Verilog®

Simulation Guide

R1-2002

Windows® and UNIX® Environments

Actel Corporation, Sunnyvale, CA 94086

© 2002 Actel Corporation. All rights reserved.

Printed in the United States of America

Part Number: 5579005-7

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.

Cadence is a registered trademark of Cadence Design Systems, Inc.

Mentor Graphics is registered trademark of Mentor Graphics, Inc.

Synopsys is a registered trademark of Synopsys, Inc.

Verilog is a registered trademark of Open Verilog International.

Viewlogic, ViewSim, and ViewDraw are registered trademarks and MOTIVE and SpeedWave are trademarks of Viewlogic Systems, Inc.

Windows is a registered trademark and Windows NT is a trademark of Microsoft Corporation in the U.S. and other countries.

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 v
Document Conventions vi
Your Comments vi
Actel Manuals vii
Online Help ix

1 Setup 1
Software Requirements 1
Verilog Libraries 1
Compiling Verilog Libraries 2

2 Design Flow 3
Actel-Verilog Design Flow Illustrated 3
Actel-Verilog Design Flow Overview 4

3 Generating Netlists 7
Generating an EDIF Netlist 7
Generating a Structural Verilog Netlist 7

4 Interpreted Simulation 9
Example Testbench 9
Example Command File 10
Behavioral Simulation 10
Structural Simulation 11
Timing Simulation 12
Verilog Switches 15

5 Simulation with ModelSim 17
Behavioral Simulation 17
Structural Simulation 18
Timing simulation 19

A Product Support 21
Actel U.S. Toll-Free Line 21
Customer Service 21
Customer Applications Center 22
Guru Automated Technical Support 22
Web Site 22
FTP Site 23
Contacting the Customer Applications Center 23
Worldwide Sales Offices 24

Index 25

Introduction

The *Verilog Simulation Guide* contains information about interfacing the Designer Series FPGA development software with Verilog simulation tools. Refer to the *Designer User's Guide* for additional information about using the Designer software. Refer to the documentation included with your Verilog simulation tool for information about performing simulation.

Document Organization

The *Verilog Simulation Guide* contains the following chapters:

Chapter 1 - Setup contains information about setting up Verilog libraries for use in simulating Actel designs.

Chapter 2 - Design Flow illustrates and describes the design flow for simulating Actel designs using Verilog Simulation tools.

Chapter 3 - Generating Netlists contains information for generating EDIF and structural Verilog netlists.

Chapter 4 - Interpreted Simulation describes the procedures for performing functional (behavioral and structural) and timing simulation on an Actel design using tools that either interpret library and design files, or compile them on-the-fly.

Chapter 5- Simulation with ModelSim describes the procedures for performing functional (behavioral and structural) and timing simulation on an Actel design using commands for the ModelSim simulator.

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

Document Assumptions

This document assumes the following:

1. If you are using a PC, you have installed the Designer Series software in the "c:\actel" directory.

2. You have installed a Verilog simulator.
3. You are familiar with UNIX workstations and operating systems.
4. You are familiar with PCs and Windows operating environments.
5. 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 contents of a file follows this format:

```
file contents
```

This document uses the following variables:

- Actel FPGA family libraries are shown as <act_fam>. Substitute the desired Actel FPGA family ACT1, ACT2 (for ACT 2 and 1200XL devices), ACT3, 3200DX, 40MX, 42MX, 54SX, 54SX-A, eX, A500K, APA, and Axcelerator as needed. For example:

  ```
  edn2vlog fam:<act_fam> <design_name>
  ```

- Compiled Verilog libraries are shown as <vlog_fam>. Substitute <vlog_fam> for the desired Verilog family ACT1, ACT2 (for ACT 2 and 1200XL devices), ACT3, A3200DX, A40MX, A42MX, A54SX, A54SX-A, eX, A500K/APA, and Axcelerator as needed. The Verilog language requires that the library names 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.

1

Setup

This chapter contains information about the directory structure of the Verilog libraries and setting up Verilog libraries for use in simulating Actel designs. Refer to the documentation included with your Verilog simulator for information about setting up your simulation tool.

Software Requirements

The information in this guide applies to the Actel Designer Series software release R1-2000 or later and IEEE-1364-compliant Verilog simulators. Additionally, this guides contains information about using the PC and UNIX ModelSim simulators.

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
```

If you are using HP-UX, you must also set the following variable:

```
setenv SHLIB_PATH $ALSDIR/lib
```

Refer to the *Designer User's Guide* and the documentation included with your simulation tool for additional information about setting environment variables.

Verilog Libraries

Migration Libraries

In addition to the standard Actel libraries, Actel provides a set of migration libraries. These libraries contain macros supported in 3.1.1u1 and earlier versions of the Designer Series software and macros possibly needed to retarget designs from a different Actel family. Actel does not recommend using the migration libraries on new designs. The Verilog migration libraries are installed in the "<actel_install_directory>/lib/vlog/<act_fam>_mig" directories.

Note: Verilog migration libraries are not available for A500K and APA devices.

Compiling Verilog Libraries

Before simulating a design with the ModelSim Verilog simulator, you must compile the Actel Verilog libraries. This section describes the procedures. Refer to the documentation included with your simulation tool for additional information about compiling libraries.

ModelSim

Use the following procedure to compile Verilog libraries for the ModelSim simulator. Type UNIX commands at the UNIX prompt. The commands below are for PC. To compile the Verilog libraries:

1. **Create a directory called “mti” in the appropriate directory:**

   “$ALSDIR\lib\vlog”

2. **Invoke the simulator (PC only).**
3. **Change to the “$ALSDIR\lib\vlog\mti” directory.** Type the following command at the prompt:

   ```
   cd $ALSDIR\lib\vlog\mti
   ```

4. **Create an <act_fam> family library directory for the simulator.** Type the following command:

   ```
   vlib <act_fam>
   ```

5. **Compile the Actel library.** Type the command at the prompt:

   ```
   vlog -work <act_fam> $ALSDIR\lib\vlog\<act_fam>.v
   ```

6. **(Optional) Compile the Migration library.** Only perform this step if you are using the migration library. Type the following command:

   ```
   vlog -work <act_fam> $ALSDIR\lib\vlog\<act_fam>_mig.v
   ```

   Note: There are no migration libraries available for A500K and APA devices.

2

Design Flow

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

Actel-Verilog Design Flow Illustrated

Figure 2-1 shows the design flow for an Actel device using CAE software, a Verilog simulation tool, and the Designer Series software[1].

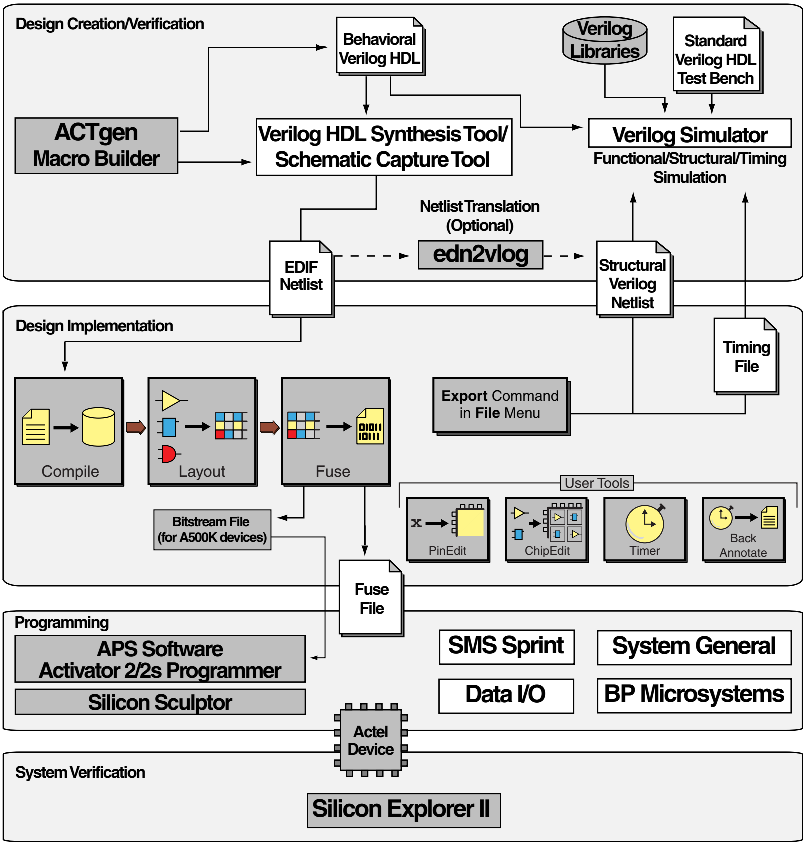
Design Creation/Verification
Behavioral Verilog HDL
Verilog Libraries
Standard Verilog HDL Test Bench
ACTgen Macro Builder
Verilog HDL Synthesis Tool/ Schematic Capture Tool
Verilog Simulator
Functional/Structural/Timing Simulation
Netlist Translation (Optional)
EDIF Netlist
edn2vlog
Structural Verilog Netlist
Design Implementation
Timing File
Compile
Layout
Fuse
Export Command in File Menu
User Tools
PinEdit
ChipEdit
Timer
Back Annotate
Bitstream File (for A500K devices)
Fuse File
Programming
APS Software Activator 2/2s Programmer
Silicon Sculptor
SMS Sprint
System General
Data I/O
BP Microsystems
Actel Device
System Verification
Silicon Explorer II

Figure 2-1. Actel-Verilog Design Flow

---

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

Actel-Verilog Design Flow Overview

The Actel-Verilog design flow has four main steps:

1. Design Creation/Verification
2. Design Implementation
3. Programming
4. System Verification

The following sections describe these steps.

Design Creation/ Verification

During design creation/verification, a design is captured as a schematic or as an RTL-level (behavioral) Verilog HDL source file.

If your design is a Verilog HDL source file, you can perform a behavioral simulation to verify that the HDL code is correct. The code is then synthesized into an Actel gate-level (structural) Verilog HDL netlist. After synthesis, you can perform a structural simulation of the design. Finally, you generate an EDIF netlist for use in Designer and a structural Verilog netlist for structural and timing simulation.

If your design is a schematic, you generate an EDIF netlist for use in Designer and a structural Verilog netlist for structural and timing simulation. You do not perform behavioral simulation or synthesis.

Design Capture

Enter your schematic using a third-party schematic-capture tool or create your Verilog HDL source file using a text editor or a context-sensitive HDL editor. Your Verilog HDL design source can contain RTL-level constructs, as well as instantiations of structural elements, such as ACTgen macros. Refer to the documentation included with your design-capture tool for information about design capture.

Behavioral Simulation

Perform a behavioral simulation of your design before synthesis. Behavioral simulation verifies the functionality of your Verilog HDL code. You can use a standard Verilog HDL testbench to drive simulation. Refer to “Behavioral Simulation” on page 10 and the documentation included with your simulation tool for information about performing functional simulation.

Synthesis

After you have created your Verilog HDL source file, you must synthesize it before placing-and-routing it in Designer. Synthesis transforms the Verilog HDL source file into a gate-level netlist and optimizes the design for a target technology. Refer to the documentation included with your synthesis tool for information about performing design synthesis.

EDIF Netlist Generation

After you have created, synthesized (if your design is an HDL source file), and verified your design, you must generate an EDIF netlist for place-and-route in Designer. If your design is a Verilog HDL source file, use the EDIF netlist to generate a structural Verilog netlist. Refer to "Generating an EDIF Netlist" on page 7 and the documentation included with your schematic-capture or synthesis tool for information about generating an EDIF netlist.

Structural Verilog Netlist Generation

Generate a structural Verilog netlist from your EDIF netlist for use in structural and timing simulation by either exporting it from Designer or by using the Actel "edn2vlog" program. Refer to "Generating a Structural Verilog Netlist" on page 7 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 structural Verilog netlist. Use default unit delays included in the Verilog libraries for every gate. Refer to "Structural Simulation" on page 11 and the documentation included with your simulation tool for information about performing structural simulation.

Design Implementation

During design implementation, you place-and-route a design using Designer. Additionally, you can perfrom static-timing analysis on a design in Designer with the Timer tool. After place-and-route, you can perform postlayout (timing) simulation with a Verilog simulator.

Place-and-Route

Use Designer to place-and-route your design. Make sure to specify Verilog 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 after placing-and-routing it. Timing simulation requires information extracted from Designer, which overrides default unit delays in the Actel Verilog libraries. Refer to "Timing Simulation" on page 12 and the documentation included with your simulation tool 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 *Designer User's Guide* and 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.

3

Generating Netlists

This chapter describes the procedures for generating EDIF and structural Verilog netlists.

Generating an EDIF Netlist

After capturing your schematic or synthesizing your design, generate an EDIF netlist from your schematic-capture or synthesis tool. Use the EDIF netlist for place-and-route in Designer. Refer to the documentation included with your schematic-capture or synthesis tool for information about generating an EDIF netlist.

Make sure to specify Verilog for the naming style when importing the EDIF netlist into Designer.

Generating a Structural Verilog Netlist

You can generate structural Verilog netlist using Designer or the “edn2vlog” program. Use the structural Verilog netlist for structural and timing simulation.

To generate a structural Verilog netlist using Designer,

1. **Invoke Designer.**
2. **Import the EDIF netlist.** Choose the Import Netlist File command from the File menu. The Import Netlist dialog box is displayed. Specify EDIF as the Netlist Type, GENERIC as the Edif flavor, and Verilog as the Naming Style. Type the full path name of your EDIF netlist or use the Browse button to select your design. Click OK.
3. **Export a structural Verilog netlist.** Choose the Export command from the File menu and then choose Netlist. The Export Netlist File dialog box is displayed. In the pull-down menu, click Verilog Files (*.v) and enter the name of the Verilog file you want to save.

To generate a structural netlist using edn2vlog,

1. **Change to the directory that contains the EDIF netlist.**
2. **Type the following command at the UNIX or DOS prompt:**

```
edn2vlog FAM:{<act_fam>}
[ EDNIN:<EdifFile1>{+<EdifFile2...>} ]
[ VLGOUT:<Verilog_File> ]
[ SIMTEMP:<Stimulus_template_file> ]
<design_name>
```

The “EDNIN” option specifies the EDIF input file(s). You can specify multiple files with the “+” delimiter between file names. The default EDIF input file is <design_name>.edn. The “VLOGOUT” option specifies the Verilog output file names. The default Verilog output file is <design_name>.v. The “SIMTEMP” option instructs the program to generate a Verilog stimulus template file. If you do not specify this option, the program does not generate a stimulus file.

4

Interpreted Simulation

This chapter describes the procedures for performing functional (behavioral and structural) and timing simulation on an Actel design by using tools that either interpret library and design files or compile them on-the-fly. Cadence Verilog-XL, Simucad SilosIII, and Synopsys VCS are simulators in this category.

This chapter includes information about creating a testbench and information about creating a command file to run a simulation in batch mode. Also, this chapter includes a description of some common Verilog simulation switches. Refer to the documentation included with your simulation tool for additional information about testbenches, command files, switches, and simulation.

Example Testbench

You can use a testbench to apply test vectors or patterns to a design during simulation to compare input and output patterns. The file can instantiate the top-level design, using a Verilog-predefined command, such as “$readmemb,” “$monitor,” and “$display.” To use the testbench, it must be in the current project directory. The following is an example testbench:

```
'timescale 1ns/100ps
module test;
//Inputs and outputs declaration
wire ...........
reg .................
//Instantiate the top module of your design in the test module
<top_module> <instance_name> (.....Pin List...);
..........
//stimulus patterns
initial
begin
.....
end
endmodule
```

Example Command File

You can use a command file for batch simulation. Your command file should include all command variables and Verilog switches you want to set during simulation. The following is an example command file:

```
<test_bench>.v
<design_name>.v
<verilog_switch_1> ... <verilog_switch_n>
```

The <design_name>.v variable is the Verilog top-level design that includes all design sublevels. This variable can represent a behavioral Verilog design for function simulation or gate-level Verilog design for structural or timing simulation.

The <verilog_switch_1> and <verilog_switch_n> variables represent Verilog switches that you can add to your command line during simulation. Refer to "Verilog Switches" on page 15 for information about available Verilog switches.

Behavioral Simulation

Use the following procedure to perform a behavioral simulation of a design. Refer to the documentation included with your simulation tool for additional information about performing behavioral simulation.

1. **Create or modify the testbench.** Make sure your testbench has a timescale definition added to it. The following is an example timescale definition:

   ```
   'timescale 1ns/100ps
   ```

   Refer to "Example Testbench" on page 9 for information about creating testbenches.

2. **Create or modify the command file.** A command file is only necessary if you are running batch simulation. Refer to "Example Command File" on page 10 for information.

3. **Simulate the design.** If your design is a Verilog HDL design, make sure that you simulate your behavioral Verilog HDL source file. Invoke the Verilog simulator by typing the following command:

   ```
   <verilog_executable> <test_bench>.v <design_name>.v -v
   $ALSDIR/lib/vlog/<act_fam>/<act_fam>.v
   ```

   If you are using the migration libraries, type the following command:

   ```
   <verilog_executable> <test_bench>.v <design_name>.v -v
   $ALSDIR/lib/vlog/<act_fam>.v -v $ALSDIR/lib/vlog/
   <act_fam>_mig.v
   ```

   The “-v” option is a Verilog switch that you can add to your command line during simulation. Refer to “Verilog Switches” on page 15 for information.

To simulate a design using a command file,

invoke the Verilog simulator by typing the following command:

```
<verilog_executable> -f <command_file>
```

The “-f” switch is necessary if you are using a command file to simulate a design.

Structural Simulation

Use the following procedure to perform a structural simulation on a design. Refer to the documentation included with your simulation tool for additional information about performing structural simulation.

1. **Create or modify the testbench.** Make sure your testbench has a timescale definition added to it. The following is an example timescale definition:

   ```
   'timescale 1ns/100ps
   ```

   Refer to “Example Testbench” on page 9 for information about creating testbenches.

2. **Create or modify the command file.** A command file is only necessary if you are running batch simulation. Refer to “Example Command File” on page 10 for information.
3. **Simulate the design.** If your design is a Verilog HDL design, make sure that you simulate the structural Verilog HDL netlist that you generated using Designer or the “edn2vlog” program. Invoke the Verilog simulator by typing the following command:

   ```
   <verilog_executable> <test_bench>.v <design_name>.v -v
   $ALSDIR/lib/vlog/<act_fam>.v
   ```

   If you are using the migrations libraries, type the following command:

   ```
   <verilog_executable> <test_bench>.v <design_name>.v -v
   $ALSDIR/lib/vlog/<act_fam> -v $ALSDIR/lib/vlog/
   <act_fam>_mig.v
   ```

   The “-v” option is a Verilog switch that you can add to your command line during simulation. Refer to “Verilog Switches” on page 15 for information.

To simulate a design using a command file,

invoke the Verilog simulator by typing the following command:

```
<verilog_executable> -f <command_file>
```

The “-f” switch is necessary if you are using a command file to simulate a design.

Timing Simulation

Use the following procedure to perform a timing simulation on a design. Refer to the documentation included with your simulation tool for additional information about performing timing simulation.

1. **Place-and-route your design in Designer.** Refer to the *Designer User’s Guide* for information about placing-and-routing a design using Designer.

2. **Extract timing information for your design from Designer.** From the File menu, click Export. Then, click Timing Files. Choose SDF and click Save (or click Back Annotate). The Back Annotate dialog box is displayed. Create a <design_name>.sdf file by specifying SDF as the CAE type. Click OK.

3. **Create or modify a testbench.** Make sure your testbench has an "sdf_annotate" construct in it. The following is an example testbench with a construct line:

   ```
   'timescale 1ns/100ps
   module test;

   //Inputs and outputs declaration
   wire ...........
   reg .................

   //Instantiate the top module of your design in the test mod-
   ule
   <top_module> <instance_name> (.....Pin List...);
   ..........

   //stimulus patterns
   initial
   begin
   .....
   end

   //Invoke SDF routine to back annotate
   initial
   $sdf_annotate("<design_name>.sdf",<instance_name>);
   endmodule
   ```

   The "<instance_name>" variable is the top-level instance name.

   Refer to "Example Testbench" on page 9 for information about creating testbenches.

4. **Create or modify the command file.** A command file is only necessary if you are running batch simulation. Refer to "Example Command File" on page 10 for information about creating command files.

5. **(VCS Only) Create a PLI table.** The PLI table is a text file that contains PLI commands for VCS. The following is an example PLI

table called “sdf.tab” that uses the module “test” in the testbench example in step 3.

```
$sdf_annotate call=sdf_annotate_call
acc+=tchk,mp,mip,prx:test+
```

Refer to the VCS documentation for information about creating a PLI table.

6. **Simulate the design.** If your design is a Verilog HDL design, make sure that you simulate the structural Verilog HDL netlist that you generated using Designer or the “edn2vlog” program. Invoke the Verilog simulator by typing the following command (for VCS):

```
vcs <test_bench>.v <design_name>.v -v $ALSDIR/lib/vlog/
<act_fam>.v -M -P sdf.tab
```

If you are using the migrations libraries, type the following command:

```
vcs <test_bench>.v <design_name>.v -v $ALSDIR/lib/vlog/
<act_fam>.v -v $ALSDIR/lib/vlog/<act_fam>_mig.v -M -P
sdf.tab
```

For other simulators:

```
<verilog_executable> <test_bench>.v <design_name>.v -v
$ALSDIR/lib/vlog/<act_fam>.v
```

If you are using the migrations libraries, type the following command:

```
<verilog_executable> <test_bench>.v <design_name>.v -v
$ALSDIR/lib/vlog/<act_fam>.v -v $ALSDIR/lib/vlog/
<act_fam>_mig.v
```

The “-v” option is a Verilog switch that you can add to your command line during simulation. Refer to “Verilog Switches” on page 15 for information.

To simulate a design using a command file,

Invoke the Verilog simulator by typing the following command:

```
<verilog_executable> -f <command_file>
```

The “-f” switch is necessary if you are using a command file to simulate a design.

Verilog Switches

This section defines and gives usage examples of some common Verilog switches for simulators that interpret design files or compile the files on-the-fly. Refer to the documentation included with your Verilog simulation tool for additional information about using switches during simulation.

Minus Switches

Table 4-1 defines and gives usage examples of Verilog minus switches.

Table 4-1. Minus Switches

Switch	Definition
-s	Stop option; initiates entry into interactive mode after successful design compilation.
-a	Accelerated option; directs accelerated, declared elements to simulate in accelerated mode (Verilog-XL only).
-c	Compile only option; compiles the text code in a data file and exits the simulation mode.
-d	Decompile option; retargets data files into existing text files.
-f	Command argument file option; reads invocation command from a text file.
-l	Log file option.
-y	Library directory option; specifies a target library directory.
-v	Specifies the library file.

Plus Switches

Table 4-2 defines and gives usage examples of Verilog plus switches.

Table 4-2. Plus Switches

Switch	Definition
+libext+	Used with -y switch.
+delay_mode_path	Specifies the path delay model for simulation.
+delay_mode_unit	Specifies the unit delay model for simulation.
+delay_mode_zero	Functional simulation option; specifies the zero delay model for simulation.
+mindelays	Back-annotation option; selects minimum delay for simulation.
+maxdelays	Back-annotation option; selects maximum delay for simulation.
+typdelays	Back-annotation option; selects typical delay for simulation.

5

Simulation with ModelSim

This chapter describes the procedures for performing functional (behavioral and structural) and timing simulation on an Actel design using commands for the ModelSim simulator. Refer to the documentation included with your simulation tool for information about simulating a design using the graphical user interface.

Behavioral Simulation

Use the following procedure to perform a behavioral simulation of a design using the ModelSim simulator. Type UNIX commands at the UNIX prompt. Type PC commands on the command line of the ModelSim Transcript window. The commands below are for PC. To make the commands work for UNIX, use forward slashes instead of back slashes.

1. **Invoke the simulator (PC only).**
2. **Change the directory to your project directory.** This directory must include your Verilog design files and testbench. Type the following command:

   ```
   cd <project_dir>
   ```

3. **Create a "work" directory.** Type the following command:

   ```
   vlib work

   vmap work ./work
   ```

4. **Compile your design source and testbench file(s).** Before simulating your design, you must compile the source files and testbench. For hierarchical designs, compile the lower-level design blocks before the higher-level design blocks. Type the following commands:

   ```
   vlog <behavioral_design_file>.v
   vlog <test_bench>.v
   ```

5. **Simulate your design.** Type the following command:

```
vsim <topmost_module_name>
```

For example:

```
vsim test_adder_behave
```

The module test_adder_behave in the testbench will be simulated.

If any Actel macros are instantiated in your Verilog source, use the following command to simulate your design with the compiled Actel Verilog library.

```
vsim -L $ALSDIR\lib\vlog\mti\<act_fam> <topmost_module_name>
```

Structural Simulation

Use the following procedure to perform a structural simulation of a design using the ModelSim simulator. Type UNIX commands at the UNIX prompt. Type PC commands on the command line of the ModelSim Transcript window. The commands below are for PC. To make the commands work for UNIX, use forward slashes instead of back slashes.

1. **Invoke the simulator (PC only).**
2. **Change directory to your project directory.** This directory must include your Verilog design files and testbench. Type the following command:

```
cd <project_dir>
```

3. **Create a “work” directory.** You only need to create a work directory if you are using a different project directory than the one you used for behavioral simulation. Type the following command:

```
vlib work
```

4. **Compile the structural netlist and testbench.** If you have not already generated a structural Verilog netlist, go to “Generating a

Structural Verilog Netlist" on page 7 for the procedure. Type the following commands:

```
vlog <structural_netlist>.v
vlog <test_bench>.v
```

5. **Simulate your design.** Type the following commands:

```
vsim -L $ALSDIR\lib\vlog\mti\<act_fam> <topmost_module_name>
```

For example:

```
vsim -L $ALSDIR\lib\vlog\mti\42mx test_adder_structure
```

The module test_adder_structure in the testbench will be simulated using the compiled 42MX Verilog library.

Timing simulation

Use the following procedure to perform a timing simulation of a design using the ModelSim simulator. Type UNIX commands at the UNIX prompt. Type PC commands on the command line of the ModelSim Transcript window. The commands below are for PC. To make the commands work for UNIX, use forward slashes instead of back slashes.

1. **Place-and-route your design in Designer.** Refer to the *Designer User's Guide* for information about placing-and-routing a design using Designer.
2. **Extract timing information for your design from Designer.** Choose the Export command from the File menu or click Back Annotate. 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. **Invoke the simulator (PC only).**
4. **Change the directory to your project directory.** This directory must include your Verilog design files and testbench. Type the following command:

```
cd <project_dir>
```

5. **Create a "work" directory.** You only need to create a work directory if you are using a different project directory than the one you used for behavioral and structural simulation. Type the following command:

```
vlib work
```

6. **Compile the structural netlist and testbench.** If you have not already generated a structural Verilog netlist, go to "Generating a Structural Verilog Netlist" on page 7 for the procedure. Type the following commands:

```
vlog <structural_netlist>.v
vlog <test_bench>.v
```

7. **Simulate your design using timing information contained in the SDF file.** Type the following command:

```
vsim -L $ALSDIR\lib\vlog\mti\<act_fam> -sdf[max|typ|min]
/<region>=<design name>.sdf -c <topmost_module_name>
```

The <region> option specifies the region (or path) to an instance in a design where back annotation begins. You can use it to specify a particular FPGA instance in a larger system design or testbench that you wish to back annotate. For example:

```
vsim -L $ALSDIR\lib\vlog\mti\42mx -sdfmax /uut=adder.sdf -c
test_adder_structural
```

In this example, the module "adder" has been instantiated as instance "uut" in the testbench. The module named "test_adder_structural" in the testbench will be simulated using the maximum delays specified in the SDF file.

A

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

A

- Accessing
  - Migration Libraries 1
  - Verilog Libraries 1
- Actel
  - FTP Site 23
  - Web Based Technical Support 22
  - Web Site 22
- Actel Manuals vii
- Assumptions v

B

- Back-Annotating Delays 13
- Behavioral Simulation 5, 10, 17

C

- Capturing a Design 4
- Command File 10
- Compiling Verilog Libraries 2
- Contacting Actel
  - Customer Service 21
  - Electronic Mail 23
  - Technical Support 22
  - Toll-Free 21
  - Web Based Technical Support 22
- Conventions vi
  - Document vi–??
- Creating
  - Command File 10
  - SDF File 13–20
  - Stimulus File 9
- Creating Testbench 9
- Customer Service 21

D

- Delays 13
- Design Creation/Verification 4
  - Behavioral Simulation 5
  - Design Capture 4
  - EDIF Netlist Generation 5
  - Structural Netlist Generation 5
  - Structural Simulation 5
  - Synthesis 5
- Design Flow 4
  - Programming 6
  - System Verification 6
- Design Implementation 6
  - Place-and-Route 6
  - Timing Simulation 6
- Design Layout 6
- Design Synthesis 5
- Designer
  - EDIF Option 7
  - Extracting Timing Information 13, 20
  - GENERIC Option 7
  - Place-and-Route 6
  - Software Installation Directory v
  - Structural Netlist Generation 7
  - Verilog Option 6, 7
- Device
  - Programming 6
  - Verification 6
- Document
  - Assumptions v
  - Organization v
- Document Conventions vi, vi–??

E

- EDIF Netlist Generation 5, 7
- EDIF Option 7
- edn2vlog 8
- Electronic Mail 23

Example Testbench on page 11 10
Extracting Timing Information 13–20

F
File
  Command 10
  SDF 13
  Stimulus 9
Functional Simulation 10, 17

G
Gate-Level Netlist 5
Generating
  EDIF Netlist 5, 7
  Gate-Level Netlist 5
  Structural Netlist 5, 7
GENERIC Option 7

I
Installation Directory
  Designer v
  Migration Libraries 1
  Verilog Libraries 1

L
Libraries
  Compiling 2
  Migration 1
  Verilog 1

M
Migration Library Installation Directory 1
Minus Switches 15

N
Netlist Generation
  EDIF 5
  Gate-Level 5
  Structural 5

O
Online Help ix

P
Place-and-Route 6
Plus Switches 16
Product Support 21–24
  Customer Applications Center 22
  Customer Service 21
  Electronic Mail 23
  FTP Site 23
  Technical Support 22
  Toll-Free Line 21
  Web Site 22
Programming a Device 6

R
Required Software 1

S
Schematic Capture 4
SDF file 13–20
sdf_annotate 13
Setup Procedures
  User Setup 2
Simulation
  Behavioral 5, 10, 17
  Functional 10, 17
  Model Technology 17–20
  SilosIII 9–15

- Structural 5, 11, 18
- Synthesis-Based 5
- Timing 6, 12, 19
- VCS 9–15
- Verilog-XL 9–15

Software Requirements 1
Stimulus File 9
Structural Netlist Generation 5
- Designer 7
- EDIF Option 7
- edn2vlog 8
- GENERIC Option 7

Structural Simulation 11
- Model Technology V-System Simulator 18
- MTI 18
- Verilog 5

Switches 15–16
- Minus 15
- Plus 16

Synthesis 5
System Requirements 1
System Verification 6
- Silicon Explorer 6

T

Technical Support 22
Testbench 9
Timescale Definition 10, 11
Timing Information 13–20
- SDF File 13–20

Timing Simulation 6, 12
- MTI 19

Toll-Free Line 21

U

Unit Delays 5
User Setup 2

V

Verilog 4
- Back-Annotating Delays 13
- Behavioral Simulation 5
- Libraries 1
- Option 7
- Option in Designer 6
- Structural Simulation 5
- Switches 15–16
- Timing Simulation 6

Verilog Design Flow
- Design Implementation 6
- Programming 6

Verilog HDL Source Entry 4
Verilog Libraries
- Compiling 2

Verilog Library Installation Directory 1

W

Web Based Technical Support 22