

Verilog®

Simulation Guide



Windows® and UNIX® Environments

Actel Corporation, Sunnyvale, CA 94086

© 1998 Actel Corporation. All rights reserved.

Printed in the United States of America

Part Number: 5579005-1

Release: July 1998

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.

Microsoft, Windows, and Windows NT are trademarks or registered trademark of Microsoft Corporation.

Synopsys is a registered trademark of Synopsys, Inc.

Verilog is a registered trademark of Open Verilog International.

Viewlogic and Workview are registered trademarks and of Viewlogic 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	v
	Document Conventions	vi
	Designer Series Manuals	vi
1	Setup	1
	Software Requirements	1
	UNIX System Setup	1
	Verilog Libraries	2
	Compiling Verilog Libraries	2
2	Design Flow	5
	Actel-Verilog Design Flow Illustrated	5
	Actel-Verilog Design Flow Overview	6
3	Generating Netlists	9
	Generating an EDIF Netlist	9
	Generating a Structural Verilog Netlist	9
4	Interpreted Simulation	11
	Creating a Test Bench	11
	Creating a Command File	12
	Behavioral Simulation	12
	Structural Simulation	13
	Timing Simulation	14
	Verilog Switches	17
5	Simulation with MTI V-System or ModelSim	19
	Behavioral Simulation	19
	Structural Simulation	20
	Timing simulation	21

6	Simulation with Mentor Graphics QuickHDL	23
	Behavioral Simulation	23
	Structural Simulation.	24
	Timing simulation	25
A	Product Support	27
	Actel U.S. Toll-Free Line	27
	Customer Service	27
	Customer Applications Center	28
	Guru Automated Technical Support	28
	Action Facts.	28
	Web Site	29
	HLD Web Site.	29
	FTP Site.	29
	Electronic Mail	29
	Worldwide Sales Offices	30
	Index	31

Introduction

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

Document Organization

The *Verilog Simulation Guide* is divided into the following chapters:

Chapter 1 - Setup contains information and procedures 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 and procedures for generating EDIF and structural Verilog netlists.

Chapter 4 - Simulation contains information and procedures about performing simulation on Actel designs with Verilog simulators.

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

Document Assumptions

The information in this manual is based on the following assumptions:

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

The following conventions are used throughout this manual.

Information that is meant to be input by the user is formatted as follows:

keyboard input

The contents of a file is formatted as follows:

file contents

Messages that are displayed on the screen appear as follows:

Screen Message

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, 1200xl, act3, 3200dx, 40mx, 42mx, and 54sx.

Designer Series Manuals

The Designer Series software includes printed and on-line manuals. The on-line manuals are in PDF format on the CD-ROM in the “/doc” directory. These manuals are also installed onto your system when you install the Designer software. To view the on-line manuals, you must have Adobe® Acrobat Reader® installed. Actel provides Reader on the Designer Series CD-ROM.

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

Designing with Actel. This manual describes the design flow and user interface for the Actel Designer Series software, including information about using the ACTgen Macro Builder and ACTmap VHDL Synthesis 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.

ACTmap VHDL Synthesis Methodology Guide. This guide contains information, optimization techniques, and procedures to assist designers in the design of Actel devices using ACTmap VHDL.

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

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

MOTIVE™ Static Timing Analysis Interface Guide. This guide contains information and procedures to assist designers in the use of the MOTIVE software to perform static timing analysis on Actel designs.

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

Viewlogic® Powerview Interface Guide. This guide contains information and procedures to assist designers in the design of Actel devices using Powerview CAE software and the Designer Series software.

Viewlogic Workview® Office Interface Guide. This guide contains information and procedures to assist designers in the design of Actel devices using Workview Office CAE software and the Designer Series software.

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

Verilog Simulation Guide. This guide contains information and procedures 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.

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

Designer Series Development System Conversion Guide UNIX Environments. This guide describes how to convert designs created in Designer Series versions 3.0 and 3.1 for UNIX to be compatible with later versions of Designer Series.

Designer Series Development System Conversion Guide Windows Environments. This guide describes how to convert designs created in Designer Series versions 3.0 and 3.1 for Windows to be compatible with later versions of Designer Series.

FPGA Data Book and Design Guide. This guide contains detailed specifications on Actel device families. Information such as propagation delays, device package pinout, derating factors, and power calculations are found in this guide.

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

In addition to the Designer Series Manuals, the *Digital Design and Synthesis with Verilog HDL*¹ book contains practical examples and techniques to use Verilog HDL for logic design and synthesis.

1. Madhavan, Rajeev, Sternbeim, Eli, Singh, Rajvir, and Trivedi, Yatin. *Automata Publishing Company*, 1993.

Setup

This chapter contains information about setting up Verilog libraries for use in simulating Actel designs and the directory structure of the libraries. 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 R2-1998 or later and IEEE-1364 compliant Verilog simulators. Additionally, this guide contains information about using the PC and UNIX MTI V-System and ModelSim simulators, the UNIX Mentor Graphics QuickHDL simulator, the UNIX Cadence Verilog-XL simulator, the PC and UNIX Simucad SilosIII simulator, and the UNIX Synopsys VCS simulator.

For specific information about which versions are supported with this release, 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

UNIX System Setup

After installing Designer, make sure the proper environment variables are set in your UNIX shell script. The following are C shell variables. If you are using another shell, adjust the syntax accordingly.

```
setenv ALSDIR <actel_install_directory>
set path=($ALSDIR/bin $path)
```

If you are using SunOS or Solaris, the following variable must also be set:

```
setenv LD_LIBRARY_PATH $ALSDIR/lib
```

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

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

Refer to the *Designing with Actel* manual and the documentation included with your simulation tool for additional information about setting environment variables.

Verilog Libraries

The Actel Verilog libraries contain Verilog models for each Actel macro in all Actel families. The Verilog libraries are installed in the “<actel_install_directory>/lib/vlog/<act_fam>” directories.

Migration Libraries

In addition to the standard Actel libraries, Actel provides a set of migration libraries. These libraries contain macros that were supported in earlier versions of the Designer Series software and macros that may be 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.

Compiling Verilog Libraries

Before simulating a design with the MTI V-System, MTI ModelSim, or the Mentor Graphics QuickHDL 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.

MTI ModelSim and V-System Simulators

Use the following procedure to compile Verilog libraries for the MTI ModelSim and V-System simulators. UNIX commands are typed at the UNIX prompt. PC commands are typed on the command line of the MTI Transcript window. The commands shown are for PC. To make the commands work for UNIX, use forward slashes instead of back slashes

1. **Create a directory called “mti” in the “\$ALSDIR\lib\vlog” directory.**
2. **Invoke the simulator (PC only).**
3. **Change to the “\$ALSDIR\lib\vlog\mti” directory.** Type the following command:

```
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 following command:

```
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
```

Mentor Graphics QuickHDL Simulator

Use the following procedure to compile Verilog libraries for the Mentor Graphics QuickHDL simulator.

1. **Create a directory called “qhdl” in the “\$ALSDIR/lib/vlog” directory.**
2. **Change to the “\$ALSDIR/lib/vlog/qhdl” directory.** Type the following command:

```
cd $ALSDIR/lib/vlog/qhdl
```

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

```
qhlib <act_fam>
```

4. **Compile the Actel family models.** Type the following command:

```
qvlcom -work <act_fam> $ALSDIR/lib/vlog/<act_fam>/*.v
```

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

```
qvlcom -work <act_fam> $ALSDIR/lib/vlog/<act_fam>_mig/*.v
```

Design Flow

This chapter illustrates and 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 software¹.

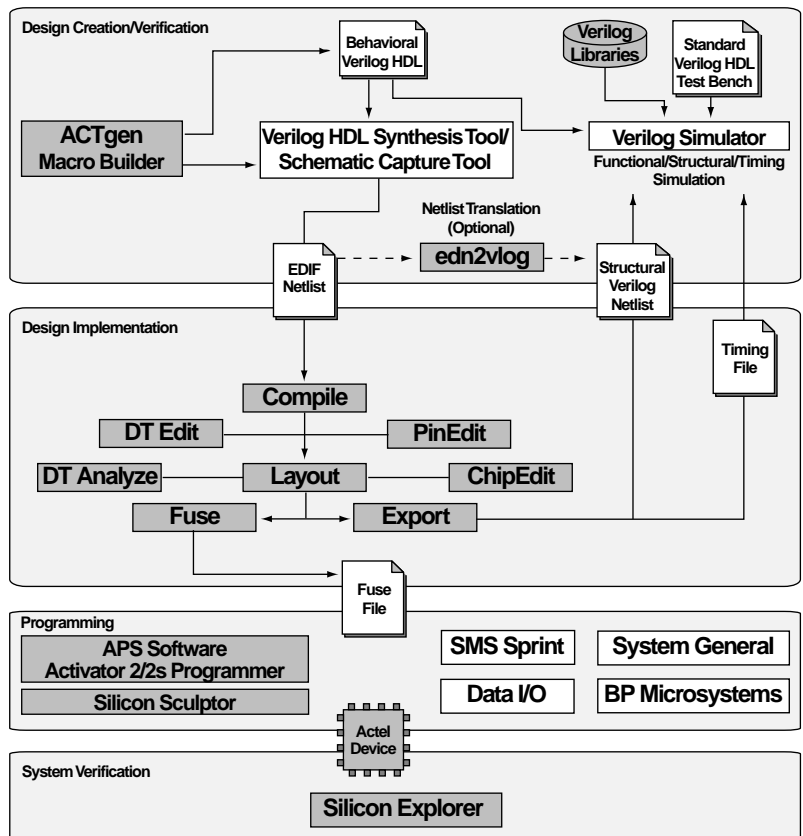


Figure 2-1. Actel-Verilog Design Flow

¹ Actel-specific utilities/tools are denoted by the grey boxes in Figure 2-1.

Actel-Verilog Design Flow Overview

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

If your design is a schematic, an EDIF netlist is generated for use in Designer and a structural Verilog netlist is generated 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. A standard Verilog HDL test bench can be used to drive simulation. Refer to “Behavioral Simulation” on page 12 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, the EDIF netlist is also used to generate a structural Verilog netlist. Refer to “Generating an EDIF Netlist” on page 9 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 9 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. Default unit delays included in the Verilog libraries are used for every gate. Refer to “Structural Simulation” on page 13 and the documentation included with your simulation tool for information about performing structural simulation.

Design Implementation

During design implementation, a design is placed and routed using Designer. Additionally, static timing analysis can be performed on a design in Designer with the DT Analyze tool. After place and route, post-layout (timing) simulation is performed 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 *Designing with Actel* manual for information about using Designer.

Static Timing Analysis

Use the DT Analyze tool in Designer to perform static timing analysis on your design. Refer to the *Designing with Actel* manual for information about using DT Analyze.

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 14 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 3rd party programming system. Refer to the *Designing with Actel* manual 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.

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. The EDIF netlist is used 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. The structural Verilog netlist is used 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. The Export dialog box is displayed. Specify Netlist File as the File Type and Verilog as the Format. Click OK.

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.

Interpreted Simulation

This chapter 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. Cadence Verilog-XL, Simucad SilosIII, and Synopsys VCS are simulators in this category.

Also included in this chapter is information about creating a test bench and information about creating a command file to run a simulation in batch mode. Finally, some common Verilog simulation switches are described. Refer to the documentation included with your simulation tool for additional information about test benches, command files, switches, and performing simulation.

Creating a Test Bench

You can use a test bench 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 pre-defined command, such as “\$readmemb,” “\$monitor,” and “\$display.” The test bench must be in the current project directory to be used. The following is an example test bench:

```
`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
```

Creating a 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 `<verilog_netlist>.v` variable is the Verilog top level design that includes all design sub-levels. 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 17 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 test bench.** Make sure your test bench has a timescale definition added to it. The following is an example timescale definition:

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

Refer to “Creating a Test Bench” on page 11 for information about creating test benches.

2. **Create or modify the command file.** A command file is only necessary if you are running batch simulation. Refer to “Creating a Command File” on page 12 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 -y
$ALSDIR/lib/vlog/<act_fam> +libext+.v
```

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

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

The “-y” and “+libext+” options are Verilog switches that you can add to your command line during simulation. Refer to “Verilog Switches” on page 17 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 test bench.** Make sure your test bench has a timescale definition added to it. The following is an example timescale definition:

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

Refer to “Creating a Test Bench” on page 11 for information about creating test benches.

2. **Create or modify the command file.** A command file is only necessary if you are running batch simulation. Refer to “Creating a Command File” on page 12 for information.
3. **Simulate the design.** If your design is a Verilog HDL design, make sure that you simulate your 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 -y $ALSDIR/lib/vlog/<act_fam> +libext+.v
```

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

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

The “-y” and “+libext+” options are Verilog switches that you can add to your command line during simulation. Refer to “Verilog Switches” on page 17 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 *Designing with Actel* manual for information about placing and routing a design using Designer.
- 2. 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 <design_name>.sdf file by choosing the SDF or SDF + Pin option from the CAE pull-down menu. Click OK.

3. **Create or modify a test bench.** Make sure your test bench has an “sdf_annotate” construct in it. The following is an example test bench 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 “Creating a Test Bench” on page 11 for information about creating test benches.

4. **Create or modify the command file.** A command file is only necessary if you are running batch simulation. Refer to “Creating a Command File” on page 12 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 test bench 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 your structural Verilog HDL netlist that you

generated using Designer or the “edn2vlog” program. Invoke the Verilog simulator by typing the following command:

VCS

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

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

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

Other Simulators

```
<verilog_executable> <test_bench>.v <design_name>.v -y $ALS-  
DIR/lib/vlog/<act_fam> +libext+.v
```

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

```
<verilog_executable> <test_bench>.v <design_name>.v -y $ALS-  
DIR/lib/vlog/<act_fam> -y $ALSDIR/lib/vlog/<act_fam>_mig  
+libext+.v
```

The “-y” and “+libext+” options are Verilog switches that you can add to your command line during simulation. Refer to “Verilog Switches” on page 17 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 be simulated in accelerated mode (Verilog-XL only).
-c	Compile only option; compiles the text code in 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.

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.

Simulation with MTI V-System or ModelSim

This chapter describes the procedures for performing functional (behavioral and structural) and timing simulation on an Actel design using commands for the Model Technology V-System or 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 MTI V-System or ModelSim simulator. UNIX commands are typed at the UNIX prompt. PC commands are typed on the command line of the MTI Transcript window. The commands shown 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 test bench. Type the following command:

```
cd <project_dir>
```

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

```
vlib work
```

4. **Compile your design source and test bench file(s).** Before simulating your design, you must compile the source files and test bench. 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 test bench will be simulated.

If any Actel macros are instantiated in your Verilog source, use the following command to simulate your design with 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 MTI V-System or ModelSim simulator. UNIX commands are typed at the UNIX prompt. PC commands are typed on the command line of the MTI Transcript window. The commands shown 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 test bench. 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 test bench.** If you have not already generated a structural Verilog netlist, go to “Generating a Structural Verilog Netlist” on page 9 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 test bench 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 MTI V-System or ModelSim simulator. UNIX commands are typed at the UNIX prompt. PC commands are typed on the command line of the MTI Transcript window. The commands shown 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 *Designing with Actel* manual 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. Invoke the simulator (PC only).**
- 4. Change directory to your project directory.** This directory must include your Verilog design files and test bench. 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 test bench.** If you have not already generated a structural Verilog netlist, go to “Generating a Structural Verilog Netlist” on page 9 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=addder.sdf -c  
test_adder_structural
```

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

Simulation with Mentor Graphics QuickHDL

This chapter describes the procedures for performing functional (behavioral and structural) and timing simulation on an Actel design using commands for the Mentor Graphics QuickHDL 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 Mentor Graphics QuickHDL simulator.

1. **Change directory to your project directory.** This directory must include your Verilog design files and test bench. Type the following command:

```
cd <project_dir>
```

2. **Create a “work” directory.** Type the following command:

```
qhlib work
```

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

```
qvlcom <behavioral_design_file>.v  
qvlcom <test_bench>.v
```

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

```
qhsim <topmost_module_name>
```

For example:

```
qhsim test_adder_behave
```

The module test_adder_behave in the test bench will be simulated.

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

```
qhsim -L $ALSDIR/lib/vlog/qhdl/<act_fam>  
<topmost_module_name>
```

Structural Simulation

Use the following procedure to perform a structural simulation of a design using the Mentor Graphics QuickHDL simulator.

1. **Change directory to your project directory.** This directory must include your Verilog design files and test bench. Type the following command:

```
cd <project_dir>
```

2. **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:

```
qhlib work
```

3. **Compile the structural netlist and test bench.** If you have not already generated a structural Verilog netlist, go to “Generating a Structural Verilog Netlist” on page 9 for the procedure. Type the following commands:

```
qvlcom <structural_netlist>.v  
qvlcom <test_bench>.v
```

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

```
qhsim -L $ALSDIR/lib/vlog/qhdl/<act_fam>  
<topmost_module_name>
```

For example:

```
qhsim -L $ALSDIR/lib/vlog/qhdl/42mx test_adder_structure
```

The module test_adder_structure in the test bench 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 Mentor Graphics QuickHDL simulator.

1. **Place and route your design in Designer.** Refer to the *Designing with Actel* manual 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. **Change directory to your project directory.** This directory must include your Verilog design files and test bench. Type the following command:

```
cd <project_dir>
```

4. **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:

```
qhlib work
```

5. **Compile the structural netlist and test bench.** If you have not already generated a structural Verilog netlist, go to “Generating a Structural Verilog Netlist” on page 9 for the procedure. Type the following commands:

```
qvlcom <structural_netlist>.v  
qvlcom <test_bench>.v
```

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

```
qhsim -L $ALSDIR\lib\vlog\qhdl\<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 test bench that you wish to back annotate. For example:

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

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

Product Support

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

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 non-technical 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) 522-4474.

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

From Canada, call (408) 522-4480.

From Europe, call (408) 522-4474 or +44 (0) 1256 305600.

From Japan, call (408) 522-4252.

From the rest of the world, call (408) 522-4252.

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

Customer Applications Center

The Customer Applications Center is staffed by applications engineers who can answer your hardware, software, and design questions.

All calls are answered by our Technical Message Center. 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 p.m., Pacific Standard Time, Monday through Friday.

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

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

Guru Automated Technical Support

Guru is a Web based automated technical support system accessible through the Actel home page (<http://www.actel.com>). 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.

Action Facts

Action Facts is a 24-hour fax-back service that allows you to have technical literature faxed directly to you. The literature includes current applications notes, device diagrams, radiation testing reports, CAE-related documents, training release schedules, and much more.

To receive your free copy of the catalog or any of our literature, contact us at (800) 262-1062 or (408) 739-1588.

Web Site

Actel has a World Wide Web home page where you can browse a variety of technical and non-technical 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 we have provided on the net.

Be sure to visit the "Actel User Area" on our Web site, which contains information regarding: products, technical services, current manuals, and release notes.

HLD Web Site

Actel also has an HLD web site at <http://www.actel.com/hld/> that you can browse for HLD design information. The site provides detailed information and examples on coding styles and synthesis and simulation flow, as well as links to our synthesis and simulation partners.

FTP Site

Actel has an anonymous FTP site located at <ftp://ftp.actel.com>. You can directly obtain library updates, software patches, design files, and data sheets.

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. The e-mail account is monitored several times per day.

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

Worldwide Sales Offices

Actel Corporation
955 East Arques Avenue
Sunnyvale, California 94086
USA
Toll Free: 888.99.ACTEL
Tel: 408.739.1010
Fax: 408.739.1540

California (North)
Tel: 408.328.2200
Fax: 408.328.2358

California (South)
Tel: 714.727.0470
Fax: 714.727.0476

Georgia
Tel: 770.844.4495
Fax: 770.844.1395

Illinois
Tel: 847.259.1501
Fax: 847.259.1572

Massachusetts
Tel: 508.635.0010
Fax: 508.635.9510

Texas
Tel: 972.235.8944
Fax: 972.235.9659

Actel Europe, Ltd.
Daneshill House, Lutyens Close
Basingstoke,
Hampshire RG24 8AG
United Kingdom
Tel: +44 (0)1256.305600
Fax: +44 (0)1256.355420

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

Germany
Tel: +49 (0)8165.9584.0
Fax: +49 (0)8165.9584.10

Italy
Tel: +39 (0)2.38.09.32.59
Fax: +39 (0)2.38.09.32.60

Actel Asia-Pacific
EXOS Ebisu Building 4F
1-24-14 Ebisu Shibuya-ku
Tokyo 150 Japan
Tel: +81 (0)3.3445.7671
Fax: +81 (0)3.3445.7668

Korea
Tel: +82 (0)2.555.7425
Fax: +82 (0)2.555.5779

Taipei, Taiwan R.O.C.
Tel: +886 (0)2.698.2448
ext. 104
Fax: +886 (0)2.698.2548

Index

<act_fam> variable vi

A

Accessing

 Migration Libraries 2

 Verilog Libraries 2

Actel

 Device Families vi

 FTP Site 29

 Web Based Technical Support 28

 Web Site 29

Action Facts 28

Assumptions v

B

Back Annotating Delays 15

Behavioral Simulation 6, 12, 19, 23

C

Capturing a Design 6

Command File 12

Compiling Verilog Libraries 2

Contacting Actel

 Customer Service 27

 Electronic Mail 29

 Technical Support 28

 Toll-Free 27

 Web Based Technical Support 28

Conventions vi

 <act_fam> variable vi

Creating

 Command File 12

 SDF File 14, 21, 25

 Stimulus File 11

Creating Test Bench 11

Customer Service 27

D

Delays 15

Design Creation/Verification 6

 Behavioral Simulation 6

 Design Capture 6

 EDIF Netlist Generation 7

 Structural Netlist Generation 7

 Structural Simulation 7

 Synthesis 7

Design Flow 6–8

 Design Creation/Verification 6

 Design Implementation 7

 Programming 8

 System Verification 8

Design Implementation 7

 Place and Route 8

 Timing Analysis 8

 Timing Simulation 8

Design Layout 8

Design Synthesis 7

Designer

 DT Analyze Tool 8

 EDIF Option 9

 Extracting Timing Information 14, 21, 25

 GENERIC Option 9

 Manuals vi

 Place and Route 8

 SDF Option 21, 25

 Software Installation Directory v, 1

 Structural Netlist Generation 9

 Timing Analysis 8

 Verilog Option 8, 9

Device

 Debugging 8

 Families vi

 Programming 8

Document

- Assumptions v
- Conventions vi
- Organization v

DT Analyze Tool 8

E

EDIF Netlist Generation 7, 9

EDIF Option 9

edn2vlog 10

Electronic Mail 29

Extracting Timing Information 14, 21, 25

F

File

- Command 12
- SDF 15
- Stimulus 11

Functional Simulation 12, 19, 23

G

Gate-Level Netlist 7

Generating

- EDIF Netlist 7, 9
- Gate-Level Netlist 7
- Structural Netlist 7, 9

GENERIC Option 9

H

HLD Web Site 29

I

Installation Directory

- Designer v, 1
- Migration Libraries 2
- Verilog Libraries 2

L

Libraries

- Compiling 2
- Migration 2
- Verilog 2

M

Migration Library Installation Directory 2

Minus Switches 17

N

Netlist Generation

- EDIF 7
- Gate-Level 7
- Structural 7

O

On-Line Documentation vi

P

Place and Route 8

Plus Switches 18

Product Support 27–29

- Action Facts 28
- Customer Applications Center 28
- Customer Service 27
- Electronic Mail 29
- FTP Site 29
- HLD Web Site 29
- Technical Support 28
- Toll-Free Line 27
- Web Site 29

Programming a Device 8

R

Related Manuals vi

Required Software 1

S

Schematic Capture 6

SDF File 21, 25

SDF file 14

SDF Option 21, 25

sdf_annotate 15

Setting Environment Variables 1

Setup Procedures

 Setting Environment Variables 1

 System Setup 1

 User Setup 2

Simulation

 Behavioral 6, 12, 19, 23

 Functional 12, 19, 23

 Mentor Graphics 23–26

 Model Technology 19–22

 SilosIII 11–16

 Structural 7, 13, 20, 24

 Synthesis-Based 6, 7

 Timing 8, 14, 21, 25

 VCS 11–16

 Verilog-XL 11–16

Software Requirements 1

Static Timing Analysis 8

Stimulus File 11

Structural Netlist Generation 7

 Designer 9

 EDIF Option 9

 edn2vlog 10

 GENERIC Option 9

Structural Simulation 13, 24

 Mentor Graphics 24

 Model Technology V-System Simulator 20

 MTI 20

Verilog 7

Switches 17–18

 Minus 17

 Plus 18

Synthesis 7

System Requirements 1

System Setup 1

System Verification 8

 Silicon Explorer 8

T

Technical Support 28

Test Bench 11

Timescale Definition 12, 13

Timing Analysis 8

Timing Information 14, 21, 25

 SDF File 14, 21, 25

Timing Simulation 8, 14, 25

 MTI 21

 SDF Option 21, 25

Toll-Free Line 27

U

Unit Delays 6

User Setup 2

V

variable, <act_fam> vi

Verilog

 Back Annotating Delays 15

 Behavioral Simulation 6

 Libraries 2

 Option 9

 Option in Designer 8

 Structural Simulation 7

 Switches 17–18

- Timing Simulation 8
- Verilog Design Flow
 - Design Creation/Verification 6
 - Design Implementation 7
 - Programming 8
 - System Verification 8
- Verilog HDL Source Entry 6
- Verilog Libraries
 - Compiling 2
- Verilog Library Installation Directory 2

W

- Web Based Technical Support 28