

Innoveda® eProduct Designer

Interface Guide

R1-2002

Actel Corporation, Sunnyvale, CA 94086

© 2002 by Actel Corporation. All rights reserved.

Printed in the United States of America

Part Number: 5579027-3

Release: June 2002

No part of this document may be copied or reproduced in any form or by any means without prior written consent of Actel.

Actel makes no warranties with respect to this documentation and disclaims any implied warranties of merchantability or fitness for a particular purpose. Information in this document is subject to change without notice. Actel assumes no responsibility for any errors that may appear in this document.

This document contains confidential proprietary information that is not to be disclosed to any unauthorized person without prior written consent of Actel Corporation.

Trademarks

Actel, and the Actel logotype, are registered trademarks of Actel Corporation.

Adobe and Acrobat Reader are registered trademarks of Adobe Systems, Inc.

Concept-HDL is a trademark or registered trademark of Cadence Design Systems, Inc.

UNIX is a registered trademark of X/Open Company Limited.

All other products or brand names mentioned are trademarks or registered trademarks of their respective holders.

Table of Contents

	Introduction	.v
	Document Organization	.v
	Document Assumptions	vi
	Document Conventions	vi
	Your Comments	vii
	Actel Manuals	vii
	Online Help	ix
1	Setup	11
	Software Requirements	11
	Actel Libraries	11
	Migration Libraries	11
	User Setup	12
	Project Setup	15
2	Actel-Innoveda Design Flow	17
	Schematic-Based Design Flow Illustrated	17
	Schematic-Based Design Flow Overview	18
	VHDL Synthesis-Based Design Flow Illustrated	20
	VHDL Synthesis-Based Design Flow Overview	21
3	Actel-Innoveda Design Considerations	25
	Naming Conventions	25
	Adding Pins to the Schematic	25
	Generating a Top-Level Symbol	26
	Buried I/Os	26
	Adding Power and Ground	26
	Sheets and Symbols	26
	Assigning Pins in a Schematic	27
	Adding ACTgen Macros	27
	Adding FPGA Express Blocks	28
	Generating an EDIF Netlist	29
	Generating a Structural VHDL Netlist	29

Table of Contents

Using FPGA Express with SpeedWave 30

Using FPGA Express with ViewSim 31

4 **Simulation Using ViewSim** 33

 Functional Simulation 33

 Timing Simulation 34

 Multichip Simulation 35

5 **Simulation Using SpeedWave** 37

 Behavioral Simulation 37

 Structural Simulation 38

 Timing Simulation 39

A **Product Support** 41

 Actel U.S. Toll-Free Line 41

 Customer Service 41

 Customer Applications Center 42

 Guru Automated Technical Support 42

 Web Site 42

 FTP Site 43

 Contacting the Customer Applications Center 43

 Worldwide Sales Offices 44

Introduction

The *Innoveda*[®] *eProduct Designer Interface Guide* contains information about using the Innoveda eProduct Designer CAE software tools with the Actel Designer Series FPGA development software tools to create designs for Actel devices. Refer to the *Getting Started User's Guide* for additional information about using the Designer series software and the Innoveda documentation for information about using the eProduct Designer software.

Document Organization

The *Innoveda eProduct Designer Interface Guide* contains the following chapters:

Chapter 1 - Setup contains information about setting up the eProduct Designer software for use in creating Actel designs.

Chapter 2 - Design Flow describes the design flow for creating Actel designs using eProduct Designer software and Designer Series software.

Chapter 3 - Actel-Innoveda Design Considerations contains information to assist you in creating Actel designs with eProduct Designer and Designer Series software.

Chapter 4 - Simulation Using ViewSim[®] contains information about simulating Actel designs with ViewSim.

Chapter 5 - Simulation Using SpeedWave[™] contains information about simulating Actel designs with SpeedWave.

Appendix A - Product Support provides information about contacting Actel for customer and technical support.

Document Assumptions

This document assumes the following:

1. You have installed the Designer Series software in the “C:\Actel” directory.
2. You have installed the eProduct Designer software in the “C:\ePD” directory.
3. You are familiar with PCs and Windows operating environments.
4. You are familiar with FPGA architecture and FPGA design software.

Document Conventions

This document uses the following conventions:

Information input by the user follows this format:

keyboard input

The content of a file follows this format:

file contents

The <act_fam> variable represents an Actel device family. To reference an actual family, substitute the name of the Actel device when you see this variable. Available families are act1, act2 (for ACT 2 and 1200XL devices), ACT3, 3200DX, 40MX, 42MX, and 54SX, 54SX-A and eX.

The <vhd_fam> variable represents Compiled VHDL libraries. To reference an actual compiled library, substitute the name of the library (act1, act2 (for ACT 2 and 1200XL devices), ACT3, A3200DX, A40MX, A42MX, A54SX, and eX) when you see this variable. Compiled VHDL libraries must begin with an alpha character.

Your Comments

Actel Corporation strives to produce the highest quality online help and printed documentation. We want to help you learn about our products, so you can get your work done quickly. We welcome your feedback about this guide and our online help. Please send your comments to **documentation@actel.com**.

Actel Manuals

Designer and Libero include printed and online manuals. The online manuals are in PDF format and available from Libero and Designer's Start Menus and on the CD-ROM. From the Start menu choose:

- Programs > Libero 2.2 > Libero 2.2 Documentation.
- Programs > Designer Series > R1-2002 Documentation

Also, the online manuals are in PDF format on the CD-ROM in the "/manuals" directory. These manuals are also installed onto your system when you install the Designer software. To view the online manuals, you must install Adobe® Acrobat Reader® from the CD-ROM.

The Designer Series includes the following manuals, which provide additional information on designing Actel FPGAs:

Getting Started User's Guide. This manual contains information for using the Designer Series Development System software to create designs for, and program, Actel devices.

Designer User's Guide. This manual provides an introduction to the Designer series software as well as an explanation of its tools and features.

PinEdit User's Guide. This guide provides a detailed description of the PinEdit tool in Designer. It includes cross-platform explanations of all the PinEdit features.

ChipEdit User's Guide. This guide provides a detailed description of the ChipEdit tool in Designer. It includes a detailed explanation of the ChipEdit functionality.

Timer User's Guide. This guide provides a detailed description of the Timer tool in Designer. It includes a detailed explanation of the Timer functionality.

SmartPower User's Guide. This guide provides a detailed description of using the SmartPower tool to perform power analysis.

Netlist Viewer User's Guide. This guide provides a detailed description of the Netlist Viewer. Information on using the Netlist Viewer with Timer and ChipEdit to debug your netlist is provided.

A Guide to ACTgen Macros. This Guide provides descriptions of macros that can be generated using the Actel ACTgen Macro Builder software.

Actel HDL Coding Style Guide. This guide provides preferred coding styles for the Actel architecture and information about optimizing your HDL code for Actel devices.

Silicon Expert User's Guide. This guide contains information to assist designers in the use of Actel's Silicon Expert tool.

Cadence[®] Interface Guide. This guide contains information to assist designers in the design of Actel devices using Cadence CAE software and the Designer Series software.

Mentor Graphics[®] Interface Guide. This guide contains information to assist designers in the design of Actel devices using Mentor Graphics CAE software and the Designer Series software.

Synopsys[®] Synthesis Methodology Guide. This guide contains preferred HDL coding styles and information to assist designers in the design of Actel devices using Synopsys CAE software and the Designer Series software.

Innoveda[®] eProduct Designer Interface Guide (Windows). This guide contains information to assist designers in the design of Actel devices using eProduct Designer CAE software and the Designer Series software.

VHDL Vital Simulation Guide. This guide contains information to assist designers in simulating Actel designs using a Vital compliant VHDL simulator.

Verilog Simulation Guide. This guide contains information to assist designers in simulating Actel designs using a Verilog simulator.

Activator and APS Programming System

Installation and User's Guide. This guide contains information about how to program and debug Actel devices, including information about using the Silicon Explorer diagnostic tool for system verification.

Silicon Sculptor User's Guide. This guide contains information about how to program Actel devices using the Silicon Sculptor software and device programmer.

Flash Pro User's Guide. This guide contains information about how to program Actel ProASIC and ProASIC^{PLUS} devices using the Flash Pro software and device programmer.

Silicon Explorer II. This guide contains information about connecting the Silicon Explorer diagnostic tool and using it to perform system verification.

Macro Library Guide. This guide provides descriptions of Actel library elements for Actel device families. Symbols, truth tables, and module count are included for all macros.

ProASIC^{PLUS} Macro Library Guide. This guide provides descriptions of Actel library elements for Actel ProASIC and ProASIC^{PLUS} device families. Symbols, truth tables, and tile usage are included for all macros.

Online Help

The Designer Series software comes with online help. Online help specific to each software tool is available in Libero, Designer, ACTgen, ACTmap, Silicon Expert, Silicon Explorer II, Silicon Sculptor, and APSW.

Setup

This chapter contains information about setting up eProduct Designer to create Actel designs. This includes information about accessing the Actel and migration libraries, setting up eProduct Designer to interface with the Designer Series software, and setting up Actel projects in eProduct Designer. Refer to the Innoveda documentation for additional information about setting up eProduct Designer.

Software Requirements

The information in this guide applies to the Actel Designer Series software release R1-2000 or later and Innoveda eProduct Designer. For specific information about which versions this release supports, go to the Guru automated technical support system on the Actel web site (<http://www.actel.com/guru>) and type the following in the Keyword box:

third party

Actel Libraries

The Actel libraries contain models for each Actel macro in all Actel families for use in Innoveda. The Actel libraries are sufficient for most cases. Refer to “Migration Libraries” below for exceptions to using the Actel libraries.

Migration Libraries

In addition to the Actel libraries, Actel provides a set of migration libraries. These libraries contain macros supported in earlier versions of the Designer Series software and macros needed to retarget designs from a different Actel family. If you are upgrading from a previous version of Designer and you have existing Actel designs, you must use the migration libraries. Actel does not recommend using the migration libraries on new designs.

The eProduct Designer software uses a “libs.lst” file to access the Actel libraries. When you install the Designer Series software, a “migrate.lst” file is automatically installed in the “c:\actel\lib\wv” directory. You must use this “migrate.lst” file as your “libs.lst” file to properly access the migration libraries.

To access the migration libraries, perform these steps:

- 1. (Optional) Create a backup copy of the “libs.lst” in the “c:\ePD\1.0\standard” directory.**
- 2. Delete the “libs.lst” file in the “c:\ePD\1.0\standard” directory.**
- 3. Copy the “migrate.lst” file in the “c:\actel\lib\wv” directory to the “c:\ePD1.0\standard” directory.**
- 4. Rename the “migrate.lst” file as the “libs.lst” file in the “c:\ePD\1.0\standard” directory.**

User Setup

Before creating designs, there are some one-time procedures to perform so that the eProduct Designer and Designer Series software can interface properly. This section describes those procedures.

Note: For ACTgen, make sure that environment variable “ePD_root” is set and “c:\ePD\1.1\wv\2000.2\win32\bin” is added to the path.

Actel EDIF Command

To automatically generate an Actel compatible EDIF netlist, add a custom command to the Tools menu in ViewDraw. Once added, this command appears as the Actel EDIF command. The following steps describe the procedure.

- 1. Invoke ViewDraw.** If you have not already set up a project, the Project Manager Wizard dialog box is displayed. You must set up an Actel project for ViewDraw to open. Go to “Project Setup” on page 15 for the procedure.

2. **Open the Customize Tools Menu dialog box.** Choose the Customize command from the Tools menu.
3. **Add the Actel EDIF command.** Click the User Menu radio button. In the Menu Text box, type “Actel EDIF.” In the Command box, type or use the Browse button to select: “c:\ePD\1.0\wv\1999.2\win32\bin\edifneto.exe.” In the Arguments box, type “-L unit -L hard \$BLOCKNAME.” Click OK. Figure 1-1 shows the configured Customize Tools Menu dialog box.

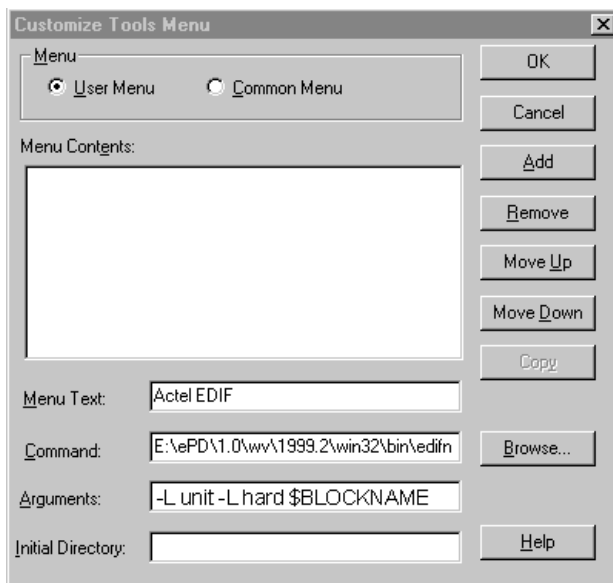


Figure 1-1. Customize Tools Menu Dialog Box

Compiling Actel VITAL Libraries

Before simulating VHDL netlists that reference Actel macros in Speedwave, you must compile Actel VITAL libraries. The following procedures describe the process.

1. **Create a directory called “swave” in the “c:\actel\lib\vtl\95” directory.**

2. **Set your project directory.** Invoke Project Manager and click the New button. Enter “swave” in the Project Name box and “c:\actel\lib\vtl\95\swave” in the Project Directory box. Click the Next button three times, then click the Finish button to complete the process. Do not set your library search order. The New Project Information box appears.
3. **Click the OK button in the New Project Information Dialog Box.**
4. **Save your project and exit Project Manager.**
5. **Open the VHDL Manager.** This displays the HDL Manager dialog box.
6. **Create a <vhd_fam> library.** Choose the Create command from the Library menu. Specify “c:\actel\lib\vtl\95\swave” in the Library Path box and <vhd_fam> in the Symbolic Name box. Click OK.
7. **Add system libraries to search order.** Make sure that the “SYNOPTSYS.LIB” and “IEEE.LIB” libraries are listed under the VHDL System Libraries section in the VHDL View window.
8. **Add the Actel VITAL library to the <vhd_fam> library.** Choose the Add Source Files command from the Library menu. Select the <act_fam>.vhd file from the “c:\actel\lib\vtl\95\” directory. Click OK.
9. **Compile the Actel VITAL library.** Select the <act_fam>.vhd file in the VHDL View section of the HDL Manager window. Choose the Analyze Source File command from the Analyze menu.
10. **(Optional) Add the Actel VITAL Migration library to the <vhd_fam> library.** Select the <act_fam>_mig.vhd file from the “c:\actel\lib\vtl\95\” directory. Click OK.
11. **(Optional) Compile the Actel VITAL Migration library.** Select the <act_fam>_mig.vhd file in the VHDL View section of the HDL Manager window. Choose the Analyze Source File command from the Analyze menu.

Project Setup

You must set up an Actel project in the eProduct Designer Project Manager for each Actel design before creating your design in eProduct Designer. The following procedures describe the process.

1. **Invoke Dashboard.** From the Start menu, select eProduct Designer and Dashboard to open the Dashboard.
2. **Create or Open a project.** Click the New Project wizard. Choose to create or open a project.
3. **Set the Project Directory.** Type the full path name of your design directory in the Project Directory box or use the Browse button. Type the name of your project in the Project Name box. Click the OK button.
4. **Select an Actel FPGA library.** Choose a library in the Configured FPGA Libraries box. Click the icon of the Actel library you want to select, then click Next.
5. **(Optional) Add additional libraries.** Click the Add button and type the full path name of the library you want to add or use the Browse button.
6. **Click the Finish button in the Creating a New Project box.** The New Project information box appears. Click OK.

Creating a Project Library in SpeedWave

If you use SpeedWave to simulate your designs, you must create a project library in SpeedWave, in addition to creating an Actel project in Project Manager, for each VHDL synthesis-based Actel project. The following procedures describe the process.

1. **Open the VHDL Manager from the simulation toolbox.** This displays the HDL Manager window.
2. **Setup a project.** Choose the Create command from the Library menu. The Create Library dialog box is displayed. Make sure the path in the Library Path box is correct and already exists. Type “user” in the Symbolic Name box and click OK. Make sure the Output window reports no errors in the Output window.

- 3. Verify that the “user” library icon appears under the VHDL User Libraries section in the VHDL View window.**
- 4. Add the compiled Actel VITAL library to the Project Libraries.**

Choose the Add to Workspace command from the Library menu. The Add Existing Library to Workspace dialog box is displayed. Click the ellipsis box to open the Select Directory dialog box. Browse to the “c:\actel\lib\vtl\95\swave\<vhd_fam>.lib” directory and click OK. Click OK again in the Add Existing Library to Workspace dialog box. If you have not compiled the Actel VITAL library, go to “Compiling Actel VITAL Libraries” on page 13 for the procedure.

Note: Only add the <vhd_fam>.lib if you have referenced Actel macros in your VHDL netlist.
- 5. Add system libraries to the Project Libraries.** Make sure the “SYNOPTYS.LIB” and “IEEE.LIB” libraries are listed under VHDL System Libraries in the VHDL View window. If they are not present you must reinstall SpeedWave making sure to choose the Synopsys IEEE libraries when prompted.
- 6. Save the project workspace.** Choose the Save command from the File menu.

Actel-Innoveda Design Flow

This chapter describes the design flow for creating Actel designs using the eProduct Designer and Designer Series software.

Schematic-Based Design Flow Illustrated

Figure 2-1 shows the schematic-based design flow for creating an Actel device using the eProduct Designer and Designer Series software¹.

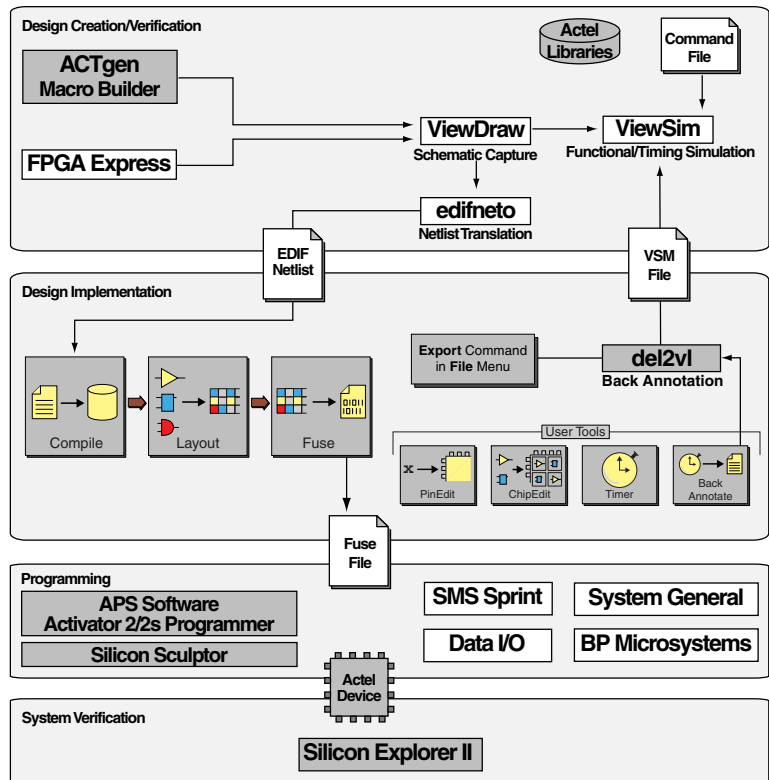


Figure 2-1. Actel-Innoveda Schematic-Based Design Flow

1. The grey boxes in Figure 2-1 denote Actel-specific utilities/tools.

Schematic-Based Design Flow Overview

The Actel-Innoveda schematic-based design flow has four main steps; design creation/verification, design implementation, programming, and system verification. These steps are described in the following sections.

Design Creation/ Verification

During design creation/verification, a schematic representation of a design is captured using the eProduct Designer ViewDraw software. After design capture, you can perform a prelayout (functional) simulation with the ePD ViewSim software. Finally, an EDIF netlist is generated for use in Designer.

Schematic Capture

Enter your schematic in ViewDraw. Refer to chapter 3, “Actel-Innoveda Design Considerations” on page 25 and the Innoveda documentation for information about using ViewDraw.

Functional Simulation

Perform a functional simulation of your design using ViewSim before generating an EDIF netlist for place-and-route. Functional simulation verifies that the logic of the design is correct. Unit delays are used for all gates during functional simulation. Refer to “Functional Simulation” on page 33 and the Innoveda documentation for information about performing functional simulation.

EDIF Netlist Generation

After you have captured and verified your design, you must generate an EDIF netlist for place-and-route in Designer. Refer to “Generating an EDIF Netlist” on page 29 for information about generating an EDIF netlist.

Design Implementation

During design implementation, a design is placed-and-routed using Designer. Additionally, you can perform static-timing analysis on a design in Designer with the Timer tool. After place-and-route, perform postlayout (timing) simulation with the eProduct Designer ViewSim software.

Place-and-Route

Use Designer to place-and-route your design. Make sure you specify INNOVEDA as the Edif Flavor and Generic as the Naming Style when importing the EDIF netlist into Designer. Refer to the *Designer User's Guide* for information about using Designer.

Static-Timing Analysis

Use the Timer tool in Designer to perform static-timing analysis on your design. Refer to the *Timer User's Guide* for information about using Timer.

Timing Simulation

Perform a timing simulation of your design using ViewSim after placing-and-routing it in Designer. Timing simulation requires information extracted and back annotated from Designer. Refer to "Timing Simulation" on page 34 and the Innoveda documentation for information about performing timing simulation.

Programming

Program a device with programming software and hardware from Actel or a supported third-party programming system. Refer to the *Activator and APS Programming System Installation and User's Guide* or *Silicon Sculptor User's Guide* for information about programming an Actel device.

System Verification

You can perform system verification on a programmed device using the Actel Silicon Explorer diagnostic tool. Refer to the *Activator and APS Programming System Installation and User's Guide* or *Silicon Explorer Quick Start* for information about using the Silicon Explorer.

VHDL Synthesis-Based Design Flow Illustrated

Figure 2-2 shows the VHDL synthesis-based design flow for an Actel device using the eProduct Designer and Designer Series software¹.

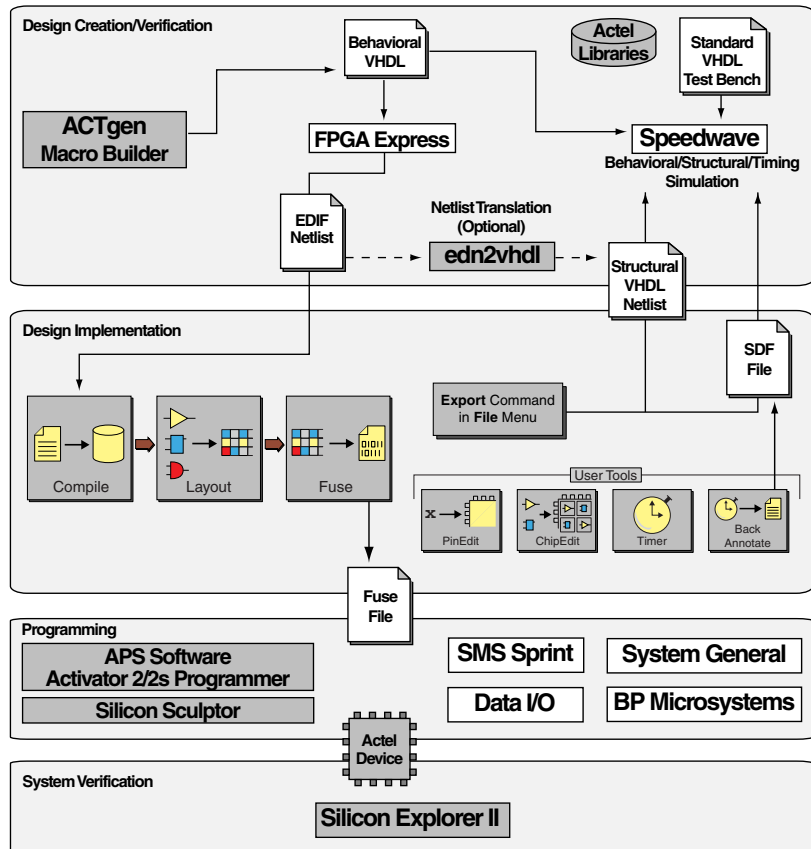


Figure 2-2. Actel-Innoveda VHDL Synthesis-Based Design Flow

1. The grey boxes in Figure 2-2 denote Actel-specific utilities/tools.

VHDL Synthesis-Based Design Flow Overview

The Actel-Innoveda VHDL synthesis-based design flow has four main steps; design creation/verification, design implementation, programming, and system verification. Two verification tools are described, SpeedWave and ViewSim. Use SpeedWave if it is available. It is a VHDL simulator that allows you to perform behavioral, structural, and timing simulation. It also allows you to write stimulus in VHDL. If SpeedWave is not available, use ViewSim for structural and timing simulation. A description of these steps follows.

Design Creation/ Verification

During design creation/verification, a design is captured in an RTL-level (behavioral) VHDL source file. After capture of the design, you may perform a behavioral simulation of the VHDL file with SpeedWave to verify that the VHDL code is correct. The code is then synthesized into a structural EDIF netlist using FPGA Express. After synthesis, you can perform a structural simulation of the design with SpeedWave or ViewSim. Import the EDIF netlist into Designer and perform a timing simulation using SpeedWave or ViewSim.

VHDL Design Source Entry

Enter your design source using a text editor or a context-sensitive VHDL editor. Your VHDL design source can contain RTL-level constructs, as well as instantiations of structural elements, such as ACTgen macros.

Behavioral Simulation

If SpeedWave is available, perform a behavioral simulation of your design before synthesis. Behavioral simulation verifies the functionality of your VHDL code. Typically, unit delays are used and a standard VHDL test bench can be used to drive simulation. Refer to “Behavioral Simulation” on page 37 and the Innoveda documentation for information about performing behavioral simulation.

Synthesis

Synthesize your design using FPGA Express. This transforms the behavioral VHDL file into a gate-level EDIF netlist, optimizing the design for a target technology.

Structural VHDL Netlist Generation

If you use SpeedWave for structural and timing simulation, generate a structural VHDL netlist from your EDIF netlist by either exporting it from Designer or by using the Actel “edn2vhdl” program. Refer to “Generating a Structural VHDL Netlist” on page 29 for information about generating a structural netlist.

Structural Simulation

Perform a structural simulation of your design before placing-and-routing it. Structural simulation verifies the functionality of your postsynthesis structural netlist. Unit delays are used for each gate. Refer to “Functional Simulation” on page 33 if using ViewSim or “Structural Simulation” on page 38 if using Speedwave. Also refer to the Innoveda documentation for information about performing structural simulation.

Design Implementation

During design implementation, use Designer to place-and-route a design. Additionally, you can perform static-timing analysis on a design in Designer with the Timer tool. After place-and-route, perform postlayout (timing) simulation with the Innoveda SpeedWave software.

Place-and-Route

Use Designer to place-and-route your design. Make sure to use GENERIC for the Edif flavor when importing the EDIF netlist into Designer. Refer to the *Designer User’s Guide* for information about using Designer.

Static-Timing Analysis

Use the Timer tool in Designer to perform static-timing analysis on your design. Refer to the *Timer User’s Guide* for information about using Timer.

Timing Simulation

Perform a timing simulation of your design after placing-and-routing it. Timing simulation uses information extracted from Designer, which overrides unit delays in the Actel VHDL. Refer to “Timing Simulation” on page 34 if using Viewsim or “Timing Simulation” on page 39 if

using SpeedWave. Also refer to the Innoveda documentation for information about performing timing simulation.

Programming

Program a device with programming software and hardware from Actel or a supported third-party programming system. Refer to the *Activator and APS Programming System Installation and User's Guide* or *Silicon Sculptor User's Guide* for information about programming an Actel device.

System Verification

You can perform system verification on a programmed device using the Actel Silicon Explorer diagnostic tool. Refer to the *Activator and APS Programming System Installation and User's Guide* or *Silicon Explorer Quick Start* for information about using the Silicon Explorer.

Actel-Innoveda Design Considerations

This chapter contains information to assist in creating Actel designs with the Innoveda eProduct Designer software. Topics include naming conventions, adding pins to the schematic, generating a top-level symbol, buried I/Os, adding power and ground, sheets and symbols, assigning pins in a schematic, adding ACTgen macros, adding FPGA Express blocks, generating an EDIF netlist, generating a structural VHDL netlist, using FPGA Express with SpeedWave, and using FPGA Express with VewSim.

Naming Conventions

Top-level blocks in ViewDraw and FPGA Express must have a design name that follows the DOS file-naming convention (e.g., 8.3 naming convention). The Actel back annotation program can only accept file names that follow this convention.

Use only alphanumeric and underscore “_” characters for schematic net and instance names. Do not use asterisks, forward and backward slashes, spaces, or periods.

Adding Pins to the Schematic

Add pins to the top-level schematic of the design by using the I/O buffer macros with a dangling net attached to the pad, as shown in Figure 3-1. The label on the dangling net becomes the I/O pin name.

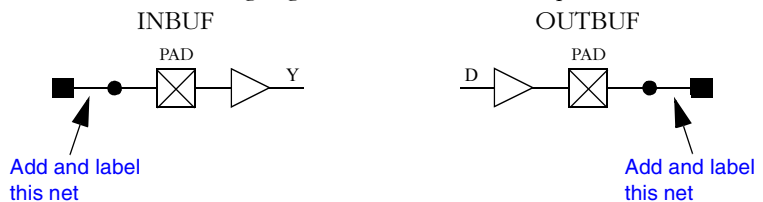


Figure 3-1. Adding Pins to a design

Generating a Top-Level Symbol

When generating a top-level symbol, ViewGen looks for an In or Out port. The convention is illustrated in Figure 3-2. ViewGen does not generate symbols for schematics without IN/OUT ports. The IN/OUT ports are found in the “c:\actel\lib\wv\asicbin” directory.

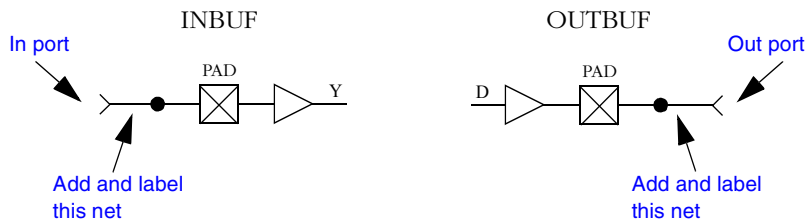


Figure 3-2. Input/Output Ports

Buried I/Os

I/O macros can be buried in the design hierarchy.

Adding Power and Ground

To add power or ground signals in the schematic, use the Actel VCC or GND symbols. You can also label the nets as VDD or GND.

Sheets and Symbols

A multiple-page design is composed of more than one schematic file or <design.n> file in the schematic directory. For a multiple-page design, treat each sheet as part of a top-level schematic and do not consider it a hierarchy level.

Assigning Pins in a Schematic

Nets in your schematic that have the “PIN” attribute assigned to them in ViewDraw are automatically assigned to that pin during design implementation in Designer.

To assign the “PIN” attribute to a net:

- 1. Double-click the net to assign pin information.** This displays the Net Properties dialog box.
- 2. Select the Attributes Tab.**
- 3. Assign the “PIN” attribute to the net.** Type “PIN” in the Name box and the pin number to be assigned in the Value box. Click OK.

Note: This procedure assigns the pin name to the signal net in the netlist. If you use this method to fix pins and you change your pin assignments in PinEdit, you will not be able to back annotate. PinEdit does not change netlist information.

Adding ACTgen Macros

The ACTgen Macro Builder can automatically generate symbols that you can add to your schematic. The following steps describe the procedure.

- 1. Invoke ACTgen.**
- 2. Select the family, macro type, and macro options.**
- 3. Generate your macro as a Innoveda symbol.** Make sure that you specify Innoveda as the Netlist/CAE Formats in the Generate Macro dialog box.
- 4. Add the macro as a component in the schematic.** Refer to the Innoveda documentation for information about adding components to a schematic.

Refer to the *Using Designer Guide*, the *Guide to ACTgen Macros*, or the ACTgen online help for additional information about using ACTgen.

Adding FPGA Express Blocks

FPGA Express can generate blocks that can be added to your ViewDraw schematic. The following steps describe the procedure.

- 1. Invoke FPGA Express.**
- 2. Create a new project or open an existing one.** Add your VHDL source file(s) to the project.
- 3. Create an implementation.** In the Create Implementation form, select “Do not insert I/O pads.”
- 4. Export the Netlist.** In the Export form, select “%s<%d:%d>” for Bus Style and NONE for the Simulation Output Format.
- 5. Save the Project and exit FPGA Express.**
- 6. Translate the EDIF file into ViewDraw’s wir format.** Invoke the EDIF Interfaces program and select the EDIF Netlist Reader tab. Specify the EDIF file created by FPGA Express as the Input and your Innoveda project directory as your Output Dir.
- 7. Invoke ViewGen to generate a symbol.** Specify the .wir file generated by the EDIF Netlist Reader as the input. Select “Generate the top level symbol” and optionally, “Generate schematic.”
- 8. Add the symbol to your ViewDraw schematic.** Refer to the Innoveda documentation for information about adding symbols to a schematic.

Refer to the FPGA Express and other eProduct Designer online help for additional information.

Generating an EDIF Netlist

This section describes the procedures for generating an EDIF netlist for your design. Use the EDIF netlist for place-and-route in Designer.

To generate an EDIF netlist from a schematic-based design:

Select the Actel EDIF command from the Tools menu of ViewDraw. If you have not added the Actel EDIF command to the tools menu in ViewDraw, go to “Actel EDIF Command” on page 12 for the procedure.

To generate an EDIF netlist from a synthesis-based design:

FPGA Express creates an EDIF file that you can import directly into Designer. No special procedure is required.

Generating a Structural VHDL Netlist

You can generate a structural VHDL netlist for SpeedWave simulation from your EDIF netlist by either exporting it from Designer or by using the “edn2vhdl” program. The structural VHDL netlist generated by Designer and the “edn2vhdl” use std_logic for all ports. The bus ports are in the same bit order as they appear in the EDIF netlist.

To generate a structural VHDL netlist using Designer, perform these steps:

- 1. Invoke Designer.**
- 2. Import the EDIF netlist.** Select the Import Netlist File command from the File menu. The Import Netlist dialog box is displayed. Specify EDIF as the Netlist Type, GENERIC (or INNOVEDA if you are using an Innoveda synthesis tool) as the Edif Flavor. Type the full path name of your EDIF netlist or use the Browse button to select your design. Click OK.
- 3. Export the structural VHDL netlist.** Select the Export command from the File menu. The Export dialog box is displayed. Specify Netlist File as the File Type and VHDL as the Format. Click OK.

To generate a structural VHDL netlist using edn2vhdl, perform these steps:

1. **Open a DOS window.**
2. **Change to the directory that contains the EDIF netlist.**
3. **Translate the EDIF netlist to a structural VHDL Netlist.** Type the following command at the prompt:

```
edn2vhdl fam:<act_fam> <design_name>
```

Using FPGA Express with SpeedWave

When using FPGA Express with SpeedWave, you do not need to use ViewDraw, ViewSim, ViewGen, or Innoveda EDIF interfaces. You use three point tools: SpeedWave, FPGA Express, and Designer.

1. **Synthesize your design in FPGA Express.** Create or open an existing FPGA Express project. Add your VHDL source file(s) to the project.
2. **In the Create Implementation dialog box, be sure that the “Do not insert I/O pads” checkbox is not selected.**
3. **In the Export dialog box, select “%s<%d:%d>” for Bus Style and NONE for the Simulation Output Format.** An EDIF netlist is exported by FPGA Express.

Using FPGA Express with ViewSim

When using FPGA Express with ViewSim, you cannot perform a behavioral simulation. You need to import the synthesized design into the ViewDraw/ViewSim environment.

- 1. Synthesize your design in FPGA Express.** Create or open an existing FPGA Express project. Add your VHDL source file(s) to the project.
- 2. In the Create Implementation form, make sure that the “Do not insert I/O pads” checkbox is not selected.**
- 3. In the Export dialog box, select “%s<%d:%d>” for Bus Style and NONE for the Simulation Output Format.** FPGA Express exports an EDIF netlist.
- 4. Import the synthesized EDIF into the ViewDraw/ViewSim environment.** Invoke the EDIF Interfaces program and select the EDIF Netlist Reader tab. Specify the EDIF file created by FPGA Express as the Input and your Innoveda project directory as your Output Dir.

Simulation Using ViewSim

This chapter describes the procedures for performing functional and timing simulations of an Actel design using the Innoveda ViewSim simulation tool.

Functional Simulation

Use the following procedure to perform a functional simulation of an Actel design:

- 1. Select your Actel project in the Dashboard.** If you have not created or setup your project, go to “Project Setup” on page 15 for the procedure.
- 2. Open the ViewSim Wirelister dialog box.** Invoke ViewVSM or, from ViewDraw, choose the Create Digital Netlist command from the Tools menu.
- 3. Generate a simulation (.vsm) wirelist.** Type in the design name or use the Browse button. Click OK. A simulation wirelist is generated and the eProduct Designer window is displayed.
- 4. Simulate the design.** Invoke ViewSim. Type in the design name in the Design Name box and click OK.

Refer to the Innoveda documentation for additional information about performing simulation with ViewSim.

Timing Simulation

Use the following procedure to perform a timing simulation of an Actel design:

- 1. Place-and-route your design in Designer.** Refer to the Using Designer Guide for information on using Designer.
- 2. Extract timing information for your design.** Choose the Export command from the File menu or click the BackAnnotate button. The Extract dialog box is displayed. Create a <design_name>.stf file by choosing the GENERIC option from the CAE pull-down menu. Click OK.
- 3. Back annotate your delays.** Select the Innoveda back-annotate icon from the Designer Series group. The Open dialog box is displayed. Select the <design_name>.stf file and click OK to generate the <design_name>.dtb file and the <design_name>.vsm file for use with ViewSim.
- 4. Select your Actel project in Project Manager.** If you have not created or setup your project see “Project Setup” on page 15.
- 5. Simulate the design.** Invoke ViewSim. Choose the Load Design from the File menu. The Load compiled VSM file dialog box is displayed. Select the <design_name>.vsm file created in step 3 and click OK.

Refer to the Innoveda documentation for additional information about performing simulation with ViewSim.

Multichip Simulation

System designs are typically divided into functional modules implemented by several Actel devices. To check the functionality of the system, you must simulate all Actel devices together. You can use ViewSim and Designer to perform multichip simulation. Use the following procedure to perform a multichip simulation of an Actel design:

Note: Because the viewdraw.ini file uses the same alias for all Actel families, you can only simulate multiple Actel devices of the same family.

- 1. Create a top-level schematic and instantiate the individual chip designs.** This example assumes there are three designs with instance names “chip1,” “chip2,” and “chip3.” The name of the top-level schematic is “top.” Figure 4-1 depicts the directory structure for this example. Names written in normal text represent file names and those in bold text represent directory names.

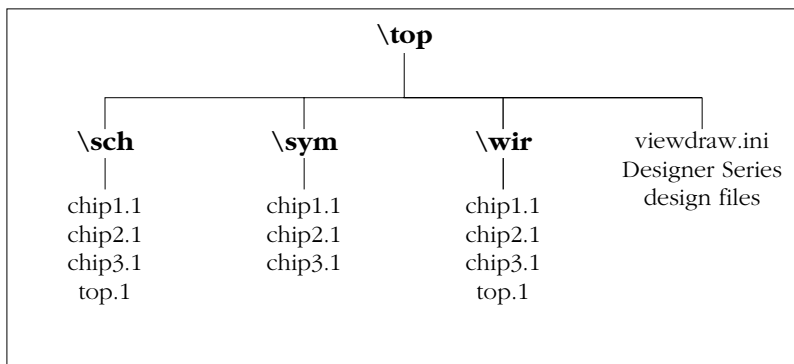


Figure 4-1. Directory Structure for Multichip Simulation

Note: This example only contains single-sheet schematics for each design. Similar procedures apply to multiple-sheet designs.

- 2. Place-and-route your design in Designer.** Refer to the Using Designer Guide for information about using Designer.

3. **Extract timing information for your design.** Choose the Extract command from the Tools menu or click the Extract button. The Extract dialog box is displayed. Create a “chip1.stf” file by choosing the GENERIC option from the CAE pull-down menu. Click OK. Repeat for “chip2.stf” and “chip3.stf.”
4. **Backannotate your delays.** Select the Innoveda back-annotate icon from the Designer Series group. The Open dialog box is displayed. Select the “chip1.stf” file and click OK to generate the “chip1.dtb” file. Repeat for the “chip2.dtb” and “chip3.dtb” files.
5. **Generate a “top.dtb” file for the top-level schematic.** The top-level DTB file includes the following lines:

```
.ba
c chip1
a dtb=chip1.dtb
c chip2
a dtb=chip2.dtb
c chip3
a dtb=chip3.dtb
.ab
```

The “c” line above specifies an instance name, “chip1.” If you have not labeled an instance, use the default handle name of an instance, “\$I138” as it appears in your top-level schematic. Also, the individual DTB files reside in the top-level design directory, “top.”

6. **Run ViewVSM on “top.dtb.”** Reference the “top.dtb” file in the VSM pop-up dialog box. The VSM program processes the DTB files for each chip and creates the “top.vsm” file with back-annotated postlayout timing delays.
7. **Simulate “top.vsm.”** Invoke ViewSim. Type “top.vsm” in the Design Name box and click OK.

Refer to the Innoveda documentation for additional information about performing simulation with ViewSim.

Simulation Using SpeedWave

This chapter describes the procedures for performing behavioral, structural, and timing simulations of an Actel design using the Innoveda SpeedWave simulation tool.

Behavioral Simulation

Use the following procedures to perform a behavioral simulation of an Actel design. Refer to the InnovedaInnoveda documentation for additional information about performing simulation with SpeedWave:

- 1. Select your Actel project in Project Manager.** If you have not created your project, go to “Project Setup” on page 15 for the procedure.
- 2. Open the HDL Manager.** This displays the HDL manager dialog box.
- 3. Open a project HDL workspace (.hws) file.** Choose the Open command from the File menu. Select an .hws file and click the Open button in the Open dialog box.
- 4. Analyze your behavioral VHDL design files and testbench.** Select the user library icon in the VHDL User Libraries section of the VHDL View window. Choose the Add Source Files command from the Library menu. The Assign Source Files dialog box is displayed. Select the behavioral VHDL and testbench files from the VHDL Source File Name window and click OK. Select each file and choose the Analyze Source File command from the Analyze menu. Check the Output window for successful completion. Save the HDL workspace and close the HDL Manager window.

Note: SpeedWave can only simulate configurations. You must have at least one configuration in your testbench.

- 5. Select a configuration to simulate.** Choose the Load Design command from the File menu. Double-click “user.lib” and select the configuration you want to simulate. Click OK. A Source Viewer window and a Hierarchy Viewer window are displayed.
- 6. Simulate your design.** Choose the Run Simulation command from the Simulate menu. The Run Simulation dialog box is displayed.

Select the desired options and click the Apply button. Click the Close button when you have completed your simulation.

Structural Simulation

Use the following procedures to perform a structural simulation of an Actel design. Refer to the Innoveda documentation for additional information about performing simulation with SpeedWave.

1. **Synthesize your design.** Refer to the documentation included with your synthesis tool for information about synthesis.
2. **Select your Actel project in Project Manager.** If you have not created your project, go to “Project Setup” on page 15 for the procedure.
3. **Open the HDL Manager.** This displays the HDL manager dialog box.
4. **Open a project HDL workspace (.hws) file.** Choose the Open command from the File menu. Select an .hws file and click the Open button in the Open dialog box.
5. **Analyze your structural VHDL netlist and testbench.** If you have not already generated a structural VHDL netlist, go to “Generating a Structural VHDL Netlist” on page 29 for the procedure. Select the user library icon in the VHDL User Libraries section of the VHDL View window. Choose the Add Source Files command from the Library menu. The Assign Source Files dialog box is displayed. Select the structural VHDL netlist and testbench files from the VHDL Source File Name window and click OK. Select each file and choose the Analyze Source File command from the Analyze menu. Check the Output window for successful completion. Save the HDL workspace and close the HDL Manager window.

Note: SpeedWave can only simulate configurations. You must have at least one configuration in your testbench.

6. **Select a configuration to simulate.** Choose the Load Design command from the File menu. Double-click “user.lib” and select the

configuration you want to simulate. Click OK. A Source Viewer window and a Hierarchy Viewer window are displayed.

7. **Simulate your design.** Choose the Run Simulation command from the Simulate menu. The Run Simulation dialog box is displayed. Select the desired options and click the Apply button. Click the Close button when you have completed your simulation.

Timing Simulation

Use the following procedures to perform a timing simulation of an Actel design. Refer to the Innoveda documentation for additional information about performing simulation with SpeedWave.

1. **Place-and-route your design in Designer.** Refer to the *Using Designer Guide* for information about using Designer.
2. **Extract timing information for your design from Designer.** Choose the Extract command from the Tools menu or click the Extract button. The Extract dialog is displayed. Create a <design_name>.sdf file by choosing the SDF option from the CAE pull-down menu. Click OK.
3. **Select your Actel project in Project Manager.** If you have not created a project, go to “Project Setup” on page 15 for the procedure.
4. **Open the HDL Manager .** This displays the HDL manager dialog box.
5. **Open a project HDL workspace (.hws) file.** Choose the Open command from the File menu. Select an .hws file and click the Open button in the Open dialog box.
6. **(Optional) Analyze your VHDL design files and testbench.** Skip this step if you are using the same structural VHDL netlist and testbench you analyzed for structural simulation. Select the user library icon in the VHDL User Libraries section of the VHDL View window. Choose the Add Source Files command from the Library menu. The Assign Source Files dialog box is displayed. Select the structural VHDL netlist and testbench files from the VHDL Source File Name window and click OK. Select each file and choose the

Analyze Source File command from the Analyze menu. Check the Output window for successful completion. Save the HDL workspace and close the HDL Manager window.

Note: SpeedWave can only simulate configurations. You must have at least one configuration in your testbench.

- 7. Select a configuration to simulate.** Choose the Load Design command from the File menu. Double-click “user.lib” and select the configuration you want to simulate. Click OK. A Source Viewer window and a Hierarchy Viewer window are displayed.
- 8. Import the timing information for your design.** Click the VITAL Timing index tab. Click the Perform SDF Back Annotation check box and click OK. Click the SDF Back Annotation tab to activate the dialog box. Type the name of your <design_name>.sdf in the SDF File box or use the Browse button.
- 9. Add Scope options.** In the Scope window, fill in the name of the instance in the testbench you want to back annotate (e.g. “/uut”). Select the Delay Mode you want use (Minimum, Typical, or Maximum). Click the Add button and verify that the Scope, File, and Timing settings are correct in the main window. Use the Change and Remove buttons to make any corrections.
- 10. Back annotate your design.** Click OK.
- 11. Simulate your design.** Choose the Run Simulation command from the Simulate menu. The Run Simulation dialog box is displayed. Select the desired options and click the Apply button. Click the Close button when you have completed your simulation.

Product Support

Actel backs its products with various support services including Customer Service, a Customer Applications Center, a web site, an FTP site, electronic mail, and worldwide sales offices. This appendix contains information about contacting Actel and using these support services.

Actel U.S. Toll-Free Line

Use the Actel toll-free line to contact Actel for sales information, technical support, requests for literature about Actel and Actel products, Customer Service, investor information, and using the Action Facts service.

The Actel toll-free line is (888) 99-ACTEL.

Customer Service

Contact Customer Service for nontechnical product support, such as product pricing, product upgrades, update information, order status, and authorization.

From Northeast and North Central U.S.A., call (408) 522-4480.

From Southeast and Southwest U.S.A., call (408) 522-4480.

From South Central U.S.A., call (408) 522-4434.

From Northwest U.S.A., call (408) 522-4434.

From Canada, call (408) 522-4480.

From Europe, call (408) 522-4252 or +44 (0) 1256 305600.

From Japan, call (408) 522-4743.

From the rest of the world, call (408) 522-4743.

Fax, from anywhere in the world (408) 522-8044.

Customer Applications Center

Actel staffs its Customer Applications Center with highly skilled engineers who can help answer your hardware, software, and design questions. The Applications Center spends a great deal of time creating application notes and answers to FAQs. So, before you contact us, please visit our online resources. It is very likely we have already answered your question(s).

Guru Automated Technical Support

Guru is a web-based automated technical support system accessible through the Actel home page (<http://www.actel.com/guru/>). Guru provides answers to technical questions about Actel products. Many answers include diagrams, illustrations, and links to other resources on the Actel web site. Guru is available 24 hours a day, seven days a week.

Web Site

Actel has a World Wide Web home page where you can browse a variety of technical and nontechnical information. Use a Net browser (Netscape recommended) to access Actel's home page.

The URL is <http://www.actel.com>. You are welcome to share the resources provided on the Internet.

Be sure to visit the Technical Documentation area on our web site, which contains information regarding products, technical services, current manuals, and release notes.

You can visit the Product Support area of the Actel website from your Designer software. Click the Product Support button in your Designer Main Window to access the latest datasheets, application notes, and more.

FTP Site

Actel has an anonymous FTP site located at **ftp://ftp.actel.com**. Here you can obtain library updates, software patches, design files, and data sheets.

Contacting the Customer Applications Center

Highly skilled engineers staff the Customer Applications Center from 7:30 A.M. to 5:00 P.M., Pacific Time, Monday through Friday. Several ways of contacting the Center follow:

Electronic Mail

You can communicate your technical questions to our e-mail address and receive answers back by e-mail, fax, or phone. Also, if you have design problems, you can e-mail your design files to receive assistance. We constantly monitor the e-mail account throughout the day. When sending your request to us, please be sure to include your full name, company name, and your contact information for efficient processing of your request.

The technical support e-mail address is **tech@actel.com**.

Telephone

Our Technical Message Center answers all calls. The center retrieves information, such as your name, company name, phone number and your question, and then issues a case number. The Center then forwards the information to a queue where the first available application engineer receives the data and returns your call. The phone hours are from 7:30 A.M. to 5:00 A.M., Pacific Time, Monday through Friday.

The Customer Applications Center number is (800) 262-1060.

European customers can call +44 (0) 1256 305600.

Worldwide Sales Offices

Headquarters

Actel Corporation
955 East Arques Avenue
Sunnyvale, California 94086
Toll Free: 888.99.ACTEL

Tel: 408.739.1010
Fax: 408.739.1540

US Sales Offices

California

Bay Area
Tel: 408.328.2200
Fax: 408.328.2358

Irvine
Tel: 949.727.0470
Fax: 949.727.0476

Newbury Park
Tel: 805.375.5769
Fax: 805.375.5749

Colorado

Tel: 303.420.4335
Fax: 303.420.4336

Florida

Tel: 407.977.6846
Fax: 407.977.6847

Georgia

Tel: 770.277.4980
Fax: 770.277.5896

Illinois

Tel: 847.259.1501
Fax: 847.259.1575

Massachusetts

Tel: 978.244.3800
Fax: 978.244.3820

Minnesota

Tel: 651.917.9116
Fax: 651.917.9114

New Jersey

Tel: 609.517.0304

North Carolina

Tel: 919.654.4529
Fax: 919.674.0055

Pennsylvania

Tel: 215.830.1458
Fax: 215.706.0680

Texas

Tel: 972.235.8944
Fax: 972.235.965

International Sales Offices

Canada

235 Stafford Rd. West,
Suite 106
Nepean, Ontario K2H 9C1

Tel: 613.726.7575
Fax: 613.726.8666

France

Actel Europe S.A.R.L.
361 Avenue General de Gaulle
92147 Clamart Cedex

Tel: +33 (0)1.40.83.11.00
Fax: +33 (0)1.40.94.11.04

Germany

Lohweg 27
85375 Neufahrn
Tel: +49 (0)8165.9584.0
Fax: +49 (0)8165.9584.1

Italy

Via de Garibaldini, No. 5
20019 Settimo Milanese,
Milano, Italy

Japan

EXOS Ebisu Building 4F
1-24-14 Ebisu Shibuya-ku
Tokyo 150

Tel: +81 (0)3.3445.7671
Fax: +81 (0)3.3445.7668

Korea

30th Floor, ASEM Tower,
159-1 Samsung-dong,
Kangam-ku,
Seoul, Korea

Tel: +82.2.6001.3382
Fax: +82.2.6001.3030

United Kingdom

Maxfli Court,
Riverside Way
Camberley,
Surrey GU15 3YL

Tel: +44 (0)1276.401452
Fax: +44 (0)1276.401490

Index

Symbols

<act_fam> variable vi
<vhd_fam> variable vi

A

Actel vi
 Device Families vi
 EDIF command 12
 FPGA Libraries 15
 FTP Site 43
 Web Based Technical Support 42
 Web Site 42
Actel Manuals vii
Actel Project Setup 15–16
 Project Manager 15
 SpeedWave 15
Actel VITAL Libraries
 Compiling 13
 Creating an swave Directory 13
ACTgen, Generating Symbols 27
Adding
 Actel EDIF command 12
 Pins in a Schematic 27
 Pins to a Top-Level Schematic 25
 Power and Ground Symbols 26
 VHDL Blocks 28
Analyzing
 Test Bench 37, 38, 39
 VHDL Source File 37, 38, 39
Arguments
 EDIF Netlist 13
Assigning
 Nets 27
 Pins 27
Assumptions vi

B

Back Annotate 34, 36, 40
 DTB File 34, 36
 Naming Conventions 25
 VSM File 34, 36
Behavioral Simulation 21, 37
 SpeedWave 21, 37
Buried I/O Macros 26

C

Capturing a Design
 Schematic-Based 18
 VHDL-Based 21
Compiling Actel VITAL Libraries 13
Configuration 37, 38, 40
Configuring EDIF Netlist 12
Contacting Actel
 Customer Service 41
 Electronic Mail 43
 Technical Support 42
 Toll-Free 41
 Web Based Technical Support 42
Conventions vi
 <act_fam> variable vi
 <vhd_fam> variable vi
Creating
 Macros 27
 SDF File 39
 STF File 34, 36
 swave Directory 13
Customer Service 41
Customizing ViewDraw 12

D

Dangling Net 25
Delay Mode 40

- Design Creation/Verification 18, 21
 - Behavioral Simulation 21
 - EDIF Netlist Generation 18
 - Functional Simulation 18
 - Schematic Capture 18
 - Structural Netlist Generation 22
 - Structural Simulation 22
 - Synthesis 21
 - VHDL Source Entry 21
- Design Flow
 - Design Creation/Verification 18, 21
 - Design Implementation 18, 22
 - Schematic-Based 18–19
 - VHDL Synthesis-Based 20–23
- Design Implementation 18, 22
 - Place-and-Route 19, 22
 - Timing Analysis 19, 22
 - Timing Simulation 19, 23
- Design Layout 19, 22
- Design Synthesis 21
- Designer
 - EDIF Option 29
 - Extracting Timing Information 34, 36, 39
 - GENERIC Option 34, 36
 - Place-and Route 22
 - Place-and-Route 19
 - SDF Option 39
 - Software Installation Directory vi
 - Timer Tool 19, 22
 - Timing Analysis 19, 22
 - VHDL Option 22
 - VIEWLOGIC Option 22, 29
- Device
 - Debugging 19, 23
 - Families vi
 - Programming 19, 23

- Digital Netlist Generation 33
- Document
 - Assumptions vi
 - Conventions vi
 - Organization v
- DTB file 34, 36

E

- EDIF Netlist Generation
 - Schematic-Based 18, 29
 - Synthesis-Based 29
- EDIF Netlister Arguments 13
- EDIF Option 29
- Electronic Mail 43
- Extracting Timing Information 34, 36, 39

F

- Fixing Pins 27
- FPGA Express
 - Adding Blocks 28
 - Design Synthesis 21
 - Generating Symbols 28
 - Synthesis Considerations 30
 - Use In VHDL Flow 21
- FPGA Libraries 15
- Functional Simulation 18, 33
 - ViewSim 18, 33

G

- Gate-Level Netlist 21
- Generating
 - Digital Netlist 33
 - DTB File 34, 36
 - EDIF Netlist 18
 - Gate-Level Netlist 21
 - Simulation Wirelist 33, 36

Structural Netlist 22
Top-Level Symbol 26
VSM File 33, 34, 36
Generating Symbols
 ACTgen 27
 FPGA Express 28
GENERIC Option 34, 36
GND 26

I

I/O Macros 26
Importing Timing Information 40
In/Out Ports 26
Installation Directory
 Designer vi
 Workview Office vi
Instance Name 36

L

Libraries
 Actel FPGA 15
 Actel VITAL 13
 Project 15

M

Macros 27
Multichip Simulation 35–36
 Directory Structure 35
Multiple-Sheet Schematic 26, 35

N

Netlist Generation
 Digital 33
 EDIF 18
 Gate-Level 21
 Structural 22

O

Online Help ix

P

PinEdit 27
Pins
 Assigning 27
 Attribute 27
 Back Annotate 27
 PinEdit 27
Place-and-Route 19, 22
Post-Synthesis Simulation 23
Primary Directory 15
Product Support 41–44
 Customer Applications Center 42
 Customer Service 41
 Electronic Mail 43
 FTP Site 43
 Technical Support 42
 Toll-Free Line 41
 Web Site 42
Programming a Device 19, 23
Project Manager
 Project Setup 15
 Selecting an Actel FPGA Library 15
 Setting the Primary Directory 15
Project Setup 15–16
 Project Manager 15
 SpeedWave 15

R

Required Software 11

S

Schematic Capture 18
Schematic Design Considerations 25–27

- Adding Pins in a Schematic 27
- Adding Pins to a Top-Level Schematic 25
- Adding Power and Ground 26
- Adding VHDL Blocks 28
- Creating Macros 27
- Generating a Top-Level Symbol 26
- I/O Macros 26
- Multiple-Page Design 26
- Naming Conventions 25
- Schematic Naming Conventions 25
- Schematic-Based Design Flow 18–19
 - Design Creation/Verification 18
 - Design Implementation 18
 - Programming 19
 - System Verification 19
- Scope Options 40
- SDF
 - File 39
 - Option 39
- Selecting
 - An Actel FPGA Library 15
 - Scope Options 40
- Setting the Primary Directory 15
- Setting Up
 - an Actel Project in Project Manager 15
 - an Actel Project in SpeedWave 15
 - SpeedWave 13
 - ViewDraw 12
- Setup Procedures 12–16
 - Adding the Actel EDIF command 12
 - Compiling Actel VITAL Libraries 13
 - Creating a Project Library 15
 - Project Setup 15–16
 - Setting Up an Actel Project 15
 - User Setup 12–14
- Simulation
 - Behavioral 21, 37
 - Configuration 37, 38, 40
 - Functional 18, 33
 - Multichip 35
 - Post-Synthesis 23
 - Schematic-Based 18, 19, 33–36
 - SpeedWave 21, 22, 23, 37–40
 - Structural 22, 33, 38
 - Synthesis-Based 21, 22, 23, 37–40
 - Test Bench 37, 38, 40
 - Timing 19, 23, 34, 39
 - ViewSim 18, 19, 22, 33–36
- Simulation Wirelist, Generating 33, 36
- Software Requirements 11
- SpeedWave
 - Behavioral Simulation 21, 37
 - Compiling Actel VITAL Libraries 13
 - Creating a Project Library 15
 - Importing Timing Information 40
 - Post-Synthesis Simulation 23
 - Scope Options 40
 - Set Up 13
 - Structural Simulation 22, 38
 - Timing Simulation 23, 39
- Static Timing Analysis 19, 22
- STF file 34, 36
- Structural Netlist Generation 22
 - GENERIC Option 29
 - VIEWLOGIC Option 29
- Structural Simulation 22, 33, 38
 - SpeedWave 22, 38
 - VIEWLOGIC Option 29
 - ViewSim 22, 33
- swave Directory 13
- Synthesis 21
- System Requirements 11

System Verification 19, 23
 Silicon Explorer 19, 23

T

Technical Support 42
 Test Bench 37, 38, 39
 Timer
 Static Timing Analysis 19, 22
 Timing Analysis 19, 22
 Timing Information 34, 36, 39, 40
 SDF File 39
 STF File 34, 36
 Timing Simulation 19, 22, 34, 39
 GENERIC Option 34, 36
 SDF
 Option 39
 SpeedWave 23, 39
 ViewSim 19, 34
 Toll-Free Line 41
 Top-Level
 Schematic 25, 35
 Symbol 26

U

Unit Delays 18, 21
 User Setup 12–14

V

variables
 <act_fam> vi
 <vhd_fam> vi
 VCC 26
 VDD 26
 VHDL Option 22
 VHDL Source Entry 21
 VHDL Source File

Analyzing 37, 38, 39
 VHDL Synthesis-Based Design Flow 20–23
 Design Creation/Verification 21
 Design Implementation 22
 Programming 23
 System Verification 23
 ViewDraw
 Alias 35
 Creating an Actel EDIF Netlist 12
 Customizing 12
 Set Up 12
 viewdraw.ini File
 Alias 35
 ViewGen
 Generating Symbols for Schematics 26
 VIEWLOGIC Option 22, 29
 ViewSim
 Adding Pins in a Schematic 27
 Adding Pins to a Top-Level Schematic 25
 Adding Power and Ground 26
 Creating Macros 27
 Functional Simulation 18, 33
 Generating a Top-Level Symbol 26
 I/O Macros 26
 Multichip Simulation 35
 Multiple-Page Design 26
 Naming Conventions 25
 Schematic Design Considerations 25–27
 Structural Simulation 22, 33
 Timing Simulation 19, 34
 ViewVSM, Generating a Simulation Wirelist 33,
 36
 VSM File 33, 34, 36

W

Web Based Technical Support 42

Workview Office, Software Installation Directory
vi