfg_package` folder for the `top` design and generate it again by expanding the design.

Important

This type of flow is not supported for copying the project where you are copying project with `mid` as root and you also have `top` in `worklib`, which in turn has another block `low` instantiated (even if it is not instantiated). The reason being that if you copy a lower-level block, all packaging information for the block, `mid` in the context of `top` will be lost.

Note: You can also use the *File – Import* or *File – Export* command to copy a project. When you import or export a project, the project file (`projectname.cpm`) is copied.

Importing a Project

When you import a project, Project Manager imports its project file into the current project, replacing the existing project file. All the setup information for the project – libraries, view names, physical part tables, and tool setup directives – is replaced.

To import a project,

Open the project into which you want to import another project.

1. Choose *File – Import*. The *Import Project* dialog box appears.
2. In the *Files of Type* list, click *Project Files (*.cpm)*.
3. Specify the location of the project you want to import by selecting the folder in the *Folders* list and then specifying the file.
4. In the *File Name* list, select the project file (`projectname.cpm`) you want to import.
5. Click *OK*.

Exporting a Project

When you export a project, Project Manager saves a copy of the current project's project file (`projectname.cpm`) in the location you specify. You can choose to export all the settings of the project, including the setup directives specified in the Cadence `cds.cpm` file and your company's `site.cpm` file. Or you can export only the setup information specified in your `projectname.cpm` file.

To export a project,

1. Open the project you want to export.
2. Choose *File – Export*. The *Export Project* dialog box appears.
3. In the *Save File as Type* list, click *Project Files (*.cpm)*.
4. Specify the location to which you want to export the project by selecting the folder in the *Folders* list and then specifying the file.
5. In the *File Name* box, type the name followed by a `.cpm` extension for the new file.

Select the *Full Settings* box if you want to copy all the settings of the current project, including the default setup directives in the Cadence `cds.cpm` file and your company's `site.cpm` file. Clear the *Full Settings* box if you want to copy only the settings specified in your project file.

6. Click *OK*.

Closing a Project

To close a project, choose *File – Close*. Project Manager returns to the initial flow which has only three options: *Open Project* and *Create Design Project*, and *Create Library Project*. Many menu commands are disabled until you open or create a project.

Importing IFF Designs

IFF (Intermediate File Format) is used to transfer a design in a format that is machine- and application-independent between Electrical Engineering design and Printed Circuit Board (PCB) design environments.

Design Entry HDL supports the import of this file in two ways:

- Importing an IFF File as a New Project
- Importing an IFF File into an Existing Project

Importing an IFF File as a New Project

1. Start Project Manager.

Project Manager appears showing the main window in which you see icons to create library or design projects or to open existing projects.

2. Choose *File – Import IFF* in Project Manager.

The *IFF Import* dialog box appears.

3. Specify the name of the project file to be created.
4. Specify the `IFF` file to import.
5. Specify the name of the library under which components are to be created.
6. Specify if you want to create a configuration.

This is needed if you intend to package the design after saving it in Design Entry.

7. Select *Import*. After the run is complete, a pop-up specifies whether the run was successful or not. To view details, click *Details* in the progress dialog.
8. Open the created project in Project Manager.
9. Click the *Design Entry* icon to launch Design Entry HDL.
10. Choose *Tools – Run Script*.
11. Select the file `iff2hdl.scr`.

This file is located in the same directory in which you have the project file. This file writes out all the symbols and schematics that have been created as part of the project.

Note: If you notice any schematic errors after you run `iff2hdl.scr`, use the `get` command in the console command window to refresh the schematic.

12. Choose *File – Save* to save the schematic.
13. View the errors in the *Markers* window that lists all the errors in the design.

If there are no errors, your design is ready for use in Design Entry HDL.

Importing an IFF File into an Existing Project

1. Open the project in Project Manager.
2. Click the *Design Entry* icon to launch Design Entry HDL.
3. Select *File – Import IFF* in Project Manager or Design Entry HDL.

The project file name is displayed in the *Import IFF* dialog box. This cannot be modified.

4. Specify the `iff` file to import.
5. Select *Library Name*.
6. Click *Browse* adjacent to *Name*.

The *Library* list box appears.

7. Select the name of the library in which the components should be imported.
8. Select *Overwrite parts* if you want the new parts to overwrite an existing part in the library.
9. Click *Import*.
10. After the run is complete, a pop-up specifies whether the run was successful or not.

11. Click *Details* in the progress dialog box to view details of the import.
12. If you already have Design Entry HDL open, type `library <libname>` in the command window. This will read in the new parts for Design Entry HDL to access.
13. Select *Tools – Run Script*.
14. Select the file `iff2hdl.scr`.

This script writes out all the symbols and schematics that have been created as part of the project.

Note: If you notice any schematic errors after you run `iff2hdl.scr`, use the *get* command in the console command window to refresh the schematic.

Starting Tools from Project Manager

You must open a project before you can start any tool from Project Manager. Once you are in a project, you can start tools from the *Tools* menu or from the project flow. You can also start tools when Project Manager is displayed as a toolbar. The tools will be started in the context of your current project. For example, if you open `myproject` and start Design Entry HDL, Design Entry HDL will display the top-level drawing of `myproject`.

The *Tools* menu lists all the tools that are included in your project. Tools you add to your project are also listed in this menu. You can change the order in which the tools are listed.

You can also start tools from the project flow. The default project flow contains icons for Design Entry HDL, PCB Editor, Packager-XL, and Setup. (The project flow does not display your changes to the Tools menu automatically. To modify the project flow, edit the HTML file that defines the flow.)

You can start tools in one of the following ways:

- Starting a Tool from the Tools Menu
- Starting a Tool from the Project Manager Toolbar
- Starting a Tool from the Project Flows

Starting a Tool from the Tools Menu

Choose *Tools – toolname*. For example, to start Design Entry HDL, choose *Tools – Design Entry*.

The Project Manager status bar displays the message `Starting ...\toolname.exe`.

Starting a Tool from the Project Manager Toolbar

You can choose either the toolbar or the project flow as the default display mode for a project. You can also customize the toolbar. To display Project Manager as a toolbar, choose *View – Hide Flow*.

To revert to the default interface where tools are accessible from the *Tools* menu, click the Project Manager icon, that is, the last icon on the Project Manager toolbar.

To start a tool from Project Manager when it is displayed as a toolbar, click the icon of the tool you want to start.

The icons that appear on the Project Manager toolbar are as shown in the following table:

Setup		Allegro SI	
Design Entry HDL		Rules Checker	
CRRefer HDL		Sig Explorer	
PCB Editor		Project Manager	
Archiver		Hierarchy Editor	
EDIF 300		Packager Utilities	

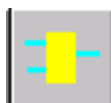
Allegro Project Manager User Guide

Project Manager Procedures

Design Sync



Synthesis



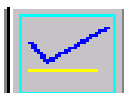
Verify Logic



Verify Synthesis



Verify Place
and Route



Place and Route



Build
Physical



Note: The Library Tools menu options are currently not available on the Project Manager toolbar.

Starting a Tool from the Project Flows

In this section, you will learn how to start a tool from the following:

- [Board Design Flow](#)
- [Library Management Flow](#)
- [Programmable IC Flow](#)

Allegro Project Manager User Guide

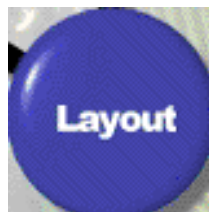
Project Manager Procedures

Board Design Flow

- In the Board Design flow, click the icon of the tool you want to start. For example, to start Design Entry HDL, click the Design Entry icon.



Design Entry HDL



PCB Editor Layout



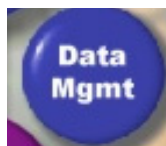
SPECTRAQuest



Design Sync



Project Setup



Design Manager





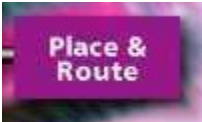

These icons can be changed. You can customize the project flow and create new icons. To modify the flow, edit the HTML file that defines the flow. The flow is not changed automatically when you add or delete any tools from Project Manager.

Allegro Project Manager User Guide

Project Manager Procedures

Programmable IC Flow

In the Programmable IC flow, click the icon of the tool you want to start. It should be noted that only Verilog simulation is supported in the Programmable Interface Flow.

ICONS	DESCRIPTION
	Launches Project Setup.
	Launches Design Entry HDL
	Allows you to import a design in Verilog or VHDL format into your project.
	Allows you to partition the design and to run Synplify on synthesizable designs and sir2edif on designs with vendor-specific primitives.
	Prompts the user to specify the mode: bottom-up or top-down. In the bottom-up mode, Place and Route takes the design from Design Entry to vendor, and invokes the vendor-specific place-and-route tools. In the top-down mode, Place and Route invokes only the vendor-specific place and route tools.
	Takes you from vendor-specific tools back to Design Entry.

Allegro Project Manager User Guide

Project Manager Procedures

ICONS

DESCRIPTION



Allows you to simulate the original Design Entry design (functional simulation).



Allows you to simulate the post-synthesis verilog.



Allows you to simulate the routed verilog, along with the timing information.

Viewing Project Settings



Project settings are the setup options you have chosen for each project. These include global settings (the selection of libraries, expansion types, view names, physical part table files, and property files) and project settings for individual tools (setup options for each tool).

Use Project Setup (*Tools – Setup*) to change global settings. Setup options for each tool have to be changed from the tool.

To view project settings,

1. Open the project.
2. Choose *View – Project Settings*.


Project Manager brings up the *Project Settings* window, which displays all the project settings in a tree form.

A  icon in front of any branch of the tree indicates there are one or more levels of hierarchy below it that are not displayed. A  icon in front of any branch of the tree indicates that the level below it is already expanded.

To collapse any branch of the hierarchy, click the  icon in front of it.

Allegro Project Manager User Guide

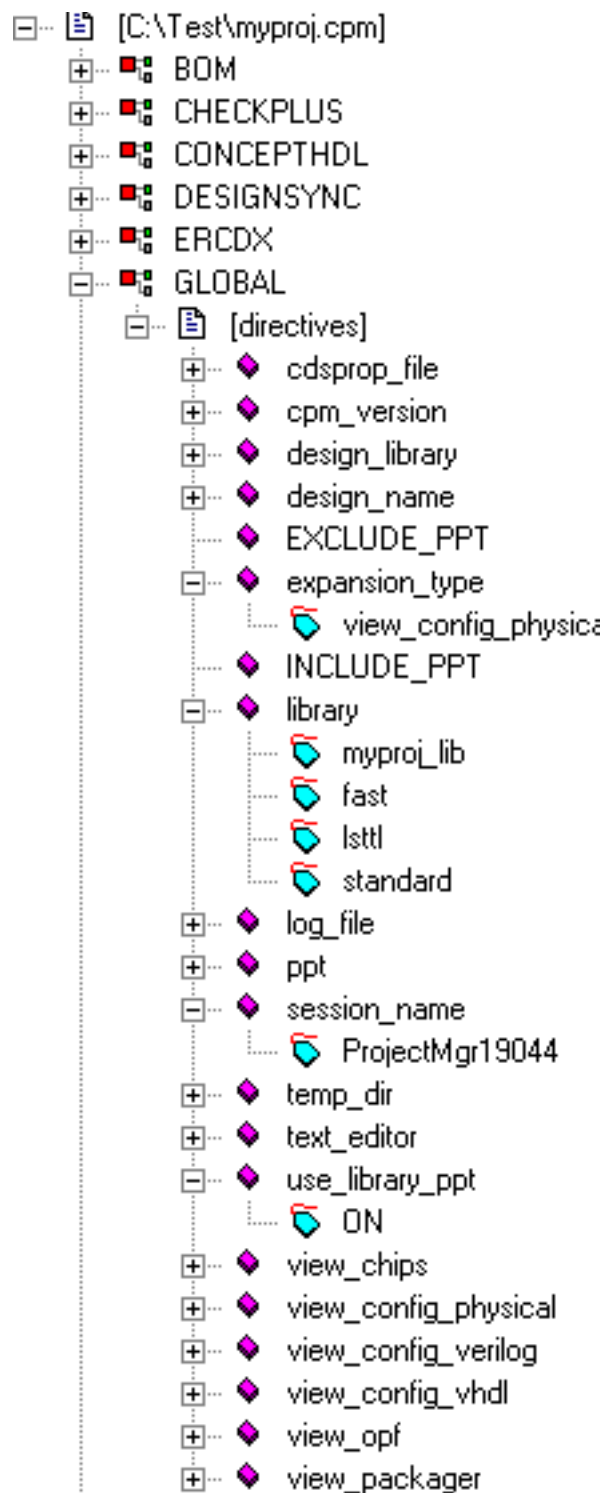
Project Manager Procedures

To display the levels below any branch of the hierarchy, click the  icon in front of it.

Allegro Project Manager User Guide

Project Manager Procedures

Project Settings Example






Viewing Project Libraries


To view project libraries for a project, do the following:

1. Open the project.
2. Choose *View – Project Libraries*.

Project Manager brings up the *Project Libraries* window, which displays all the project libraries in a tree form.

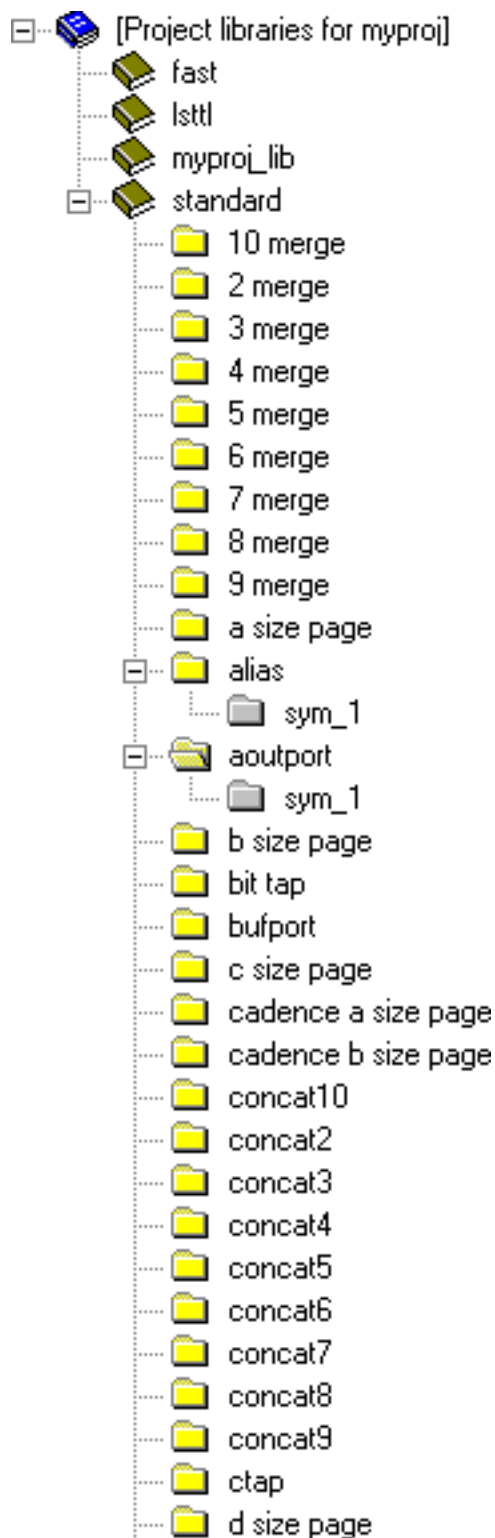
A  icon in front of any branch of the tree indicates there are one or more levels of hierarchy below it that are not displayed. A  icon in front of any branch of the tree indicates that the level below it is already expanded. A  icon denotes a text file – double-click on the icon to view the file.

To collapse any branch of the hierarchy, click the  icon in front of it.

To display the levels below any branch of the hierarchy, click the  icon in front of it.

Note: To change the libraries for the project, use Project Setup (*Tools – Setup*).

Project Libraries Example



Viewing Running Tools

To view the tools running from Project Manager, choose *View – Running Tools*.

Project Manager brings up the *Running Tools* window, which displays the tools currently running from Project Manager, as well as the process ID, user name, host name, and session name for each tool. The equivalent command-line arguments are also displayed.

Customizing Project Manager

You can customize Project Manager for each project. You can choose the tools it launches, the display mode (flow or toolbar), and the flow for each project.

- [Customizing the Project Manager Tools](#)
- [Customizing the Project Manager Display](#)
- [Customizing the Project Manager Toolbar](#)
- [Customizing the Project Flow](#)

The Customize and Add Tool dialog boxes, described at the end of this section, are commonly used for these procedures.

- [“Customize Dialog Box”](#) on page 121
- [“Add Tool Dialog Box”](#) on page 123

Customizing the Project Manager Tools

In addition to the default tools, Design Entry HDL, and Packager-XL, that you can start from Project Manager, you can add more tools to a project. For each project, you can do the following:

- [Adding a Tool to Project Manager](#)
- [Removing a Tool from Project Manager](#)
- [Changing Tool Settings](#)
- [Changing the Order of Tools](#)

Adding a Tool to Project Manager

1. Open your project.

Allegro Project Manager User Guide

Project Manager Procedures

2. Choose *Options – Customize*.

3. Under *Add Tools*, click *Add*. The *Add Tool* dialog box appears.

4. If you know the name and location of the executable file of the tool you want to add,

- ☐ In the *Application Path* box, type the path to the tool's executable file.
- ☐ In the *Application Name* box, type the name of the tool.
- ☐ In the *Menu Mnemonic* box, type the access key you want to use for keyboard access to the tool.

If you would like the entry for the tool to go within a submenu, type the submenu name followed by a forward slash (/) and then the access key as follows:

<sub-menu-name>/access key

For example, to indicate that the `LibExp` application must go within a submenu called `Library Tools`, enter the following for Menu Mnemonic:

&Library Tools &Library Explorer

5. If you do not know the name of the executable file and its location, click *Browse*.

6. In the Choose *Executable File* dialog box, in the *Files of Type* list, click *Executable Files (*.exe)*.

- ☐ In the *Look in* list, select the drive that contains the tool you want to add to Project Manager. (Cadence tools are installed in the `C:\cads` directory.)
- ☐ In the list of folders below the *Look in* box, select the executable file for the tool. (Executable files for most applications have an `.exe` extension.)
- ☐ Click *Open*. The *Application Path*, *Application Name*, and *Menu Mnemonic* boxes are displayed in the *Add Tool* dialog box. You can edit these.

7. In the *Command Args* box, type the command arguments with which you want to start the tool from Project Manager. For example, if you specify a project file as a command-line argument to Design Entry HDL, Design Entry HDL will be started in the context of that project.

8. In the *Maximum Instances* list, if you want to allow more than one instance of the tool to be started from Project Manager, type or select the number of instances. For example, if you select 3, you can run three sessions of the tool from Project Manager at the same time.

Note: If you select the *Use Project File* option, you cannot select the number of instances. Only one instance of the tool is allowed.

9. If you want to set any environment variables for the tool, click *Set Environment Variables*. In the *Set Environment Variables* dialog box,
 - ☐ Type the variable name in the *Name* box.
 - ☐ Type the value for the variable in the *Value* box.
 - ☐ Click *Set*.
 - ☐ Repeat the above steps for all variables you want to add.
10. Click *OK*.

The *Environment Variables* box in the *Add Tool* dialog box displays the environment variables in the following format: *variable_name = value*.
11. Select the *Use Project File* box if you want to pass the current project file (<projectname>.cpm) as an argument to the tool when it is started from Project Manager. Clear the *Use Project File* box if you do not want to pass the project file as an argument to the tool when it is started from Project Manager.
12. Select the *Use MPS Args* box if you want to pass MPS arguments to the tool when it is started. Clear the *Use MPS Args* box if you do not want to pass MPS arguments to the tool.
13. Click *OK*.

Note: When you add a tool to your project, it is added to the *Tools* menu and to the Project Manager's toolbar display. The new tool is not automatically added to the Project Manager's flow display. To add a tool icon to the project flow, you need to edit the HTML file that defines the flow. For more information, see ["Customizing the Project Flow"](#) on page 79.

Note: There might be times when you want to add a custom menu to the Tools menu in Project Manager without first loading a design. In this case, you can specify the custom tool information with the necessary parameters in the `START_TOOLS` section of `site.cpm` and define the `CDS_SITE` environment variable so that the `site.cpm` file is located at `$CDS_SITE/cdssetup/projmgr`.

Removing a Tool from Project Manager

1. Open your project.
2. Choose *Options – Customize*.
3. Under *Add Tools*, select the tool you want to remove.
4. Click *Remove*.
5. Click *OK*.

Changing Tool Settings

To change the settings of a tool you have added to Project Manager, do the following:

1. Open your project.
2. Choose *Options – Customize*.
3. Under *Add Tools*, select the tool.
4. Click *Edit*.
5. If you want to change the application path, delete the text in the *Application Path* box and type the new path.
6. If you want to change the menu mnemonic, delete the text in the *Menu Mnemonic* box and type the new access key (for keyboard access to the tool).
7. If you want to change the command arguments, delete the text in the *Command Args* box and type the new command arguments.
8. If you want to allow more than one instance of the tool to be started from Project Manager, type or select the number of instances in the *Maximum Instances* box. For example, if you select 3, you can run three sessions of the tool from Project Manager at the same time.

Note: If you select the *Use Project File* option, you cannot select the number of instances. Only one instance is allowed.

9. If you want to change the environment variables, click *Set Env. Variables*. In the *Set Environment Variables* dialog box,
 - ☐ To add an environment variable, type its name in the *Name* box and its value in the *Value* box, and then click *Set*.
 - ☐ To remove an environment variable, select it in the *Environment Variables* list, and then click *Remove*.
 - ☐ To change the value of an environment variable, select it in the *Environment Variables* list, delete the text in the *Value* box and type the new value, and then click *OK*.
10. Select the *Use Project File* box if you want to pass the current project file (<projectname>.cpm) as an argument to the tool when it is started from Project Manager. Clear the *Use Project File* box if you do not want to pass the current project file as an argument to the tool when it is started from Project Manager.

11. Select the *Use MPS Args* box if you want to pass MPS arguments to the tool when it is started. Clear the *Use MPS Args* box if you do not want to pass MPS arguments to the tool.
12. Click *OK*.

Changing the Order of Tools

To change the order of tools in the Tools menu, do the following:

1. Open your project.
2. Choose *Options – Customize*.
3. Under *Add Tools*, select the tool you want to move.
4. Click *Move Up* to move the tool one level up or *Move Down* to move the tool one level down.
5. Click *OK*.

Important

The order you define for customized tools is only valid for your current session. Changes in the order of the tools are not stored in any external files, such as `.cpm` files, and hence cannot be restored in the next session.

Customizing the Project Manager Display

For each project, you can set the default Project Manager display as either a project flow or a toolbar.

To customize the Project Manager display, do the following:

1. Open your project.
2. Choose *Options – Customize*.
3. Under *Display*, do one of the following:
 - ☐ Click *Flow* to display Project Manager as a flow.
 - ☐ Click *ToolBar* to display Project Manager as a toolbar.
4. Click *OK*.

Customizing the Project Manager Toolbar

You can customize the size of the icons displayed in the Project Manager toolbar. You can also choose whether the toolbar should be displayed on top of all open windows or in the background.

To customize the toolbar, do the following:

1. Open the project for which you want to customize the toolbar.
2. Choose *Options – Customize*.
3. Under *ToolBar*, do one of the following:
 - ☐ Click *Large Buttons* for large icons on the toolbar.
 - ☐ Click *Small Buttons* for small icons on the toolbar.
4. Select the *Always on Top* box to display the toolbar on top of any open window. Clear the *Always on Top* box if you do not want to display the toolbar on top of any open window.

Note: This option is supported only on Windows NT.

5. Click *OK*.

Customizing the Project Flow

The Project Manager flow is an HTML file, which is in the `<your_inst_dir>/share/cdssetup/projmgr/flows` directory. To customize the project flow, you need to edit this file or replace it with your own HTML file that defines a flow.

Note: When you add or remove tools from Project Manager, changes appear only in the Tools menu. The project flow does not change automatically. To change the project flow, you must edit the HTML file.

To use a custom project flow,

1. Create an HTML file that defines a text flow or a graphical flow. Use these HREFs for links to Cadence tools.
When evaluating an HREF defined in the flow, Project Manager first looks at its own list of tools to see if the URL matches a tool name. If it does, the tool is launched as a separate process. If no match is found, Project Manager attempts to load the referenced URL.
2. If you want to use the custom flow for an individual project, do the following in Project Manager:

- a. Open your project with the *File – Open* command.
- b. Choose *Options – Customize*.
- c. In the *Flow URL* box under *Project Flow*, type the path to your HTML file or click *Browse* and use the file browser to select it.
- d. Click *OK*.

3. If you want to use your custom flow as the default for all your projects, name the HTML file `home.htm` and place it in the following directory:

`<your_inst_dir> /share/local/cdssetup/projmgr/flows`

If Project Manager finds a `home.htm` file in this location, it uses it as the default project flow. Otherwise, it picks up the default flow from the Cadence installation directory.

HTML Code for Default Board Design Flow

The built-in HTML code for the standard Cadence Board Design Flow is shown as follows.

```
<!DOCTYPE HTML PUBLIC "-//IETF//DTD HTML//EN"><html><head><meta http-  
equiv="Content-Type" content="text/html; charset=iso-8859-1"><title>Project  
Manager: Main</title></head>  
  
<body><map name="index"><area shape="rect" coords="135,260,221,352"  
href="Allegro"><area shape="rect" coords="32,200,112,278" href="Boardquest"><area  
shape="rect" coords="135,143,220,219" href="Pkgrxl"><area shape="rect"  
coords="137,26,220,101" href="Concept"><area shape="rect" coords="42,76,103,135"  
href="Setup"></map>  
  
  
</body></html>
```

HREFs for Cadence Tools

The following HREFs are used to call the built-in Cadence applications:

- Allegro (Allegro Layout)
- Boardquest (SI Floorplanner)
- Pkgrxl (Packager-XL UI)
- Concept (Concept Design Entry)
- Setup (Setup UI)
- New (New Project Wizard)

- Open (Open Project dialog box)

Example: Creating a Simple Text-Based Flow

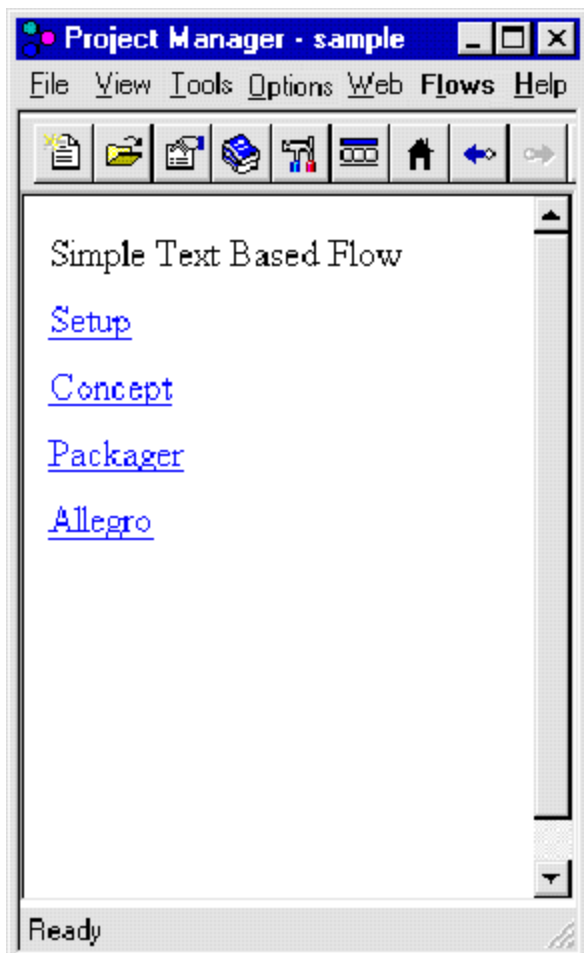
To create a simple text-based flow, you only need to create links in your HTML document to the application names that Project Manager recognizes. The following example shows a basic flow that calls Setup, Design Entry HDL, Packager-XL, and PCB Editor.

```
<!DOCTYPE HTML PUBLIC "-//IETF//DTD HTML//EN">
<html>
<head>
<meta http-equiv="Content-Type"
content="text/html; charset=iso-8859-1">
<meta name="GENERATOR" content="Microsoft FrontPage 2.0">
<title>Simple Text Based Flow</title>
</head>
<body>
<p>Simple Text Based Flow</p>
<p><a href="setup">Setup</a></p>
<p><a href="concept">Concept</a></p>
<p><a href="pkgrxl">Packager</a></p>
<p><a href="allegro">Allegro</a></p>
</body>
</html>
```

Allegro Project Manager User Guide

Project Manager Procedures

To load this flow in Project Manager, you choose the *Options> Customize* command and enter the path to the main HTML file in the *Project Flow* box in the *Customize* dialog box. The flow displayed in the following image then becomes your project flow.



Example: Creating a Graphical Multiple Page Flow

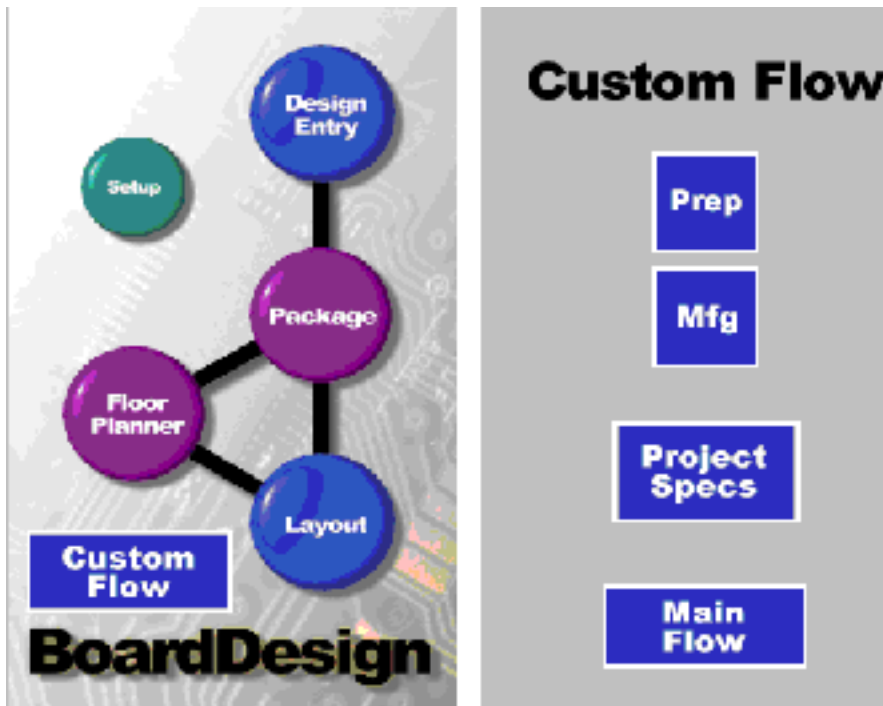
To create a graphical flow, you have to create an image file in either the .gif or .jpg format and define an image map that associates URLs with specific areas on the graphic. There are many utilities available for image and image map creation. The following example was created with Corel Photo Paint. Photo Paint allows URLs to be associated with objects in an image and will automatically create HTML files that load the image and define the image map. Using this method, you can create complex, graphical flows that require no additional HTML programming.

Allegro Project Manager User Guide

Project Manager Procedures

In this example, we define a multiple-page flow. It includes a modified version of the Cadence Board Design Flow and a custom page that launches custom applications and contains Web links related to the design project.

To begin, we created the images that would be used for the flows. We took a screen shot of the Cadence Board Design Flow and added a new box that is used to call the custom flow. The custom flow can launch two custom applications—a prep tool used to do customer-specific library preparation for the project and a board post-processing program for manufacturing. There is also a link to an HTML page that allows access to online specifications and review documentation for the project. The two images are shown as follows:



Next, we assigned the HREFs for the various applications used in the flows. The HTML files for both flows are shown as follows.

```
<HTML>
<HEAD>
<TITLE>customflow1.htm</TITLE>
</HEAD>
<BODY>
<IMG SRC="CustomFlow1.jpg" ALT="Clickable Image" USEMAP="#mainflow"></A>

<MAP NAME="mainflow">
```

Allegro Project Manager User Guide

Project Manager Procedures

```
<!-- PHOTO-PAINT 7.0 IMAGE MAP -->
<AREA SHAPE=CIRCLE COORDS="71,106,29" HREF=Setup ALT="Setup"></AREA>
<AREA SHAPE=CIRCLE COORDS="177,63,40" HREF=Concept ALT="Concept-HDL"></AREA>
<AREA SHAPE=CIRCLE COORDS="71,239,39" HREF=Boardquest ALT="BoardQuest"></AREA>
<AREA SHAPE=RECT COORDS="137,142,216,220" HREF=Pkggrxl ALT="Packager-XL"></AREA>
<AREA SHAPE=CIRCLE COORDS="179,305,22" HREF=Allegro ALT="Allegro"></AREA>
<AREA SHAPE=RECT COORDS="12,307,126,355" HREF=myflow.htm ALT="My Flow"></AREA>
</MAP>
</BODY>
</HTML>
```

The only difference from the standard Cadence Board Design Flow is the addition of the HREF to *myflow.htm*. This reference loads the custom design flow.

```
<HTML>
<HEAD>
<TITLE>myflow.htm</TITLE>
</HEAD>
<BODY>
<IMG SRC="Myflow.jpg" ALT="Clickable Image" USEMAP="#myflow"></A>

<MAP NAME="myflow">
```

```
<!-- PHOTO-PAINT 7.0 IMAGE MAP -->
<AREA SHAPE=RECT COORDS="96,86,156,144" HREF=Prep ALT="Project Library Prep"></AREA>
<AREA SHAPE=RECT COORDS="96,153,157,212" HREF=Mfg ALT="Post Process Board"></AREA>
<AREA SHAPE=RECT COORDS="72,243,180,303" HREF=specs.htm ALT="Project Specifications"></AREA>
<AREA SHAPE=RECT COORDS="65,336,185,390" HREF=customflow1.htm ALT="Main Flow"></AREA>
</MAP>
</BODY>
</HTML>
```

Prep and Mfg are both applications that were added to Project Manager and *specs.htm* and *customflow1.htm* are links to an additional HTML page and the original flow.

Adding a Customized Flow

Project Manager can open multiple flows on a project. This section documents how to add a customized flow to Project Manager. For details on how to create a customized flow, see [Customizing the Project Flow](#).

Allegro Project Manager User Guide

Project Manager Procedures

To add a customized flow to Project Manager, you need to edit the `site.cpm` file at `your_inst_dir/share/local/cdssetup/projmgr/site.cpm` or `$CDS_SITE/cdssetup/projmgr/site.cpm`. If this file does not exist, you need to create one.

If `site.cpm` exists in either of the locations specified above, modify the file as follows:

1. Open the `site.cpm` file in a text editor.
2. Add the following text to the file:

```
START_DESIGN_FLOWS
flow<num> '<Flow Name>' '<Path to flow html file>'
END_DESIGN_FLOWS
```

The parameter `flow<num>` represents a text string in the form `flow0`, `flow1`, and so on. The integer value after the text flow should be incremented by 1 for every flow.

To find out the next value, open the `cds.cpm` file at `your_inst_dir/share/cdssetup/projmgr/cds.cpm` and locate the last `flow<num>` entry in the `START_DESIGN_FLOWS` section. The value for `<num>` must be this number incremented by 1.

Note: If the `site.cpm` file already has existing entries for flows, you should locate the last `flow<num>` entry in the `START_DESIGN_FLOWS` section of `site.cpm`.

For example, the `cds.cpm` file has the following text:

```
START_DESIGN_FLOWS
flow0 'Board Design' '$CDS_INST_DIR/share/cdssetup/projmgr/flows/main.htm'
flow1 'Library Management' '$CDS_INST_DIR/share/cdssetup/projmgr/flows/
lmanflow_main.htm'
flow2 'Programmable IC' '$CDS_INST_DIR/share/cdssetup/projmgr/flows/
synth.htm'
END_DESIGN_FLOWS
```

Here, `flow2` is the last flow entry. Therefore, the next entry in `site.cpm` should be `flow3`.

The parameter `<Flow Name>` represents a user-friendly name you give to a flow. Enclose the flow name string within single quotes if the name contains spaces. This name will appear in the *Flows* menu of Project Manager and can be selected. In the above example, 'Board Design', 'Library Management' and 'Programmable IC' are flow names.

The `<Path to flow html file>` parameter represents the path to the html file that represents the flow.

Note: If you specified any project flows using the *Flow URL* button in the *Tools – Customize* dialog box, this flow will appear in the *Flows* menu with the name *Custom Flow*.

Starting Project Manager from the Command Line

You can start Project Manager from the command line by using the following command:

```
projmgr -proj <path-to-project> [-product <suite-name>] [-mpsession <session-name>]
```

The `-proj` option is compulsory. You specify the name of the project in `<path-to-project>`.

You specify the name of the suite joined with underscores with the `-product` option. You can write the suite names as follows:

- `concept_hdl_expert`
- `concept_hdl_studio`
- `pcb_librarian_expert`
- `concept_hdl_expert`
- `concept_hdl_studio`
- `allegro_performance`
- `Allegro_Design_Editor_620`
- `pcb_librarian_expert`
- `Allegro_Frontend_PCB_Solution`
- `Allegro_Venture_SDA`
- `Allegro_Enterprise_SDA`
- `Allegro_Enterprise_PCB_Designer`
- `Allegro_Venture_PCB_Designer`
- `specctraquest_si_expert`

Suite names are not case-sensitive.

Note: The option `Tool Licenses (legacy)` is available only through the GUI.

You specify the messaging session in which you want to start Project Manager with the `-mpsession` option. All tools in the same session can interoperate. If you do not specify this, Project Manager reads the session name from the project file or creates one if no session name is found in the project.

Allegro Project Manager User Guide

Project Manager Procedures

Session names are symbolic strings that are used to enable intertool communication. All tools running in the same session can communicate with each other. Project Manager passes the session name provided to all tools it invokes.

The following rules apply to creating session names:

- If a `-mpsession` argument is provided on the command line, that string is taken as it is without modifications, for the session name.
- If the session name is read from the project file, or created by Project Manager, it is prefixed by the string `<username>_` so that multiple users working on the same project do not run into problems when invoking tools. Here, `<username>` is the login ID of the user invoking Project Manager.

Allegro Project Manager User Guide
Project Manager Procedures

Setting Up Projects

Overview

The Project Setup window displays the current global, physical part table, tools, expansion, and views settings for your project.

You can perform a number of tasks from Project Setup.

Changing the Root Design for a Project

To change the root design for a project from Project Manager, do the following:

1. Open the project for which you want to change the root design.
2. Choose *Tools – Setup*. The *Project Setup* window appears.
3. Select the *Global* tab.
4. In the *Library Name* list, select the library containing the design.
5. In the *Design Name* field, type the name of the design or click *Browse* and select the design from the *Select Cell* list.
6. Click *Apply* to save your changes, or *OK* to save your changes and exit *Project Setup*.

Note: You can also create a new root design from Project Manager.

Creating a New Root Design for a Project

To create a new root design for a project from Project Manager, do the following:

1. Open the project in which you want to create a new design.
2. Choose *Tools – Setup*. The *Project Setup* window appears.
3. Select the *Global* tab.

4. In the *Library Name* list, select the library in which you want to create the new design.

Note: The *Library Name* list is the list of project libraries.

5. In the *Design Name* field, delete the text and type the new design name. Click *Browse* to see a list of existing cell names for the library you have selected.
6. Click *Apply* to save your changes, or *OK* to save your changes and exit *Project Setup*.

Editing the cds.lib File

Each project has a `cds.lib` file. Project Manager creates the `cds.lib` file when you create a project in a new folder or in a folder that does not contain a `cds.lib` file.

The new `cds.lib` contains the following:

- A directive to include the installed Cadence libraries. (For example: `INCLUDE <your_install_dir>/share/cdssetup/cds.lib`).
- A define statement that maps the logical project library (`projectname_lib`) to its physical name (`worklib`). (For example: `DEFINE myproject_lib worklib`.)

You can edit the `cds.lib` file and add directives to include any other libraries such as your company libraries. You can add libraries to `cds.lib` directly by specifying their logical names and physical locations. Or you can add a file that contains a list of libraries and their physical locations.

The `cds.lib` file determines the list of available libraries from which you can choose the project libraries for a project.

To edit the `cds.lib` file,

1. Open the project.
2. Choose *Tools – Setup*. The *Project Setup* window appears.
3. Select the *Global* tab.
4. Click the *Edit* button next to the `cds.lib` field. The `cds.lib` file is opened in the default text editor.
5. Edit the `cds.lib` file.

You can add libraries to the `cds.lib` file directly by specifying their logical names and their physical locations. (For example, `DEFINE MYLIB C:/Libraries/IEEE`). You can also add files that contain a list of libraries and their locations. (For example, `INCLUDE C:/`

Libraries/company.lib, where `company.lib` contains a list of libraries and their locations.) See Syntax for adding libraries to the `cds.lib` file.

6. Save the file and exit the text editor.
7. In the confirmation window, click *Yes* to update the library list.
8. Click *Apply* to save your changes, or *OK* to save your changes and exit *Project Setup*.

Adding Libraries to the cds.lib File

To add a library,

- Add the following statement to the `cds.lib` file for the project:

```
DEFINE libraryname librarypath
```

where `libraryname` is the logical name for the directory specified in `librarypath`.

The `libraryname` is the name that appears in the list of Available Libraries in Project Setup.

Example:

```
DEFINE MYLIB C:/Libraries/IEEE  
DEFINE lsttl C:/Libraries/lsttl
```

To add a file containing a list of libraries,

- Add one of the following statements to the `cds.lib` file for the project:

```
INCLUDE filename
```

```
SOFTINCLUDE filename
```

where `filename` is the name of a file containing a list of libraries and their locations. (`filename` can also be another `cds.lib` file.) If you use the `INCLUDE` statement, an error message is generated if Cadence tools cannot find this file. If you use the

`SOFTINCLUDE` statement, an error message is not generated if Cadence tools cannot find this file.

All the libraries in the file will appear in the list of Available Libraries in Project Setup.

Example:

```
INCLUDE C:/Libraries/mycompany.lib
```

To remove a library,

- Add the following statement to the `cds.lib` file for the project:

```
UNDEFINE libraryname
```

where `libraryname` is the library you want to remove.

Use this statement when you want to remove some of the libraries defined in a file you included with `INCLUDE` or `SOFTINCLUDE` statements.

Selecting Libraries for a Project

1. Open the project.
2. Choose *Tools – Setup*. The *Project Setup* window appears.
3. Select the *Global* tab.
4. If you want to view the contents of a library, select the library and click *View*. A window displaying the contents of the library appears. You cannot make any changes in this window.
5. Modify the *Project Libraries* list under *Library*.
6. To add one library, select the library in the *Available Libraries* list and click *Add*.
7. To add all the libraries in the *Available Libraries* list, click *Add All*.
8. To remove one library, select the library in the *Project Libraries* list and click *Remove*.
9. To remove all the libraries in the *Project Libraries* list, click *Remove All*.
10. Choose the search order for the project libraries. The order in which the libraries are listed in the *Project Libraries* list determines their search order.
11. To move a library one level up, select the library and then click *Up*.
12. To move a library one level down, select the library and then click *Down*.
13. Click *Apply* to save your changes, or *OK* to save your changes and exit *Project Setup*.

Available Libraries and Project Libraries

Available Libraries

They are the libraries available to you for any project. These are determined by the directives in the `cds.lib` file. The Cadence-installed libraries are included in `cds.lib` as the default libraries. You can edit the `cds.lib` file to add other libraries to the list of available libraries.

Project Libraries

They are the libraries you select for your project from the list of available libraries. You can select project libraries when you create a project or at any time from the Setup tool. If you create a project in a new folder or in a folder that does not have a `cds.lib` file, a `projectname_lib` library is also created and placed in the Project Libraries list.

You can modify the Project Libraries list from Project Manager.

Adding Physical Part Table Files to a Project

The Physical Part Table (`.ptf`) file contains the data you need to add or modify the physical properties of a symbol. The `.ptf` files can be located at the cell level under the Part Table view, or in any other directory. Cell-level `.ptf` files contain information about the primitives for that cell.

To access the information contained in the Physical Part Table files, you must include them in your project. When you include cell-level Physical Part Tables, all the `.ptf` files in the Part Table view of that cell are included. You can also include other `.ptf` files by specifying their location.

You can include cell-level `.ptf` files and other `.ptf` files in the same project. If you have a cell-level `.ptf` file, then Packager-XL does not read it if the `INCLUDE_PPT` directive is set. To include the cell-level `.ptf` file, you will have to add it in the `INCLUDE_PPT` directive.

To add physical part tables to your project, you can add either `.ptf` files directly or directories that contain `.ptf` files. For example, if the `lsttl` directory contains the `lsttl.ptf` file, you can add either the complete path to the `lsttl.ptf` file or just the path to the `lsttl` directory. When you add a directory, all the `.ptf` files in that directory are added to the project. You can then exclude some of the `.ptf` files if you do not want them in the project.

The following steps are required to add Physical Part Table files to a project:

1. Open the project.
2. Choose *Tools – Setup*. The *Project Setup* window appears.
3. Select the *Part Table* tab.

4. To add cell-level .ptf files to your project, select the *Use Cell-Level Physical Part Table Files* check box. All the .ptf files contained in the Part Table view of the cells will be read by Packager-XL.
5. To add other Physical Part Table files,
 - a. Under *Physical Part Table Files*, click *Add*. The *Add Physical Part Table* dialog box appears.
 - b. Type the name and the path of the .ptf file or the directory containing the .ptf files. To add more than one path, separate each path with a space.
—or—
To add a file, click *File...* and select the .ptf file in the *Choose Physical Part Table Files* dialog box. (To select more than one file, select the first one, then press *CTRL* and select the others.) To add a directory, click *Directory...* and select a directory in the *Choose Directory* dialog box.
 - c. Click *OK*.
6. To exclude any unwanted .ptf files contained in the directories you have added, do one of the following:
 - ☐ Under *Exclude Physical Part Table Files*, click *Add* and enter the name and path of the .ptf file you want to exclude from your project. Repeat this step for all the files you want to exclude.
 - ☐ Under *Include PTFs*, click *Add* and enter the name and path of the .ptf file you want to include in your project. Repeat this step for all the files you want to include.
 - ☐ To remove a Physical Part Table file or directory, select the file and click *Remove*.
 - ☐ Select the *Merge Physical Part Table Files* check box to merge the information in all included Physical Part Table files.
 - ☐ Select the *Perform Case Sensitive Row Match* check box to perform a case-sensitive match of key properties for a part in the physical part table files.
7. Click *Apply* to save your changes, or *OK* to save your changes and exit *Project Setup*.

You can include cell-level .ptf files and other .ptf files in the same project – Packager-XL reads the contents of each.

Setting Up Tools

The Tools tab in the Project Setup window allows you to select the setup options for PCB Editor, Design Entry HDL, Project Manager, Packager-XL, Programmable IC, Simulation, and

Mixed Signal simulation. You can specify setup directives for these tools from Project Manager or directly from the tools. The Tools tab also displays the default settings for the property file, text editor, project log file, and the temp directory.

In this tab, you can do the following:

- Specify setup directives for PCB Editor, Design Entry HDL, Project Manager, Packager-XL, Programmable IC, Simulation, and Mixed Signal simulation. Simulation Interface provides a simulation environment to simulate your schematics from Design Entry HDL with Verilog or Leapfrog. You can specify setup directives for Design Entry HDL and Packager-XL directly from the tools or from Project Manager.
- Choose a default text editor.
- Specify an Application Temp Directory.
- Select a Property File.
- Set up a Log File.

Note: You can specify setup directives for Design Entry HDL and Packager-XL directly from the tools or from Project Manager.

Specifying the Application Temp Directory

The Application Temp directory is the directory in which applications such as Design Entry HDL store temporary files. You can delete the contents of this directory.

An Application Temp directory (`temp`) is created automatically when you create a new project. However, you can specify any directory as the Application Temp directory for a project.

To specify the Application Temp directory,

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Tools* tab.
3. In the *Temp Directory* field, type the full path to the *temp* folder, or click *Browse* and use the file browser to select the location of the *temp* folder.
4. Click *Apply* to save your changes, or *OK* to save your changes and exit Project Setup.

Selecting a Text Editor

For each project, you can select a text editor as the default text editor for Cadence tools. When you view or edit any text file from a Cadence tool, it will be displayed in the text editor you have specified. The default editor is WordPad on Windows NT.

To select a text editor,

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Tools* tab.
3. In the *Default Text Editor Path* field, type the full path to the text editor you want to use, or click *Browse* and use the file browser to select the text editor.
4. Click *Apply* to save your changes, or *OK* to save your changes and exit *Project Setup*.

Note: You can set the default text editor for Design Entry HDL by setting the `CDS_TEXT_EDITOR` environment variable. To do this, use the following command:

```
setenv CDS_TEXT_EDITOR<text_editor>
```

If this variable is set, Design Entry HDL will not use the text editor specified in the *Tools* tab of *Project Setup*. You should unset this variable if you want Design Entry HDL to use the text editor setting in the *Tools* tab of *Project Setup*.

Selecting a Property File

The property file for a project contains directives that control how properties are handled during expansion. It specifies whether a property is inherited by other objects, whether it is a parameter, what objects it can be attached to, and whether it is passed to the destination tool.

Cadence provides a default property file called `cdsprop.paf`, which is located in the `<your_install_dir>/share/cdssetup` directory. Do not modify this file. You can use your own property file by specifying its path in *Project Setup*.

To select a property file,

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Tools* tab.
3. In the *Property File* field, type the full path of the property file you want to use, or click *Browse* and use the file browser to select the file.

4. Click *Apply* to save your changes, or *OK* to save your changes and exit Project Setup.

Sample cdsprop.paf File

```
FILE_TYPE=ATTRIBUTES;
{ Default attributes for properties.
  Attributes for user designed properties should be added
  to the user's property attribute file. This file should
  not be modified. }
ALLOW_CONNECT:inherit(signal), permit(pin, body,signal);
AUTO_GEN:                                inherit(),
permit(body);
BIDIRECTIONAL:inherit(), permit(pin);
BODY_NAME:  inherit(), permit(body);
CHIP_DELAY: inherit(pin), permit(pin, signal);
CLOCK_DELAY:inherit(pin), permit(pin, signal);
COMMENT_BODY:filter;
COUPLED:    inherit(), permit(body);
DELAY:      inherit(), parameter, permit(body);
DIR:        inherit(), permit(body), case_sensitive;
EVAL:       inherit(pin), permit(pin, signal);
EXPR:       filter;
FALL:       inherit(), parameter, permit(body);
GROUP:      inherit(body), permit(body);
HAS_FIXED_SIZE:inherit(), permit(body);
HIGH:       inherit(), permit(body);
INPUT_LOAD: inherit(), permit(pin);
IO_NET:     inherit(signal), permit(signal);
LAST_MODIFIED:inherit(), filter;
LOCATION:    inherit(body), permit(body);
LOCATION_CLASS:inherit(body), permit(body);
LOW:        inherit(), permit(body);
MODEL:      inherit(), permit(body);
```

Setting Up a Log File

A log file for a project tracks information such as the date and time of any activity, the tools launched from the project, the user's name, and MPS sessions and hosts.

If you want to maintain a log file for the project, you must select the option in Project Setup. A log file will not be generated by default.

To set up a log for a project,

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Tools* tab.

3. In the *Project Log File* field, type a name for the log file. The file will be created in the project directory. To specify an existing file, click *Browse* and use the file browser to select it.
4. Click *Apply* to save your changes, or *OK* to save your changes and exit Project Setup.

Selecting an Expansion Type

The expansion type and configuration you select in Project Setup determine the current configuration for Design Entry HDL and Hierarchy Editor. If you change the expansion type in Project Setup, the current configuration for Design Entry HDL and Hierarchy Editor changes. The default expansion type is Physical Layout.

To select the expansion type for your design,

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Expansion* tab.
3. Do one of the following:
 - ☐ Select the *Physical Layout* option to expand your design for PCB Editor and other back-end tools.
 - ☐ Select the *Verilog Simulation* option to expand your design for simulation with Verilog-XL and other Verilog-based simulators.
 - ☐ Select the *VHDL Simulation* option to expand your design for simulation with Leapfrog and other VHDL-based simulators.
 - ☐ Select the *PIC Configuration* option to expand your design for simulation with Programmable IC such as Verilog-XL.
 - ☐ Select the *Mixed Signal* option to expand your design for mixed-signal simulation.
4. In the *View* field next to the expansion type you selected, click *Browse*. The *Select View* dialog box appears. Select the configuration you want to expand, and click *OK*.
5. If you want to view or edit the configuration, click *Edit*. The configuration is opened in the Cadence Hierarchy Editor. Edit the configuration, save it with the *File – Save* command, and exit the Cadence Hierarchy Editor.
6. Click *Apply* to save your changes, or *OK* to save your changes and exit Project Setup.

Note: When you package a design, Packager-XL always uses the Physical Layout expansion, irrespective of which expansion type you select in Project Setup.

Selecting the Configuration for Expansion

When you create a new project, the default configurations for each expansion type are created automatically. These are listed as follows:

- `cfg_package` for Physical Layout
- `cfg_verilog` for Verilog Simulation
- `cfg_vhdl` for VHDL Simulation
- `cfg_pic` for PIC Simulation
- `cfg_mixed` for Mixed Signal Simulation

You can select a different configuration for each expansion type.

To select the configuration for each expansion type,

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Expansion* tab.
3. Click the *Browse* button against an expansion type. The *Select View* dialog box appears. Select the configuration you want to use and click *OK*.
4. Click *Apply* to save your changes, or *OK* to save your changes and exit Project Setup.

Note: You can also create a new configuration view from Project Manager.

Editing a Configuration

You edit a configuration with the Hierarchy Editor, a graphical tool for creating and editing configurations.

To edit a configuration,

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Expansion* tab.
3. In the *View* field next to an expansion type (Physical Layout, Verilog Simulation, VHDL Simulation, PIC Configuration and Mixed Signal), click *Browse* and select the configuration you want to edit.

4. Click *Edit*. The configuration you specified in the *View* field is opened in the Cadence Hierarchy Editor.
5. Make the required changes in the Hierarchy Editor, save the changes, and exit the Hierarchy Editor.
6. Click *Apply* to save your changes, or *OK* to save your changes and also exit Project Setup.

Creating a New Configuration View

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Expansion* tab.
3. In the *View* field next to the *Expansion Type* option you have chosen, delete the existing view name and type the name for the new view.
4. Click *Apply* to save your changes, or *OK* to save your changes and exit Project Setup.

Project Manager creates the new view as well as an `expand.cfg` file in the view.

Selecting Views for the Project

Views are created when you work with your designs. When you package a design, a packaged view is created and all Packager-XL output files and log files are stored in it. The `chips.prt` file is placed in the `chips` view, the cell-level physical part table files in the `part_table` view, and PCB data in the `physical` view.

Similarly, the root design views for Board Design, PIC Design, Verilog Simulation, and VHDL Simulation will be used for expanding your design for physical layout, PIC simulation, Verilog simulation, and VHDL simulation, respectively.

Project Manager assigns default view names as follows:

:

Type of View	View Name
Packaged	packaged
Chips	chips
Part table	part_table

Allegro Project Manager User Guide

Setting Up Projects

Type of View	View Name
Physical	physical
Constraints	constraints
Board Design	sch_1
Programmable IC Design	sch_1
Verilog Simulation	sim_sch_1
VHDL Simulation	sim_sch_1

You can change the view names for each project.

Changing View Names

You can create a new view name or select an existing view name. Open the project for which you want to change view names.

1. Choose *Tools – Setup*. The *Project Setup* window appears.
2. Select the *Views* tab.
 - ☐ To change the view name, click on the drop-down list to select from an existing view name.
 - ☐ To create a new view name, enter the new view name.
3. Click *Apply* to save your changes, or *OK* to save your changes and exit Project Setup.

Allegro Project Manager User Guide
Setting Up Projects

Menu Commands

This sections explains the following menu commands in Project Manager:

- File – New
- File – New – New Design
- File – New – New Library
- File – Open
- File – Close
- File – Project Path
- File – Copy Project
- File – Import
- File – Import IFF – RF-PCB
- File – Import IFF – Standard
- File – Export
- File – File Viewer
- File – Import IFF
- File – Change Product
- File – Exit
- View – Toolbar
- View – Status Bar
- View – Project Settings
- View – Project Libraries
- View – Running Tools

Allegro Project Manager User Guide

Menu Commands

- View – Hide Flow
- Tools – Library Tools
- Tools – ViewCaptureSymbol
- Tools – Variant Editor
- Tools – Programmable IC
- Tools – Simulate
- Tools – EDIF 300
- Tools – SigXplorer
- Tools – Rules Checker
- Tools – CRefer
- Tools – Archive
- Tools – Packager Utilities
- Tools – Hierarchy Editor
- Tools – Setup
- Tools – Design Entry
- Tools – Design Sync
- Tools – PCB Editor
- Tools – Allegro SI
- Options – Customize
- Web – Back
- Web – Forward
- Web – Home
- Web – Stop
- Web – Reload
- Web – Add To Favorites
- Web – Go to Favorites
- Flows – Board Design

- Flows – Library Management
- Flows – Programmable IC

File – New

Opens a submenu using which you can specify if you want to create a design project or a library project. Depending on your choice, a wizard comes up to help you create a new project.

File – New – New Design

Opens the New Project Wizard, using which you can create a new design project.

File – New – New Library

Opens the New Project Wizard, using which you can create a new library project.

File – Open

Opens a project. The project flow is displayed in Project Manager and the menu commands are no longer disabled. When you start any tool such as Design Entry HDL, it comes up in the context of the project you have opened. To see current project settings, choose View – Project Settings.

File – Close

Closes a project. Project Manager displays the initial flow with three options – Open Project, Create Design Project, and Create Library Project. Many menu commands are disabled until you open or create a project.

File – Project Path

Displays the project path. Project Path displays the full path of the current project in the status bar.

File – Copy Project

Copies a project. Copy Project functionality can be used to copy an existing project to a new location or to copy an existing project to the same location with different design and library names. When you copy a project from one location to the other. The entire project hierarchy is copied from the source to the destination, thereby ensuring that there is no loss or corruption of data. Also, if a design library or a root design is renamed, Design Entry HDL ensures that the copied hierarchy is updated with respect to the root design cell and design library.

File – Import

Imports the project file (*projectname.cpm*) into the current project, replacing the existing project file. All the setup information for the project – libraries, view names, physical part tables, and tool setup directives – is replaced.

File – Import IFF – RF-PCB

Procedure

Starts the RF-PCB IFF Import wizard, which helps you import a schematic IFF file into a project.

File – Import IFF – Standard

Procedure

Starts the standard IFF Import dialog box, which helps you import an RF design (IFF file) into a Design Entry HDL project.

File – Export

Saves a copy of the project file of the current project *<projectname>.cpm* in the location you specify. You can choose to export all the settings of the project, including the default setup directives specified in the Cadence *cds.cpm* file and your company's *site.cpm* file. Or, you may want to export only the setup information specified in your *<projectname>.cpm* file.

File – File Viewer

Displays the Open dialog box, which lets you select the file you wish to view. This option can be used to view the log files that are created in different locations.

File – Import IFF

Displays the IFF Import dialog box, which allows you to import a schematic in the IFF format into Design Entry HDL.

File – Change Product

Displays the *Cadence Product Choices* dialog box, which allows you to select the product suite whose license you want to use.

File – Exit

Closes Project Manager.

Tools started from Project Manager are not closed automatically when you close Project Manager.

View – Toolbar

Displays the toolbar in the Project Manager window. The toolbar contains icons of some of the commonly used commands in the menus. Place and hold your cursor over an icon to see its function. When the toolbar is selected, the Toolbar command in the View menu has a check-mark in front of it.

You can move the toolbar and place it anywhere within the Project Manager window by dragging it.

To hide the toolbar, choose View – Toolbar. When the toolbar is not selected, the Toolbar command in the View menu does not have a check-mark in front of it.

View – Status Bar

Displays the status bar at the bottom of the Project Manager window. When the status bar is selected, the Status Bar command in the View menu has a check-mark to its left.

To hide the status bar, choose View – Status Bar. When the status bar is not selected, the Status Bar command in the View menu does not have a check-mark to its left.

View – Project Settings

Brings up a window displaying all the settings for the current project: global directives, Project Manager directives, directives for individual tools such as Design Entry HDL, and Web page directives.

Project settings are the setup options you have chosen for each project. Global settings include the selection of libraries, expansion types, view names, physical part table files, and property files. Project settings for individual tools are the setup options for each tool.

Project settings are displayed in a tree form. A plus icon to the left of any branch of the tree indicates there are one or more levels of hierarchy below it that are not displayed. Click the plus icon to display the level below it. A minus icon to the left of any branch of the tree indicates that the level below it is already expanded. To collapse any branch of the hierarchy, click the minus icon on its left.

View – Project Libraries

Brings up a window displaying the libraries for the current project in a tree form.

A plus icon to the left of any branch of the tree indicates there are one or more levels of hierarchy below it that are not displayed. Click the plus icon to display the level below it.

A minus icon to the left of any branch of the tree indicates that the level below it is already expanded. To collapse any branch of the hierarchy, click the minus icon in front of it.

A page icon denotes a text file. Double-click on it to view the contents of the text file.

View – Running Tools

Brings up a window displaying the list of tools currently running from Project Manager, as well as the process ID, user name, host name, and session name for each tool. The equivalent command-line arguments are also displayed.

View – Hide Flow

Switches the Project Manager display mode from project flow to toolbar.

To access the Project Manager commands from the toolbar, right-click the Project Manager icon.

To switch the Project Manager display back to the project flow, click the Project Manager icon.

Tools – Library Tools

Opens a submenu from which you can select tools used in the library management flow.

- The Setup Templates option helps you create a new template, edit an existing template, or extract a template from a part. When you click this button, the Setup Template dialog box appears.
- The Search Part option opens the Cadence Web page that provides information about the Part Browser utility and lets you freely download the utility.
- The Library Explorer option opens the Library Explorer window.
- The Import submenu lets you import a Capture part or an XML file into a Design Entry HDL library. It also lets you convert an XML file to a Capture part.
- The Part Developer option opens Part Developer, which helps you create, edit, and verify part data. It gives you one interface in which you can create simulation views and edit schematic symbols, physical pin data, and part table data.
- The Edit Simulation Views option opens the VHDL/Verilog Support dialog box where you can specify an existing or new VHDL or Verilog wrapper or map file for a part.
- The View Design Entry Symbol option opens the View Symbol dialog box in which you can specify a symbol that you want to view and then view it in Design Entry HDL.
- The View PCB Editor Symbol option opens the Footprint dialog box where you can specify a footprint and view it in PCB Editor.

- The Pad Stack Editor option launches the Pad Stack Designer, which lets you create and edit padstacks and save them to your library.
- The Verification submenu helps you verify part templates, Design Entry HDL libraries and parts, and footprints.
- The Allegro Symbol Editor launches PCB Editor.
- The Export submenu helps you export a Design Entry HDL part to Capture or XML file. It also helps you convert a Capture part to XML.

Tools – ViewCaptureSymbol

Starts the Capture tool.

Tools – Variant Editor

Starts the tool Variant Editor, which lets you create and manage designs that are different from each other by small differences.

Tools – Programmable IC

Opens a submenu with options using which you can start tools used in the PIC flow.

- The *Verify Synthesis* option allows you to simulate the post-synthesis verilog.
- The *Verify PnR* option allows you to simulate the routed verilog, along with the timing information.
- The *Build Physical* option takes you from vendor-specific tools back to Design Entry.
- The *Place and Route* option invokes vendor-specific Place and Route tools, and takes you from vendor to Design Entry HDL.
- The *PIC Netlister* option starts the FPGA/PLD Netlister window, which allows you to synthesize the high-level design in HDL using Synplify.
- The *PIC HDL Import* option invokes the HDL Design Entry window, which allows you to import a design in the Verilog or VHDL format into your project.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – Simulate

Brings up the Simulation Interface tool.

Tools – EDIF 300

Starts EDIF 300, which consists of the EDIF Reader and the EDIF Writer. The EDIF Reader translates an EDIF netlist to a Design Entry HDL view and the EDIF Writer generates an EDIF netlist from Design Entry HDL schematics.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – SigXplorer

Starts SigXplorer, a tool for working with topology elements, transmission lines, and constraints.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – Rules Checker

Starts Rule Checker, a tool for checking designs for violation of design rules.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – CRefer

Starts CRefer HDL, a tool for cross-referencing the signals in a design and generating a list of all the signals in the design.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – Archive

Opens a submenu in which you can create or open an archive by using Archiver, a tool that archives all the data needed to run a design at another location.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – Packager Utilities

Includes utilities for generating a Bill of Materials, performing electrical rule checks, generating netlist reports, and packaging a design in both the forward mode and the feedback mode.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – Hierarchy Editor

Starts the Hierarchy Editor, a tool for navigating hierarchical designs, and creating and editing configurations.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – Setup

Brings up the Project Setup dialog box. This dialog box displays the current global, physical part table, tools, and view setup information for your project. You can change your project settings from Project Setup.

You must open a project before you can start any tool from Project Manager.

Tools – Design Entry

Starts Design Entry HDL, a schematic editing tool with which you can create logic drawings (schematics) and body drawings. Design Entry HDL displays the top-level drawing of your current project.

You must open a project before you can start any tool from Project Manager.

Tools – Design Sync

Starts Design Synchronization, a tool with which you can export a design from Design Entry HDL to PCB Editor, import a design from PCB Editor to Design Entry HDL, compare the logical view (schematic) with the physical view (board layout), and package a design.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – PCB Editor

Starts PCB Editor, a physical layout tool for printed circuit board (PCB) design.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Tools – Allegro SI

Starts SI, an environment for high-speed system-level and board-level design planning. SI includes the SI Design Editor, SigXplorer, and the Timing Spreadsheet.

You must open a project before you can start any tool from Project Manager. Tools are started in the context of the project currently open.

Options – Customize

Brings up the dialog box for customizing Project Manager. You can customize Project Manager for each project by selecting the tools it launches, the display mode (project flow / toolbar), and the project flow.

Web – Back

Displays the previous Web page you loaded, if any.

This option is supported only on Windows NT.

Web – Forward

Displays the next Web page, if any.

This option is supported only on Windows NT.

Web – Home

Displays the Project Manager project flow.

This option is supported only on Windows NT.

Web – Stop

Stops loading the page.

This option is supported only on Windows NT.

Web – Reload

Reloads the current Web page.

This option is supported only on Windows NT.

Web – Add To Favorites

Adds the current page to your list of favorite URL addresses. To edit this list, choose Web – Go to Favorites.

Web – Go to Favorites

Brings up the Web Favorites dialog box, which displays your list of favorite URL addresses. You can choose an external browser or add, remove, or edit URL addresses.

Flows – Board Design

Opens the Board Design Flow for an open project.

Flows – Library Management

Opens the Library Management Flow for an open project.

Flows – Programmable IC

Opens the Programmable IC flow for an open project.

Allegro Project Manager User Guide
Menu Commands

Project Manager Dialog Box Help

This section explains the following dialog boxes of Project Manager:

- [New Project Wizard](#)
- [Setup Template Dialog Box](#)
- [View Symbol Dialog Box](#)
- [Footprint Dialog Box](#)
- [Import from XML to Design Entry HDL Dialog Box](#)
- [Import from Capture to Design Entry HDL Dialog Box](#)
- [VHDL/Verilog Support Dialog Box](#)
- [Export from Design Entry HDL to XML Dialog Box](#)
- [Export from Design Entry HDL to XML - Select Package Dialog Box](#)
- [Convert from Capture to XML Dialog Box](#)
- [Export from Design Entry HDL to Capture Dialog Box](#)
- [Select Symbols Dialog Box](#)
- [Verify Design Entry HDL Libraries with Template Dialog Box](#)
- [Verify Design Entry HDL Libraries Using Rules CheckerDialog Box](#)
- [Verify Design Entry HDL Libraries in Verilog Simulation Flow Dialog Box](#)
- [Verify Design Entry HDL Libraries in Packaging Flow Dialog Box](#)
- [Project Setup – Global Tab](#)
- [Project Setup – Tools Tab](#)
- [Project Setup – Expansion Tab](#)
- [Project Setup – Views Tab](#)

New Project Wizard

New Project Wizard – Name and Location

This page has two fields, *Name* and *Location*.

Name Enter the name for the project that you are creating. For example `My_Project`. This field can take any number of characters. However, note that you should not enter special characters or spaces. The special characters are changed to `#21` and the spaces are converted to `#20`. For example, if you enter the project name as `My Project`, it is converted to `My#20Project`.

Location Specify the location of your project here.

New Project Wizard – Libraries

This page displays a list of libraries that can be added as reference libraries. The list of libraries is displayed from the Cadence-supplied `cds.lib` file, which is stored in the `$cds_inst_dir/share/library` location.

By default, the displayed libraries are added to the reference library list. To remove any of the displayed libraries from the list of reference libraries, select the library and click *Remove*.

To manage the number of libraries required for reference, use the Add, Import, and Remove buttons.

Add Click this button if you want to add more libraries to your reference libraries list. Clicking on Add displays a dialog box in which you can browse to and select the libraries that you want to add.

Import Click this button if you want to add another library definition file (`cds.lib`) to your reference libraries.

Remove Click this button to delete the selected libraries from the reference library list.

New Project Wizard – Summary

The Summary page displays the details of the project. If you need to change any of the entries, you can go back and incorporate the required changes.

Product Choices Dialog Box

Related Info

This dialog box helps you specify the product suite you want to use in this session. A product suites allows you to access components that are not available in the current product suite.

<i>Select the Project Manager Product</i>	Lists the names of the product suites you can choose from
<i>Use As Default</i>	Helps you specify that the product suite you select must be used by default.
<i>OK</i>	Accepts your selection and starts Project Manager with the specified suite.
<i>Cancel</i>	Cancels the process of starting Project Manager

Related Info

Product Choices

Customize Dialog Box

Use this dialog box to customize the Project Manager display and the tools launched from Project Manager.

<i>Add Tools</i>	Lists the tools that can currently be launched from Project Manager. The tools are displayed in this order on the <i>Tools</i> menu.
<i>Move Up</i>	Moves the selected tool one level up on the <i>Tools</i> menu.
<i>Move Down</i>	Moves the selected tool one level down on the <i>Tools</i> menu.
<i>Add</i>	Brings up the Add Tools dialog box in which you can add a tool to Project Manager.
<i>Edit</i>	Brings up the Edit Tool dialog box in which you can edit the application path, mnemonic, command arguments, and environment variables for the selected tool.
<i>Remove</i>	Removes the selected tool from the <i>Tools</i> menu.
<i>Flow</i>	Displays Project Manager as a flow.
<i>ToolBar</i>	Displays Project Manager as a toolbar.
<i>Large Buttons</i>	Displays large icons when Project Manager is viewed as a toolbar.
<i>Small Buttons</i>	Displays small icons when Project Manager is viewed as a toolbar.
<i>Always on Top</i>	Keeps the Project Manager toolbar on top of any open window. This is supported only on Windows NT.
<i>Project Flow</i>	Enables you to use a custom flow for the project. In the <i>Flow URL</i> box, type the path to the HTML file that defines the flow, or click <i>Browse</i> and select the file.
<i>Reset</i>	Discards your changes and resets the data to the last-saved version.

Allegro Project Manager User Guide

Project Manager Dialog Box Help

<i>OK</i>	Saves your changes and closes the Customize dialog box.
<i>Cancel</i>	Discards your changes and closes the Customize dialog box.

Add Tool Dialog Box

Use this dialog box to add tools to Project Manager for the current project. You can access these tools from the Tools menu. You can also access them when Project Manager is displayed as a toolbar. The new tools are not, however, added automatically to the project flow. To add the tools to the project flow, you must edit the HTML file that defines the flow.

<i>Application Path</i>	Type the full path to the tool's executable file (.exe). Or click <i>Browse</i> , select the executable file, then click <i>Open</i> . The <i>Application Path</i> , <i>Application Name</i> , and <i>Menu Mnemonic</i> boxes are filled in.
<i>Application Name</i>	Type the name of the tool. (If you selected the application path by clicking <i>Browse</i> , this box is already filled in for you.)
<i>Menu Mnemonic</i>	Type the access key you want to use for keyboard access to the tool. (If you selected <i>Application Path</i> by clicking <i>Browse</i> , this box is already filled in for you.)
<i>Command Arguments</i>	Type the command arguments with which you want to start the tool from Project Manager.
<i>Environment Variables</i>	You cannot enter or edit information in this box. To set or change environment variables, click <i>Set Env. Variables</i> .
<i>Maximum Instances</i>	Type or select the number of instances of the tool you want to allow. For example, if you select 3, you can run three sessions of the tool from Project Manager at the same time. Note: If you select the <i>Use Project File</i> option, you cannot select the number of instances. Only one instance is allowed.

*Set Env.
Variables...*

Brings up the Set Environment Variables dialog box in which you can set variables.

Name: Type the variable name.

Value: Type the value of the variable.

Set: Sets the variable you entered.

Remove: Removes the selected variable.

OK: Saves changes and closes the dialog box.

Cancel: Discards changes and closes the dialog box.

Use Project File

Select this box if you want to pass the current project file (*projectname.cpm*) as an argument to the tool when it is started from Project Manager. The tool will be started in the context of the project. Clear this box if you do not want to pass the current project file as an argument to the tool.

Use MPS Args

Select this box if you want to pass MPS arguments to the tool when it is started. Clear this box if you do not want to pass MPS arguments to the tool when it is started.

Browse

Brings up the Choose Executable File dialog box, which you can use to select the executable file of the tool you are adding. When you select a tool using this dialog box, the *Application Path*, *Application Name*, and *Menu Mnemonic* boxes are filled in for you automatically.

OK

Saves your changes and closes the Add Tool dialog box.

Cancel

Discards your changes and closes the Add Tool dialog box.

Setup Template Dialog Box

Procedures

Allegro Project Manager User Guide

Project Manager Dialog Box Help

The Setup Template dialog box helps you create a new template, edit an existing template, or extract a template from a part.

<i>Create a new template</i>	Helps you open the New Template dialog box in which you can enter property, pin, and symbol setup specifications for a new .tpl template and save it.
<i>Edit an existing template</i>	Helps you open the Edit Template dialog box in which you can modify property, pin, and symbol setup specifications for an existing .tpl template.
<i>Extract template from an existing part</i>	Helps you extract a .tpl template from an existing part. When you select this, the Library and Cell fields become active. You need to specify a cell to extract the template from.
<i>Library</i>	Helps you choose from a list of Design Entry HDL libraries.
<i>Cell</i>	Helps you choose from a list of cells within the selected library.
<i>OK</i>	Implements the changes you made.
<i>Close</i>	Closes the dialog box.

Procedures

Setting Up a Template

View Symbol Dialog Box

Procedures

This dialog box helps you specify a symbol that you want to view and then view it in Design Entry HDL.

<i>Build Libraries</i>	Specifies that the symbol you want to view is in a build library.
------------------------	---

<i>Reference Libraries</i>	Specifies that the symbol you want to view is in a reference library.
<i>Library</i>	Displays the list of available build or reference libraries in the <code>cds.lib</code> or <code>refcds.lib</code> file
<i>Cell</i>	Displays the parts contained in the selected library.
<i>View</i>	Displays the available symbols in the selected part.
<i>View</i>	Opens the symbol in Design Entry HDL

Procedures

Viewing a Symbol

Footprint Dialog Box

Procedures

This dialog box helps you specify a footprint and view it in PCB Editor.

<i>Filter</i>	Accepts the initial few characters of the footprint name to narrow down the search.
<i>Footprint</i>	Lists all the footprints in the design. This list is modified by what you type in the <i>Filter</i> field.
<i>View/Verify</i>	Opens PCB Editor and shows the footprint.
<i>PCB Editor Setup</i>	Opens the User Preferences Editor window where you can set or modify the <code>PSMPATH</code> directive. To open PCB Editor and view a footprint in it, you need to specify the <code>PSMPATH</code> directive in the project (<code>.cpm</code>) file.
<i>Close</i>	Closes the dialog box.

Procedures

Viewing and Verifying a Footprint

Import from XML to Design Entry HDL Dialog Box

Procedures

Related Info

This dialog box helps you specify an XML file to import it into a Design Entry HDL library.

<i>File</i>	Helps you specify the absolute path of the XML file by typing the absolute path to the file or by selecting it using the browse button.
<i>Master component alone</i>	Indicates that you want to import only the master component and not the aliases
<i>Enter alias as individual primitive</i>	Indicates that you want to import each alias as an individual primitive
<i>Enter alias as individual cell</i>	Indicates that you want to import each alias as an individual cell
<i>Library</i>	Helps you specify a Design Entry HDL library into which you want to import the XML file.
<i>OK</i>	Closes the dialog box and imports the specified XML file into the library you selected.

Procedures

Importing from XML to Design Entry HDL

Related Info

XML to Design Entry HDL Conversion

Import from Capture to Design Entry HDL Dialog Box

Procedures

This dialog box helps you specify a Capture part from a .olb library file and import it into a Design Entry HDL library.

<i>Library</i>	Helps you specify a library in the <i>Library</i> text box by typing the absolute path to the file or by selecting it using the browse button. The <i>Part</i> drop-down box is populated with names of parts in the selected library.
<i>Part</i>	Helps you specify the part you want to import
<i>Aliases</i>	Displays all aliases of the selected part.
<i>Master component alone</i>	Indicates that you want to import only the master component and not the aliases
<i>Enter alias as individual primitive</i>	Indicates that you want to import each alias as an individual primitive
<i>Enter alias as individual cell</i>	Indicates that you want to import each alias as an individual cell
<i>Library</i>	Helps you specify a Design Entry HDL library into which you want to import the Capture part.
<i>OK</i>	Closes the dialog box and imports the specified XML file into the library you selected.

Procedures

Importing from Capture to Design Entry HDL

VHDL/Verilog Support Dialog Box

Procedures

This dialog box helps you specify an existing or new VHDL or Verilog wrapper or map file for a part to be opened in Part Developer for modification.

<i>Library</i>	Helps you select a Design Entry HDL Library from a list of build libraries in the <code>cds.lib</code> file of the project.
<i>Cell</i>	Helps you select a part from a list of the parts available in the selected Design Entry HDL library.
<i>Verilog Wrapper/ Map File</i>	Helps you specify that the map file you want to open is in Verilog.
<i>VHDL Wrapper/ Map File</i>	Helps you specify that the map file you want to open is in VHDL.
<i>Create new</i>	Helps you specify that you want to create a new map file.
<i>Open existing</i>	Helps you specify that you want to open an existing map file. The browse button alongside helps you specify the file.
<i>OK</i>	Signifies that you have finished making the specifications required in this dialog box. Part Developer opens and shows the wrapper or file.

Procedures

Verifying VHDL/Verilog Support

Export from Design Entry HDL to XML Dialog Box

Procedures

This dialog box helps you specify a Design Entry HDL cell whose package you want to convert into an XML file.

<i>Library</i>	Helps you choose from a list of Design Entry HDL libraries.
<i>Cell</i>	Helps you choose from a list of cells within the selected library.
<i>OK</i>	Takes you to the <u>Export from Design Entry HDL to XML - Select Package Dialog Box</u> in which you can specify more details.

Procedures

Exporting from Design Entry HDL to XML

Export from Design Entry HDL to XML - Select Package Dialog Box

Procedures

This dialog box helps you select a package from the part you selected.

<i>Package</i>	Displays the packages in the cell you selected and lets you select a package. All the symbols in the package are automatically selected.
<i>Directory</i>	Helps you specify the directory into which you want to export the selected part by typing the absolute path to the directory or by selecting it using the browse button.
<i>OK</i>	Exports the specified cell as an XML file into the directory you specified. The name of the XML file is derived from the name of the package you selected.

Procedures

Exporting from Design Entry HDL to XML - Select Package

Convert from Capture to XML Dialog Box

Procedures

Related Info

To import a part from Capture to XML, do the following:

<i>Library</i>	Helps you specify a library in the <i>Library</i> text box by typing the absolute path to the file or by selecting it using the browse button. The <i>Part</i> drop-down box is populated with names of parts in the selected library.
<i>Part</i>	Helps you specify the part you want to import
<i>Aliases</i>	Displays all aliases of the selected part.
<i>XML</i>	Helps you specify the directory into which you want to export the selected part by typing the absolute path to the directory or by selecting it using the browse button.
<i>OK</i>	Exports the specified part as an XML file into the directory you specified. The name of the XML file is derived from the name of the part you selected.

Procedures

Converting from Capture to XML

Related Info

Capture to XML Conversion

Export from Design Entry HDL to Capture Dialog Box

Procedures

This dialog box helps you specify a Design Entry HDL cell that you want to export to Capture.

<i>Library</i>	Helps you choose from a list of the libraries in the <code>cds.lib</code> file of the project.
<i>Cell</i>	Helps you select a cell you want to save as a Capture part from a list of cells within the selected library.
<i>OK</i>	Takes you to the <u>Select Symbols Dialog Box</u> dialog box in which you can specify more details.

Procedures

- Exporting from Design Entry HDL to Capture
- Selecting Symbols

Select Symbols Dialog Box

Procedures

To select symbols, do the following:

<i>Package</i>	Displays the names of packages in the cell you selected and lets you select a package.
<i>Symbols</i>	Displays groups of symbols for the selected package. Select symbols by clicking the boxes to their left. You can select symbols from one of two groups at a time.

<i>Use Pin Name to write Capture Port Name</i>	Helps you specify that you want pin names rather than pin text to be used for the selected cell in Capture. If you do not check this check box, Capture will use pin text for each alias.
<i>Library</i>	Helps you specify the Capture library into which you want to export the selected cell in the <i>Library</i> text box by typing the absolute path to the file or by selecting it using the browse button.
<i>OK</i>	Exports the specified cell into the Capture library you specified.

Procedures

- [Exporting from Design Entry HDL to Capture](#)
- [Converting XML to Capture](#)
- [Selecting Symbols](#)

Verify Design Entry HDL Libraries with Template Dialog Box

Procedures

This dialog box helps you specify a part and a template against which to verify it.

<i>Build Libraries</i>	Helps you indicate that you want to run the verification on the build area libraries.
<i>Reference Libraries</i>	Helps you indicate that you want to run the verification on the reference area libraries.
<i>Options</i>	Helps you indicate whether you want to verify symbol information, pin loads, or property values. You need to select at least one of these options.

- | | |
|--------------------------|--|
| <i>Select a Template</i> | Helps you specify a template by typing its name or by selecting it by using the browse button next to this field. |
| <i>OK</i> | Opens the <i>Part Verification with Template</i> box, which shows a report of checks done and verification results obtained. |

Procedures

Verifying Design Entry HDL Libraries with Template

Verify Design Entry HDL Libraries Using Rules CheckerDialog Box

Procedures

This dialog box helps you verify Design Entry HDL libraries or parts using Rules Checker.

- | | |
|----------------------------|--|
| <i>Build Libraries</i> | Helps you indicate that you want to run the verification on the build area libraries. From the library tree structure, select a cell or a library on which to run the verification by clicking in the empty box to its left. |
| <i>Reference Libraries</i> | Helps you indicate that you want to run the verification on the reference area libraries. From the library tree structure, select a cell or a library on which to run the verification by clicking in the empty box to its left. |
| <i>View Verification</i> | Helps you specify if you want to run minimal checks or advanced checks using Rules Checker. |

Minimal Checks

Helps you specify that you want to run the following minimal checks on the selected libraries or cells:

- *Symbol origin is centered*—to check whether the origin always lies within the symbol and whether the symbol outline is at a distance less than the maximum allowed offset from the origin.
- *Tristated pins have Input and Output loads defined*—to check for the presence of pin properties `OUTPUT_LOAD` and `INPUT_LOAD` for every tristate pin. The presence of tristated pins is denoted by the property `OUTPUT_TYPE = TS, TS`.
- *Consistent symbol name in symbol and package file*—to check whether the cell name is the same as `BODY_NAME` in the `chips.prt` file.
- *Mandatory properties present in package file*—check whether the properties `BODY_NAME`, `PART_NAME`, `CLASS`, and `JEDEC TYPE` are present in the packages.
- *Consistent symbol and package in pin list*—check whether the pins are consistent across symbol and package views.

Advanced Checks

Helps you specify that you want to run advanced checks on the selected libraries or cells.

Launch Rules Checker

Starts Rules Checker.

OK

Runs the specified checks are done and displays a verification report.

Save As

Saves the report as a `.rep` file

Procedures

Verifying Design Entry HDL Libraries Using Rules Checker

Verify Design Entry HDL Libraries in Verilog Simulation Flow Dialog Box

Procedures

This dialog box helps you verify Design Entry HDL libraries or parts using the hlibsim utility.

<i>Build Libraries</i>	Helps you indicate that you want to run the verification on the build area libraries. From the library tree structure, select a cell or a library on which to run the verification by clicking in the empty box to its left.
<i>Reference Libraries</i>	Helps you indicate that you want to run the verification on the reference area libraries. From the library tree structure, select a cell or a library on which to run the verification by clicking in the empty box to its left.
<i>Wrappers or Mapfiles</i>	Helps you specify what you want to verify.
<i>Wrapperview(s)/ Mapview(s)</i>	Helps you specify the names of one or more wrapper or mapfile views.
<i>Stop On Netlist</i>	Helps you specify that you want the verification to stop right after netlisting

Options

Helps you specify the following:

- *Default paths*—if the file that contains the paths to Verilog Models and user-defined primitives (UDPs) is named `vlog_model_path.txt` and is in the default path, that is, in the library directory.
- *Specify paths*—to specify the absolute paths to the Verilog Models and user-defined primitives (UDPs). Type the paths in the *Models* and *Udps* text boxes or specify the directory by using the browse buttons provided next to them.
- *Path to Options file*—to specify an options file, if you have created one. It is a `verilogcmd` file that can be passed to the simulator as it is without any further processing. Type the path to this file in the text box or specify the path by using the browse button provided next to it.

OK

Runs the verification process.

You can see the verification details by clicking the *Details* button in the *Verifying* process progress box.

A success or error message box appears and a log file is generated. You can view this file by clicking the *View Log File* button.

Procedures

Verifying Design Entry HDL Libraries in Verilog Simulation Flow

Verify Design Entry HDL Libraries in Packaging Flow Dialog Box

Procedures

Allegro Project Manager User Guide

Project Manager Dialog Box Help

This dialog box helps you verify Design Entry HDL libraries or parts using the hlibftb utility.

Build Libraries Helps you indicate that you want to run the verification on the build area libraries. From the library tree structure, select a cell or a library on which to run the verification by clicking in the empty box to its left.

Reference Libraries Helps you indicate that you want to run the verification on the reference area libraries. From the library tree structure, select a cell or a library on which to run the verification by clicking in the empty box to its left.

Use project ptf files for verification Helps you specify that the project part table files be used in instantiation and packaging. If no part table files are specified in the project file, the cell-level ptf is used by default.

Upto PCB Editor board (netrev) Helps you specify that you want the hlibftb utility to verify the cell or library for the complete Front-to-Back flow.

Generate pass/fail report Helps you specify that you want Part Developer to verify each part separately and you want a separate report for each part. A message box appears with a warning that the verification has to be run on each part separately and that this may take a long time. Click *OK*.

OK Runs the verification process.

You can see the verification details by clicking the *Details* button in the *Verifying* process progress box.

A success or error message box appears and a log file is generated. You can view this file by clicking the *View Log File* button.

Or

If you had selected the *Generate pass/fail report* radio button, a *Verification Results* dialog box appears and log files are generated. You can view a log file by clicking each part and then clicking the *View Log File* button.

Procedures

Verifying Design Entry HDL Libraries in Packaging Flow

Project Setup – Global Tab

Procedures

Use the Global tab to:

- Select libraries for the project.
- Edit the cds.lib file.
- Change the root design for a project.
- Create a new root design for a project.

Project Name Displays the name of the current project.

Project Location Displays the location of the project in the file system.

Library Name Displays the library that contains the current root design (top-level design) of the project. To change the root design, select the library containing the new root design from the list of libraries (only the Project Libraries for the current project are available in this list), and then specify the design in the *Design Name* field.

Design Name Displays the root design (top-level design) of the project. To change the root design for the current project, click *Browse* and select a design. To create a new root design in the library that is currently selected in the *Library Name* field, type the name of the new design.

Browse Displays the list of designs in the library that is currently selected in the *Library Name* field. To change the root design for the current project, select the design from this list, and then click *OK*.

Allegro Project Manager User Guide

Project Manager Dialog Box Help

<i>cds.lib</i>	Displays the <i>cds.lib</i> file for the project. Each project has a <i>cds.lib</i> file created by Project Manager when you create the project. The <i>cds.lib</i> file determines the list of available libraries from which you choose the libraries for your project.
<i>Edit</i>	Opens the <i>cds.lib</i> file for editing in the default text editor. You can add directives to include other libraries, such as your company libraries, to the list of available libraries. You can add libraries directly by specifying their logical names and physical locations, or you can add a file that contains a list of libraries and their locations. See Syntax for adding libraries to <i>cds.lib</i> .
<i>Library</i>	<p><i>Available Libraries</i> are the libraries available for all projects, from which you select the libraries for each project. Available libraries are determined by the directives in the <i>cds.lib</i> file.</p> <p><i>Project Libraries</i> are the libraries you select for your project from the list of <i>Available Libraries</i>.</p>
<i>Add</i>	Adds the selected libraries to the <i>Project Libraries</i> list. (To select more than one library, select a library, then press CTRL and select the other libraries.)
<i>Remove</i>	Removes the selected libraries from the <i>Project Libraries</i> list. (To select more than one library, select a library, then press CTRL and select the other libraries.)
<i>Add All</i>	Adds all libraries in the <i>Available Libraries</i> list to the <i>Project Libraries</i> list.
<i>Remove All</i>	Removes all the libraries from the <i>Project Libraries</i> list.
<i>Up</i>	Moves up the selected library in the <i>Project Libraries</i> list. The order in which the libraries are listed in the <i>Project Libraries</i> list determines their search order.
<i>Down</i>	Moves down the selected library in the <i>Project Libraries</i> list. The order in which the libraries are listed in the <i>Project Libraries</i> list determines their search order.
<i>View</i>	Displays the details of the selected project library in a tree form.
<i>OK</i>	Saves your changes and closes Project Setup.

Allegro Project Manager User Guide

Project Manager Dialog Box Help

<i>Cancel</i>	Cancels your changes and closes Project Setup.
<i>Reset</i>	Discards your changes and resets the data to the last-saved version.
<i>Apply</i>	Saves your changes.

Procedures

- Changing the Root Design for a Project
- Creating a New Root Design for a Project
- Selecting Libraries for a Project

Project Setup – Part Table Tab

Procedures

Use the Part Table tab to add or remove Physical Part Table (.ptf) files from a project. You can either add .ptf files directly or add directories that contain .ptf files. For example, if the lstd directory contains the lstdl.ptf file, you can add either the complete path to the lstdl.ptf file or just the path to the lstd directory. When you add a directory, all the .ptf files in that directory are added to the project. You can then exclude the ones you do not want.

<i>Physical Part Table Files</i>	Displays the .ptf files (and directories containing .ptf files) currently selected for this project.
<i>Add</i>	<p>Brings up the Add Physical Part Table dialog box, which you use to add .ptf files or directories that contain .ptf files.</p> <p>To add a .ptf file, type its name and path in the <i>Enter Physical Part Table File (.ptf) or Directory to add</i> field, or click <i>File</i> and use the file browser to select the file. (If you type the path to more than one file, separate each path with a space; to select more than one file with the file browser, select the first file, then press CTRL and select the other files.)</p> <p>To add a directory, type its name and path in the <i>Enter Physical Part Table File (.ptf) or Directory to add</i> field, or click <i>Directory</i> and use the file browser to select the directory. (If you type the path to more than one directory, separate each path with a space.)</p> <p>When you add a directory, all the .ptf files contained in it are included in your project, unless you exclude some files with the <i>Exclude Physical Part Table Files</i> option.</p>
<i>Remove</i>	Removes the selected file or directory from the list of Physical Part Table files for the current project.
<i>Exclude Physical Part Table Files</i>	Use this option if your Physical Part Table Files list includes directories that contain several .ptf files and you want to exclude some of the files from your project. Use either this option to exclude files or the <i>Include Physical Part Table Files</i> option to include the files you want.

Allegro Project Manager User Guide

Project Manager Dialog Box Help

<i>Add</i>	<p>Brings up the Exclude Physical Part Table dialog box.</p> <p>Type the name of the Part Table file you want to exclude, then click <i>OK</i>. You can enter more than one file name; separate each name with a space. The files are added to the list of excluded files.</p>
<i>Remove</i>	<p>Removes the selected file from the Exclude Physical Part Table Files list.</p>
<i>Include Physical Part Table Files</i>	<p>Use this option if your Physical Part Table Files list includes directories that contain several .ptf files and you want to choose some files from them. Use either this option to include files or the Exclude Physical Part Table option to exclude unwanted files.</p>
<i>Add</i>	<p>Brings up the Include Physical Part Table dialog box.</p> <p>Type the name of the Part Table file you want to include, then click <i>OK</i>. You can enter more than one file name; separate each name with a space. The files are added to the list of included files.</p>
<i>Remove</i>	<p>Removes the selected file from the Include Physical Part Table Files list.</p>
<i>Use Cell Level Physical Part Table File</i>	<p>Select this check box to include cell-level .ptf files for all the cells in the project. All the .ptf files contained in the Part Table view of the cells will be read by Packager-XL.</p>
<i>Merge Physical Part Table Files</i>	<p>Select this check box to merge the information in all included Physical Part Table files.</p> <p>If this is not selected, and there is more than one file in the Physical Part Table Files field that has entries for the same components, the entry in the last file is picked up.</p> <p>If this is not selected, and the Use Cell Level Physical Part Table Files check box is selected, the .ptf files at the cell level are picked up.</p>
<i>Perform Case-Sensitive Row Match</i>	<p>Select this check box to perform a case-sensitive match of key properties for a part in the Physical Part Table files.</p>

<i>OK</i>	Saves your changes and closes Project Setup.
<i>Cancel</i>	Cancels your changes and closes Project Setup.
<i>Reset</i>	Discards your changes and resets the data to the last-saved version.
<i>Apply</i>	Saves your changes.

Procedures

Adding Physical Part Table Files to a Project

Project Setup – Tools Tab

Procedures

Use the Tools tab to:

- Specify setup directives for PCB Editor, Design Entry HDL, Project Manager, Packager-XL, Programmable IC, Simulation, and Mixed Signal Simulation.
- Choose a default text editor for all Cadence tools.
- Specify a directory for applications to store temporary files.
- Select a property file.
- Specify a log file for the project.

Design Entry HDL Setup Brings up the Design Entry Options dialog box in which you can select default options for Design Entry HDL.

Project Manager Setup Brings up the Project Setup dialog box in which you can customize the Project Manager display, add or remove tools from the *Tools* menu, and select a different project flow.

Packager-XL Setup Brings up the Project Setup dialog box in which you can select the default options for Packager-XL.

<i>Simulation Setup</i>	Brings up the Choose Simulator dialog box, where you can specify the ‘Simulator Type’ and perform advanced simulator-specific setup.
<i>PCB Editor Setup</i>	Brings up the Physical Paths dialog box.
<i>Programmable IC Setup</i>	Brings up the Select Vendor dialog box, where you can specify the vendor, and set up the options for the vendor.
<i>Mixed Signal</i>	Brings up the Mixed WorkBench Stopping Views dialog box, from where you can specify the stopping views for mixed-signal simulation. It indicates that the views are at the leaf level and that the hierarchy below the views are not to be expanded. The order of the views specified determines the view that will be used for netlisting.
<i>Default Text Editor Path</i>	<p>Displays the default editor for editing text files from all Cadence tools. Type the full path to the editor’s executable file, or click <i>Browse</i> and select the file. If you do not specify a text editor, Cadence tools will use WordPad on Windows NT as the default text editor.</p> <p>Note: You can set the default text editor for Design Entry HDL by setting the CDS_TEXT_EDITOR environment variable. To do this, use the following command:</p> <pre>setenv CDS_TEXT_EDITOR<text_editor></pre> <p>If this variable is set, Design Entry HDL will not use the text editor specified in the <i>Tools</i> tab of Project Setup. You must unset this variable if you want Design Entry HDL to use the text editor setting in the <i>Tools</i> tab of Project Setup.</p>
<i>Temp Directory</i>	<p>Displays the directory in which Cadence applications such as Design Entry HDL store temporary files. Type the path to the directory, or click <i>Browse</i> and select it.</p> <p>If you do not specify a directory, Cadence tools will use the <i>temp</i> directory, created automatically in your project directory when you created the project, to store temporary files.</p>

<i>Property File</i>	Displays the name and path of the property file for the project. Type the path to the property file you want to use or click <i>Browse</i> and select it. If you do not specify a property file, the default Cadence <i>cdsprop.paf</i> file is used.
<i>Project Log File</i>	<p>Type the name of the file you want to use as the log file for the project. This file will be created in the project directory. To choose an existing file, click <i>Browse</i> and select it.</p> <p>The log file tracks information such as the date and time, user name, tools launched, and MPS sessions and MPS hosts. If you do not specify a log file, a log will not be generated for the project.</p>
<i>Physical Directory</i>	<p>Type the name of the folder that contains PCB Editor board files. To choose an existing folder, click <i>Browse</i> and select it.</p> <p>If you leave this field blank, the default physical view of the project is selected.</p>
<i>OK</i>	Saves your changes and closes Project Setup.
<i>Cancel</i>	Cancels your changes and closes Project Setup.
<i>Reset</i>	Discards your changes and resets the data to the last-saved version.
<i>Apply</i>	Saves your changes.

Procedures

- [Setting Up Tools](#)
- [Specifying the Application Temp Directory](#)
- [Specifying the Application Temp Directory](#)
- [Selecting a Text Editor](#)
- [Selecting a Property File](#)
- [Setting Up a Log File](#)

Project Setup – Expansion Tab

Procedures

Use the Expansion tab to do the following:

- Select the expansion type for your design.
- Select the configuration for each expansion type.
- Edit a configuration view list.

<i>Design Location</i>	This displays the location of your design in the file system.
<i>Physical Layout</i>	Select this option if you want to expand your design for physical layout with tools such as PCB Editor. This expansion type is selected by default.
<i>View</i>	<p>Type the name of the configuration that you want to expand for physical layout. Or click <i>Browse</i> and select the configuration from the <i>Select View</i> list, which displays all the views available for the cell.</p> <p>To create a new configuration, type its name in the <i>View</i> field. A new configuration view is generated in the design directory when you save the details, and a new configuration file (called <i>expand.cfg</i>) is created in the configuration view. The configuration file has the default view list for Physical Layout expansion.</p> <p>If you do not select or create a configuration, the default configuration for Physical Layout is <i>cfg_package</i>, which is created automatically when you create a new project.</p>
<i>Edit</i>	This brings up the Hierarchy Editor, for you to view or edit the configuration you specified in the <i>View</i> field.
<i>Verilog Simulation</i>	Select this option if you have a Verilog design and want to expand it for simulation with a Verilog simulator such as Verilog-XL.

<i>View</i>	<p>Type the name of the configuration that you want to expand for Verilog simulation. Or click <i>Browse</i> and select the configuration from the <i>Select View</i> list, which displays all the views available for the cell.</p> <p>To create a new configuration, type its name in the <i>View</i> field. A new configuration view is generated in the design directory when you save the details, and a new configuration file, <i>expand.cfg</i>, is created in the configuration view. The configuration file has the default view list for Verilog Simulation expansion.</p> <p>If you do not select or create a configuration, the default configuration for Verilog simulation is <i>cfg_verilog</i>, which is created automatically when you create a new project.</p>
<i>Edit</i>	<p>This brings up the Hierarchy Editor, for you to view or edit the configuration you specified in the <i>View</i> field.</p>
<i>VHDL Simulation</i>	<p>Select this option if you have a VHDL design and want to expand it for simulation with a VHDL simulator such as Leapfrog.</p>
<i>View</i>	<p>Type the name of the configuration that you want to expand for VHDL simulation. Or click <i>Browse</i> and select the configuration from the <i>Select View</i> list, which displays all the views available for the cell.</p> <p>To create a new configuration, type its name in the <i>View</i> field. A new configuration view is generated in the design directory when you save the details, and a new configuration file (called <i>expand.cfg</i>) is created in the configuration view. The configuration file has the default view list for VHDL Simulation expansion.</p> <p>If you do not select or create a configuration, the default configuration for VHDL simulation is <i>cfg_vhdl</i>, which is created automatically when you create a new project.</p>
<i>Edit</i>	<p>This brings up the Hierarchy Editor, for you to view or edit the configuration you specified in the <i>View</i> field.</p>

<i>PIC Configuration</i>	Select this option if you have a Programmable IC design and want to expand it for simulation with a PIC simulator such as Verilog-XL.
<i>View</i>	<p>Type the name of the configuration that you want to expand for PIC simulation. Or click <i>Browse</i> and select the configuration from the <i>Select View</i> list, which displays all the views available for the cell.</p> <p>To create a new configuration, type its name in the <i>View</i> field. A new configuration view is generated in the design directory when you save the details, and a new configuration file, <i>expand.cfg</i>, is created in the configuration view. The configuration file has the default view list for PIC Simulation expansion.</p> <p>If you do not select or create a configuration, the default configuration for PIC simulation is <i>cfg_pic</i>, which is created automatically when you create a new project.</p>
<i>Edit</i>	Brings up the Hierarchy Editor for you to view or edit the configuration you specified in the View field.
<i>Mixed Signal</i>	Select this option if you have a Mixed Signal design and want to expand it for simulation with a Mixed Signal simulator such as the Cadence Analog Workbench (AWBHDL).
<i>View</i>	<p>Type the name of the configuration that you want to expand for Mixed Signal simulation. Or click <i>Browse</i> and select the configuration from the <i>Select View</i> list, which displays all the views available for the cell.</p> <p>To create a new configuration, type its name in the <i>View</i> field. A new configuration view is generated in the design directory when you save the details, and a new configuration file, <i>expand.cfg</i>, is created in the configuration view. The configuration file has the default view list for Mixed Signal expansion.</p> <p>If you do not select or create a configuration, the default configuration for Mixed Signal simulation is <i>cfg_mixed</i>, which is created automatically when you create a new project.</p>

<i>Edit</i>	This brings up the Hierarchy Editor for you to view or edit the configuration you specified in the <i>View</i> field.
<i>Continue expansion even if netlisting errors</i>	Check this to package a design even if it has netlisting errors.
<i>OK</i>	This saves your changes and closes Project Setup.
<i>Cancel</i>	This cancels your changes and closes Project Setup.
<i>Reset</i>	This discards your changes and resets the data to the last-saved version.
<i>Apply</i>	This saves your changes.

Procedures

- [Selecting an Expansion Type](#)
- [Selecting the Configuration for Expansion](#)
- [Editing a Configuration](#)
- [Creating a New Configuration View](#)

Project Setup – Views Tab

Procedures

Use the Views tab to specify view names for your designs. A view is the directory that contains all the data for a particular representation of the design. Views are created when you work with your designs. For example, when you create a schematic and save it, a schematic view is created and all the schematic files are stored in it.

<i>Design Location</i>	This displays the location of the design in the file system.
------------------------	--

Allegro Project Manager User Guide

Project Manager Dialog Box Help

View Names:
Packaged

Type the name for the packaged view or click on the drop-down to select from the list of existing view names for the cell (design). All the Packager-XL output files will be stored in this view. The default name for the packaged view is *packaged*.

View Names:
Chips

Type the name for the chips view or click on the drop-down to select from the list of existing view names for the cell (design). The default name for the chips view is *chips*. The *chips.prt* files will be stored in this view.

View Names: Part
Table

Type the name for the Part Table view or click on the drop-down to select from the list of existing view names for the cell (design). The cell-level physical part table files will be stored in this view. The default name for the Part Table view is *part_table*.

View Names:
Physical

Type the name for the physical view or click on the drop-down to select from the list of existing view names for the cell (design). The default name for the physical view is *physical*. The PCB data will be saved in this view.

Root Design
Views: Board
Design

Type the name for the root design view for board design or click on the drop-down to select from the list of existing view names for the cell (design). The default name for the root design view for board design is *sch_1*.

This view will be used for expanding your design for physical layout.

Root Design
Views: PIC Design

Type the name for the root design view for PIC design or click on the drop-down to select from the list of existing view names for the cell (design). The default name for the root design view for PIC design is *sch_1*.

This view will be used for expanding your design for PIC simulation.

*Root Design
Views: Verilog
Simulation*

Type the name for the root design view for Verilog simulation or click on the drop-down to select from the list of existing view names for the cell (design). The default name for the root design view for Verilog simulation is *sim_sch_1*.

This view will be used for expanding your design for Verilog simulation.

*Root Design
Views: VHDL
Simulation*

Type the name for the root design view for VHDL simulation or click on the drop-down to select from the list of existing view names for the cell (design). The default name for the root design view for VHDL simulation is *sim_sch_1*.

This view will be used for expanding your design for VHDL simulation.

OK

This saves your changes and closes Project Setup.

Cancel

This cancels your changes and closes Project Setup.

Reset

This discards your changes and resets the data to the last-saved version.

Apply

This saves your changes.

Procedures

Selecting Views for the Project

Index

A

Adding Libraries to the cds.lib File [91](#)
Application Temp Directory [95](#)

C

cds.lib
 About [90](#)
 Adding Libraries [91](#)
 Editing [90](#)
Changing View Names [101](#)
Configuration [99](#)
 Editing [99](#)
 View [100](#)
Configuration for Expansion Type
 Selecting [99](#)
copy
 project [53](#)
copy project
 contents of a copied project [58](#)
 file contents [58](#)
 preserved views [59](#)
 root design cell [58](#)
 creating a new project [53](#)
 data classification [50](#)
 derived data [52](#)
 essential data [51](#)
 renaming an existing project [57](#)
 use model [52](#)
copy project functionality [50](#)
copying a project [53](#)
copyproject
 command line [53](#)
creating
 design project [23](#)
 library project [27](#)

D

derived data [52](#)
design project
 creating [23](#)

E

essential data [51](#)
Expansion Type
 About [98](#)
 Selecting [98](#)

F

Files Created for Your New Project [20](#)

H

HREFs for Cadence Tools [80](#)

L

Libraries
 Adding to cds.lib file [91](#)
 Selecting for a Project [92](#)
library project
 create [27](#)
Log File
 About [97](#)
 Setting Up [97](#)

P

Part Table File
 About [93](#)
 Adding [93](#)
Project Flows [17](#)
Project Structure [8](#)
Property File [96](#)
Psetup window [94](#)
ptf file [93](#)
 Adding to a Project [93](#)

R

renaming a project [57](#)

Root Design
 Changing [89](#)
 Creating new [89](#)

S

Selecting a Property File [96](#)
Selecting a Text Editor [96](#)
Setting Up a Log File [97](#)
Starting the Project Manager UI [18](#)
Starting Tools from the Project Manager [64](#)

T

Tools
 Setup [94](#)
Tools Setup [94](#)

V

Viewing Project Libraries [72](#)
Viewing Project Settings [69](#)
Viewing Running Tools [74](#)